DesignSpark Mechanical

Guidebook

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Chapter 1 – Introduction and Installation and the User Interface of DesignSpark Mechanical

1-1 Introduction of DesignSpark Mechanical

DesignSpark Mechanical (DSM in short) is free 3D CAD graphics software launched by RS Components, providing a powerful and intuitive way to draw. DSM allows all engineers and designers to draw 3D models easily without a strong CAD background (better if you have). In addition, it also provides a rich graphic library, and online quotes purchase service, and can greatly reduce development time.

1-2 DesignSpark Mechanical Installation

DesignSpark Mechanical is a free 3D modelling software that you can download from its official website (https://www.rs-online.com/designspark/home), click on the software icon on the menu on the front page after entering the official website.

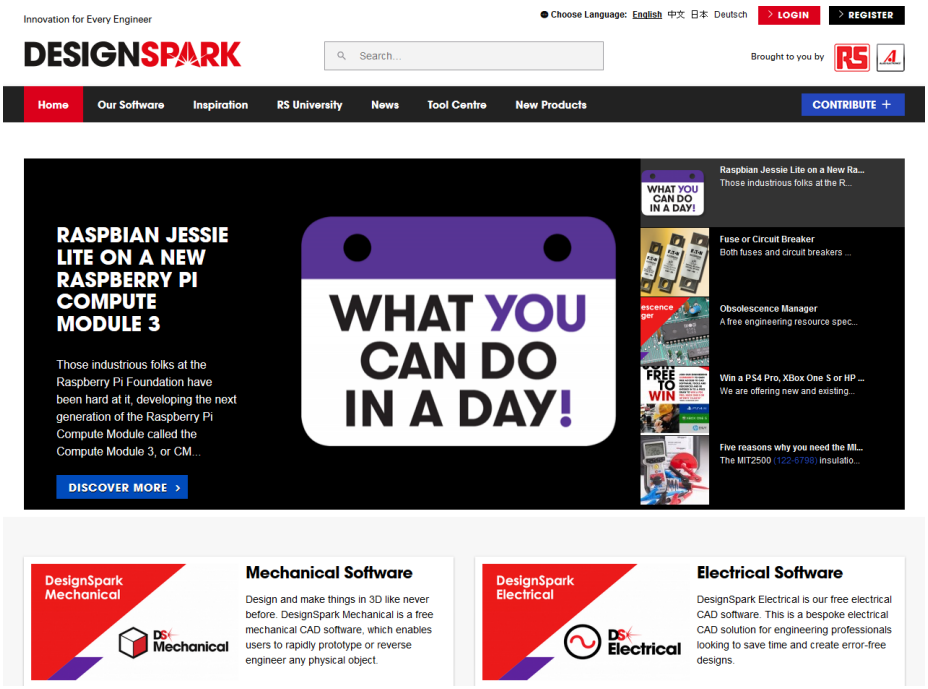

Fig. 1-1 Official website of DesignSpark

After entering the download page, make sure your computer meets the minimum hardware requirements, and then download the program according to the operating system version (32-bit or 64-bit), please unzip the downloaded file.

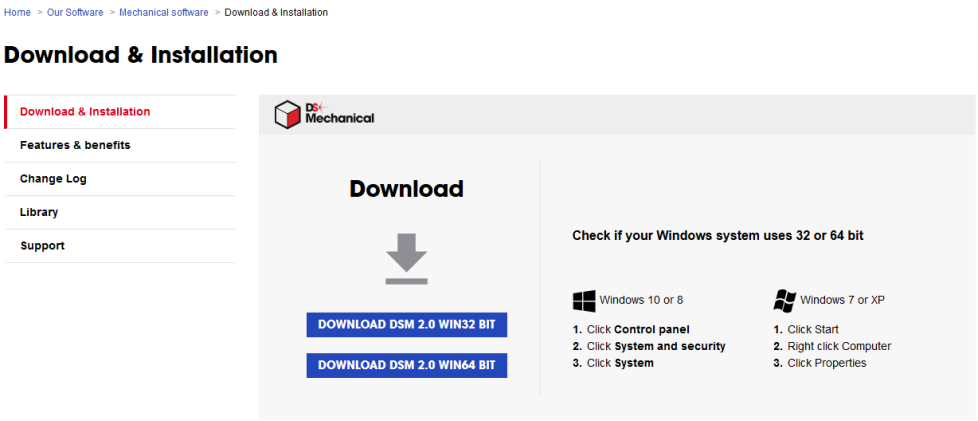

Fig. 1-2 View hardware requirements and download

Open the unzipped folder, and click the installer to start the installation, select the installation path and agree to the license.

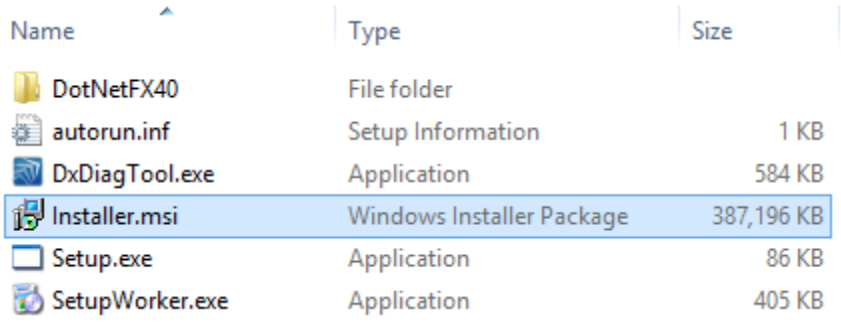

Fig. 1-3 Starts to install

Follow the installation steps and agree to the license to install DesignSpark Mechanical.

Fig. 1-4 Agree the license

After the installation is complete, please open DesignSpark Mechanical on your computer. You can log in directly if you are already a DesignSpark user (user account is

the same as DesignSpark PCB's). If you are without an account, click the "Create a new DesignSpark.com account" and refer to the first chapter of this book to register a new account.

Fig. 1-5 Open DesignSpark shortcut icon

When the username and password are correct, the initial screen as Fig. 1-6 will pop up, which means DesignSpark Mechanical is opening.

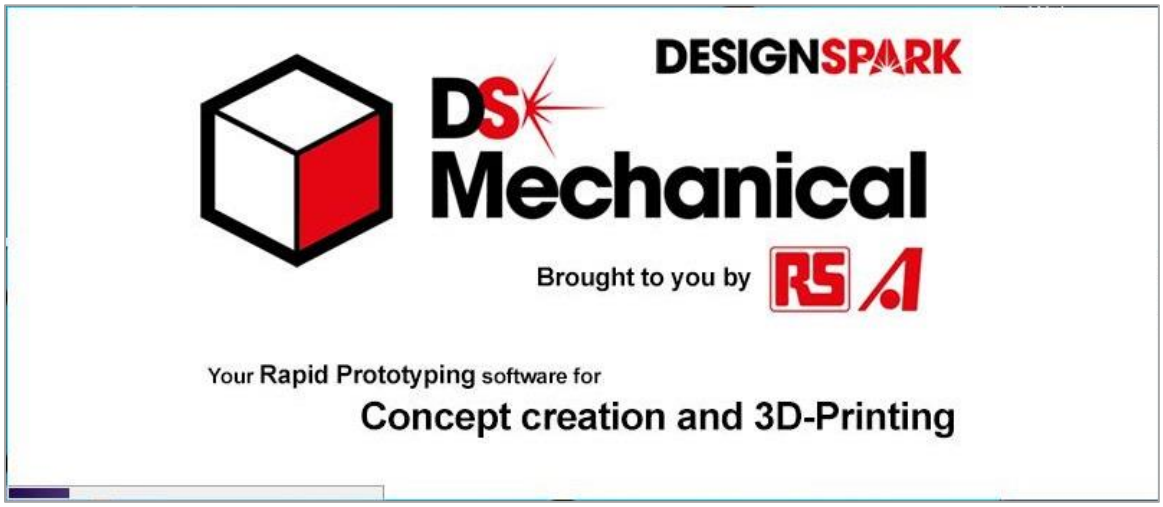

Fig. 1-6 Initial screen of the program

1-3 Introduction of the Design Environment

The main screen, as shown in Fig. 1-7, will appear after DesignSpark Mechanical has opened, the screen design of the program is similar to general word processing software.

At the top of the screen is the toolbar for drawing tools, viewing angle, etc. On the left of the screen is the setting area, you can set drawing elements, layers and properties and so on. And the last part is the drawing area on the right of the screen. You can change the viewing angle by mouse and toolbar settings, and plotting with the assistance of auxiliary grid.

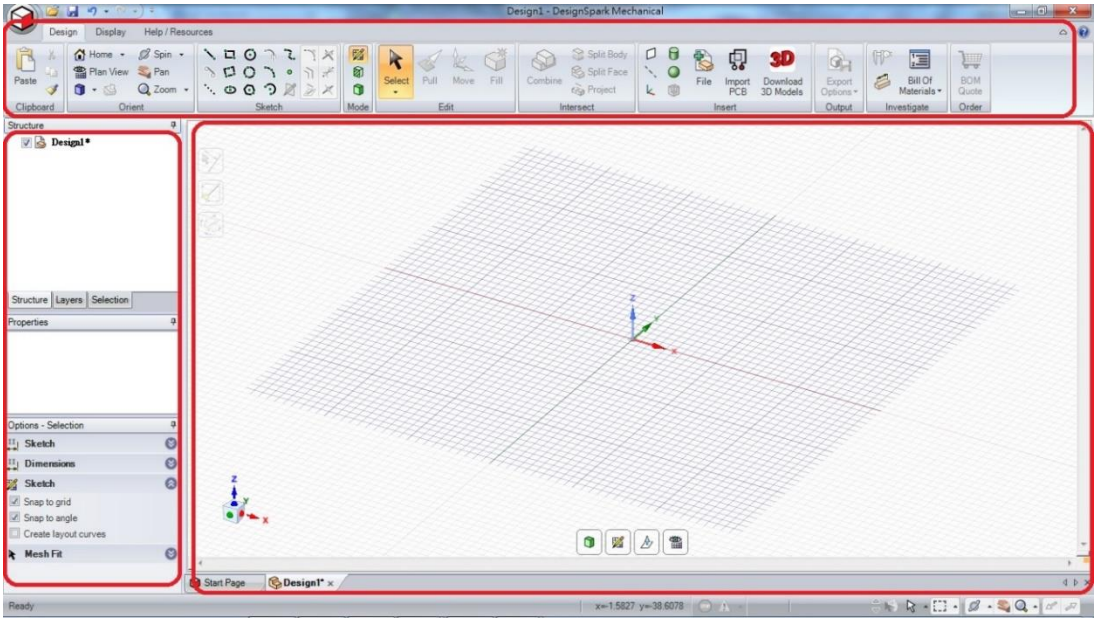

Fig.1-7 Main screen of DesignSpark Mechanical

There are four primary tabs on DSM toolbar, which are design, display, detail, and Help / resources. When starting the program, "design" tool will be turned on by default. The design tools are further divided into more subtabs according to different functions.

Fig. 1-8 Toolbar

1-3-1 Design Tools

1. Clipboard: You can cut, copy, paste and do other activities.

2. Orient: By selecting different viewing angles, such as main view, front view, left, right, up, down view to view the material. You can set a best drawing view angle by rotation, translation and scaling.

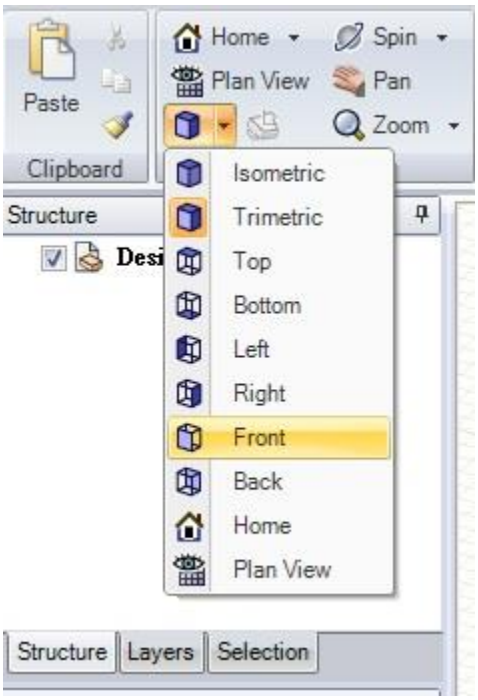

Fig. 1-9 Locating function

3. Sketch: Sketch is divided into two blocks, in the left block, you can choose different lines, such as lines, curves, and other geometric drawing and modeling; the right block has modified line, such as fillet, parallel copy and so on.

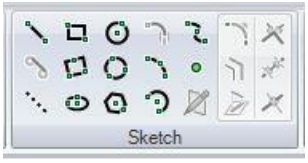

Fig. 1-10 Sketch

4. Mode: Different ways to view a 3D model. You can select sketches, sections, and 3D.

Fig. 1-11 Mode

5. Edit: Through select, pull, move, and other ways to fill the model:

- Select: use the left mouse button to select point, line and surface. While using other functions, Select is still valid, you can select the mode to select objects in the options below. In addition, the effects of left-click on "Select" will vary depends on the number of clicks is 1, 2, or3. Left-click once to select the line is clicked, twice will select an edge loop (repeat this step will cycle through all available edge loops), 3 clicks will select the whole entity (but the entity information will not be collected while the entity is selected).

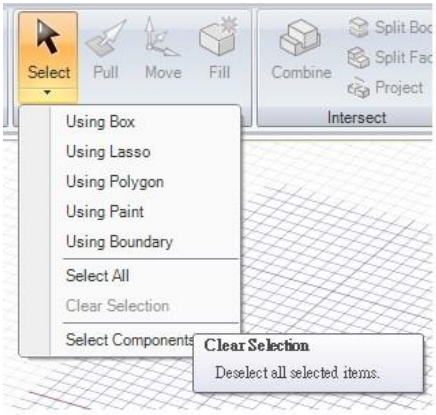

Fig. 1-12 Select

- Pull: The pull tool can be used in several ways. They are used to pull 2D sketches into 3D graphics. When forming a three-dimensional object the pull tool can extend or reduce the object, chamfer, or beveled edges by pulling a surface. And by pulling the sketch, you can add new objects (see Chapter 3) on one surface.

- Move: use the move tool to handle different linear and radial movements, this depends on three factors: 1. Which part of the model is chosen; 2. Where to put the mobile control point; 3. Where to put the mobile control axis. When selecting the moving function, “Select Components” can be found at the top left of the design window, use this tool to select an object to ensure that the object can be retained after copying.

- Fill: Advanced tool can delete objects.

6. Intersect: Split or combine one or more objects.

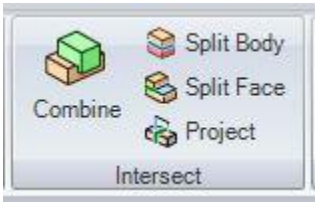

Fig. 1-13 Intersection

7. Insert: This toolbar provides the function to import external model files. You can

find ready-to-use graphics and online models and use them after importing.

Fig. 1-14 Insert

1-3-2 Display Tools

These tools contain file output, dimensions, bill of materials and quotes and other functions, you can mark detailed dimensions of all objects we will show you in later chapters. Also it can export parts list, and offer online purchase orders.

1. Output: Export your object to other formats, including QuickPart 3D Service (an online 3D service provider, https://eu.quickparts.com/), .pdf, .stl and .dxf format.

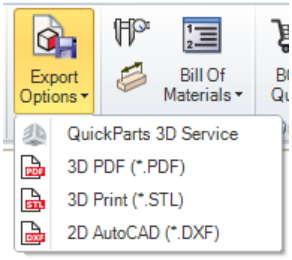

Fig. 1-15 Output

2. Investigate: Retrieve details of your design.

- Measure: Click edge, surface or any part of your object, related measurement parameters will show up.

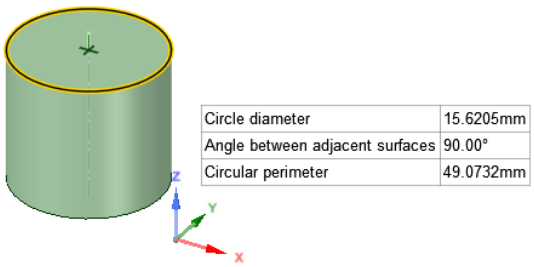

Fig. 1-16a Measure

- Dimension: Click and edge or face to preview possible dimensions, as Fig. 1-16b.

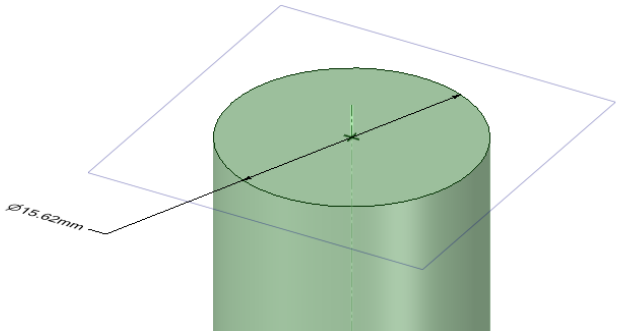

Fig. 1-16b Dimension

- Bill Of Materials: generate Bill Of Materials of your design, as Fig 1-16c.

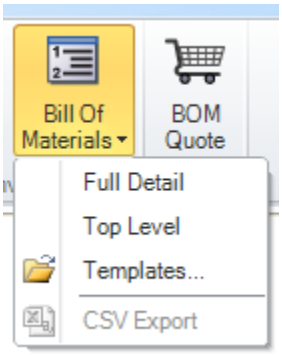

Fig. 1-16c Bill of materials

3. Order: You can upload BOM to quote and order necessary components/parts from RS or any other retailer in your country, as per the description in Fig. 1-17.

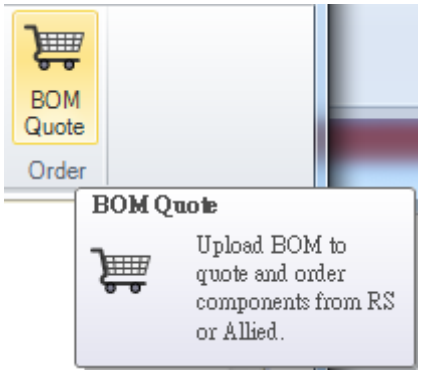

Fig. 1-17 Order

1-4 Summary

DesignSpark Mechanical is easy-to-use and resource-rich graphics software, with its intuitive and simple drawing method, online graphic library, and offer quoting and shipping, you can significantly reduce development time. This chapter briefly introduced

the various main functions, in the next chapter we will teach you how to use these tools to create various 3D designs.

Chapter 2 – Basic Modeling

2-1 Cube

DesignSpark Mechanical provides an intuitive drawing method, so you only need to draw a quadrilateral, and then pull the plane out to the establishment of the body to make a cube.

Step 1: Select the design goals in the toolbar.

Step 2: Select the front view of the locating tool.

Step 3: Select rectangle in the sketch tool.

Fig. 2-1 Select rectangle

Step 4: Click the left mouse key on anywhere in the drawing area of the sketch grid, and

pull, a rectangle will appear in the drawing area.

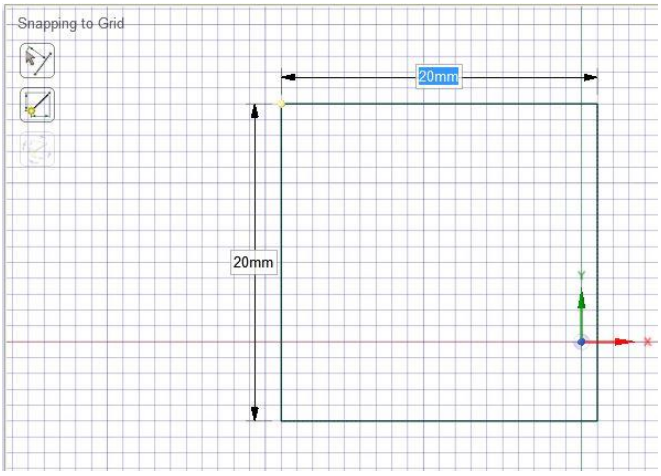

Fig. 2-2 Drawing rectangle

Step 5: Enter the appropriate length (20mm), and press Enter, then the quadrilateral is complete. While entering the length, you can also press Tab to switch coordinate axis (side) and enter the length.

Step 6: Select the main view in the locating tool.

Step 7: In the editing tool, select the pull function.

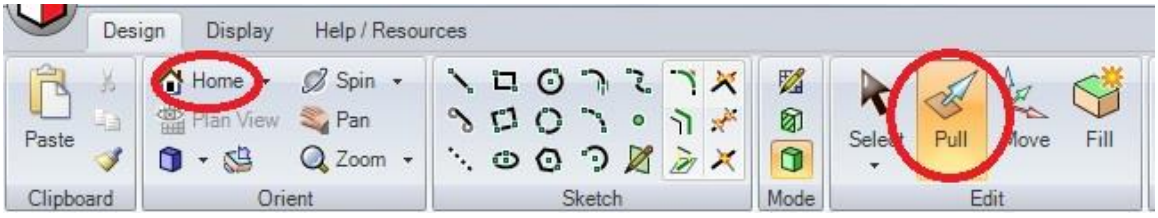

Fig. 2-3 Select pull

Step 8: Left-click the quadrilateral plane and move along the direction of the yellow arrow and pull out the length you want, and then a cube emerges.

Step 9: Enter the appropriate length (in this case is 20mm), and press Enter.

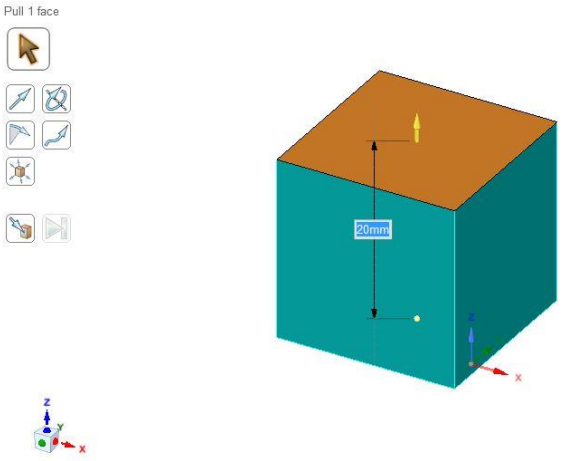

Fig. 2-4 Enter the height

Step 10: Left-click on the space to end the pulling action.

2-2 Fillet Cube

Step 1: First of all, please select the edge of cube to fillet, left-click the edge to select. To select multiple edges, you can press Ctrl and the left mouse button simultaneously. You can also double left-click on one of the edges, and all the edges on the same plane will be automatically selected.

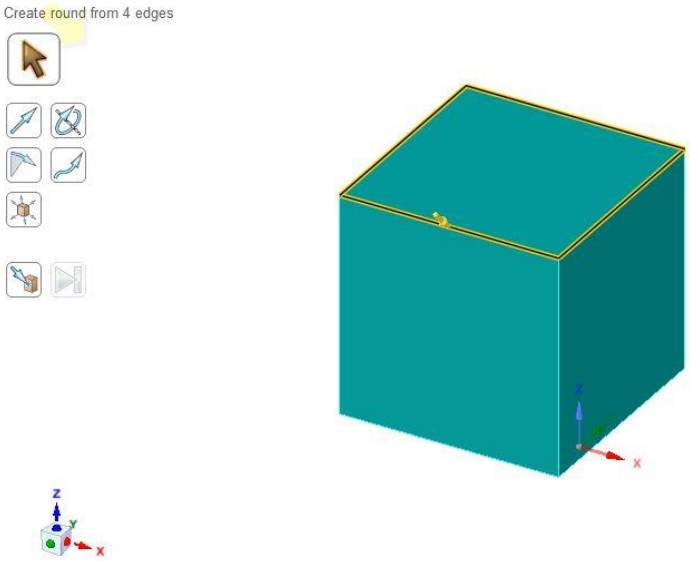

Fig. 2-5 Select the edges

You can zoom in and out and rotate the cube using the middle mouse wheel, to increase the ease of edge selection. Also, you can cancel smart drawing in the option menu at the bottom right of the screen and select the edges, as a result, you can only select the edges in the cube.

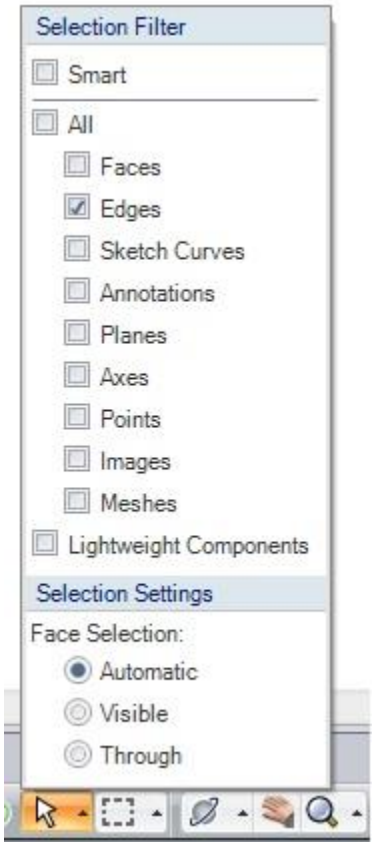

Fig. 2-6 Filter selection

Step 2: After selecting all the edges (they will be highlighted in yellow), select pull, left-click on the edge and pull along the direction of the arrow, and type the desired radius of curvature R (in this case is 2mm), and press Enter.

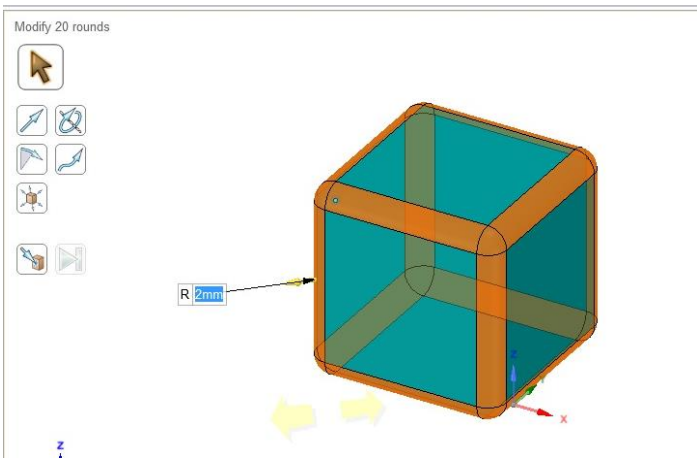

Fig. 2-7 Fillet

Finally, click the File menu on the top-left corner of the screen, save this file named Square.

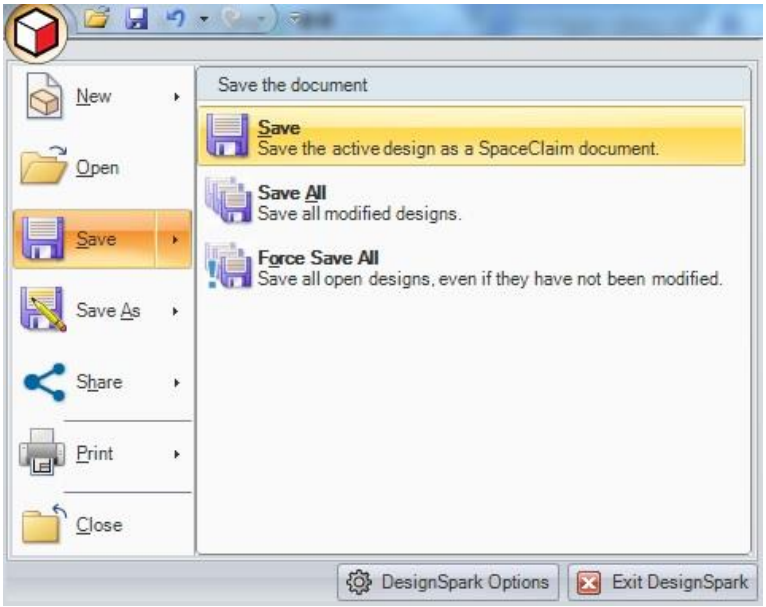

Fig. 2-8 Save

2-3 Shaft and Hole

2-3-1 Shaft

Continuing to use the cube we created in the previous two sections, this section will produce the shaft and the hole in the cube.

Step 1: Click on any plane, select draft in mode tool, this calls out the grid of the plane.

Step 2: Select the circular function in sketch tool.

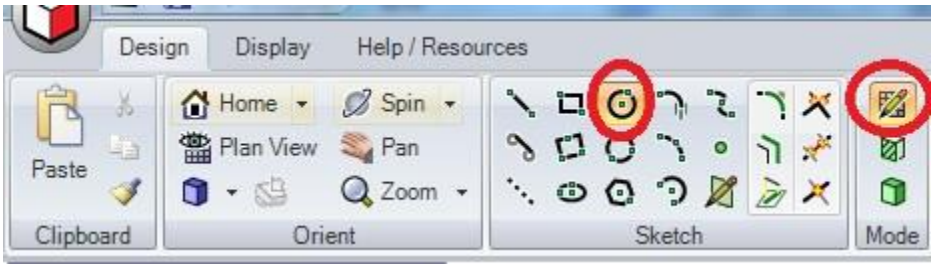

Fig. 2-9 Select circular

Step 3: Select one of the planes on the cube, left-click on its center point, pull aside, enter 10mm and press Enter.

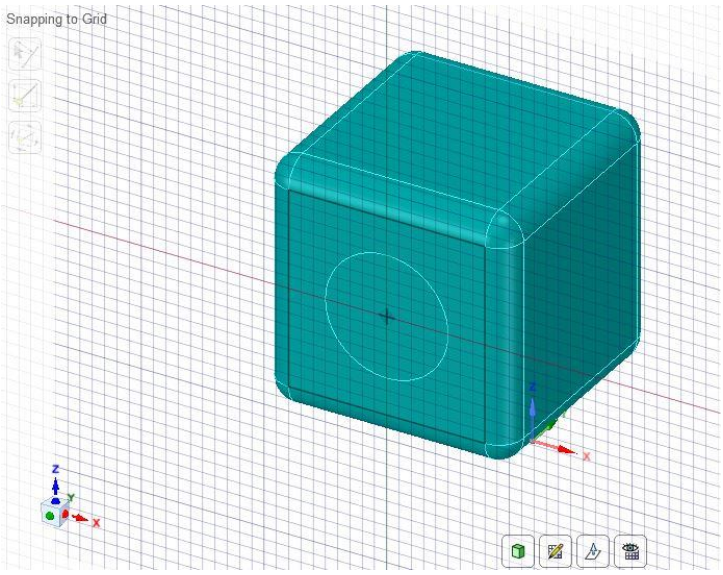

Fig. 2-10 Draw a circle on cube surface

Step 4: Choose pull under the edit tool, left-click the circular plane, pull along the yellow arrow, enter 7mm and press Enter.

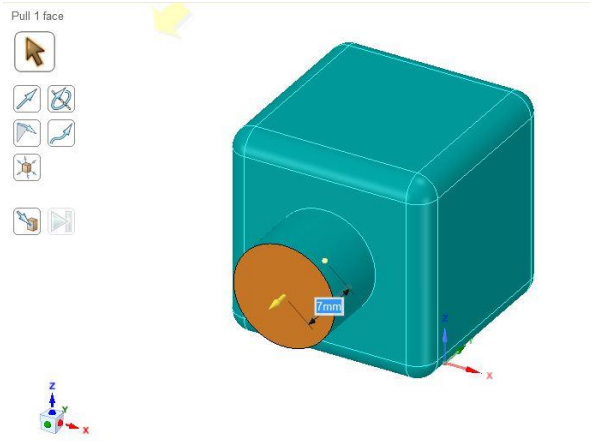

Fig. 2-11 Pull out 7mms.

Step 5: Repeat the Step1~Step4 on the other 5 planes, let all of the planes have the shafts

extended out. You can change the viewing angle by the locating tool, click the plane and choose draft to choose the center point.

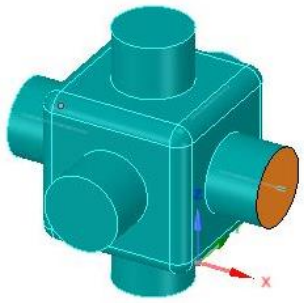

Fig. 2-12 Finished object after all the shafts are pulled out

Step 6: Simultaneously press the Ctrl and left mouse key to choose all the edges of the shafts and choose pull, fillet all the edges at the same time, enter 1.5mm (fillet radius) and press Enter.

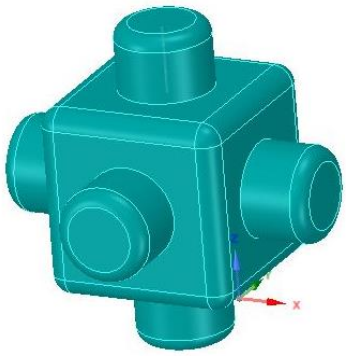

Fig. 2-13 Create the fillet

Step 7: After drawing, save it as a new file, and name it Pin Cube.

2-3-2 Hole

In this paragraph, we want to drill holes on a cube's surfaces, and combined it with the object in Fig. 2-13.

Step 1: Draw a cube with dimensions of 20mms, as shown in Fig. 2-14.

Step 2: Click on any plane, select Sketch tool in mode, this produces a grid.

Step 3: Select the circular sketch feature tool.

Step 4: Select one of the surfaces of the cube and left click at its center, drag to the side, type 10mm, and press Enter.

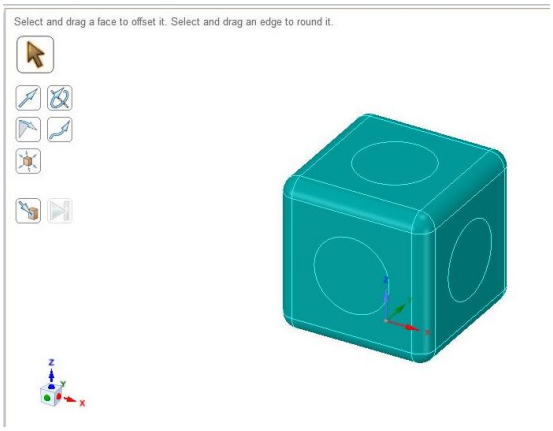

Fig. 2-14 Draw another circle

Step 5: Select Pull in the Edit tool, left click the plane with the circle, pull along the direction of the arrow toward the inside, pulling the distance needed to more than the length of the cube, which is 20mm. And then left click the blank space, a hole will appear in this side of the cube, as shown in Fig. 2-15.

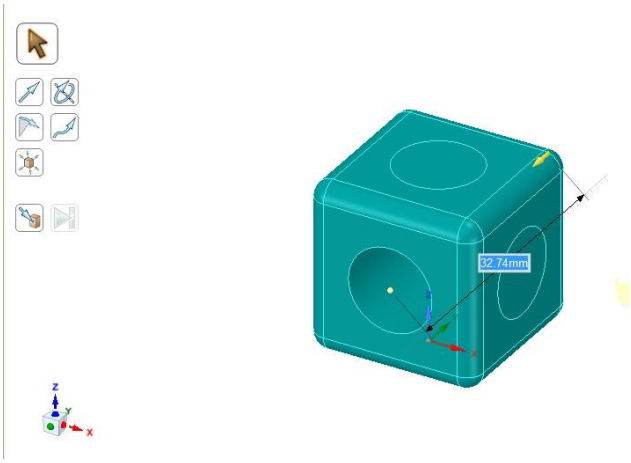

Fig. 2-15 Pulling and cutting

Step 6: Repeat steps 2-5 on all sides of the cube, making one hole on each plane.

Step 7: Chamfered holes in each plane. Next, choose Select Items in the element setting area on the left side, and choose “Edges with same length”, these edges will be highlighted at once.

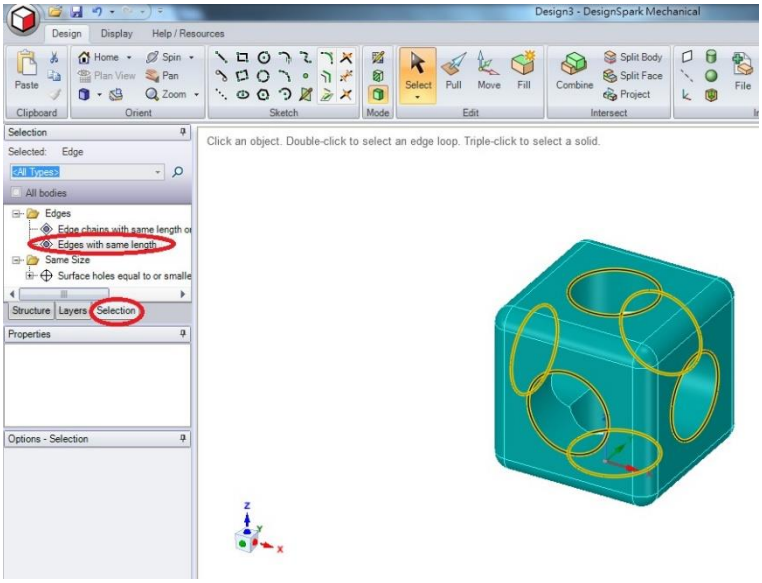

Fig. 2-16 Choose edges

Step 8: Select Pull, making each edge chamfered at the same time. Select Chamfer icon in the Pull mode menu at the bottom left of the screen.

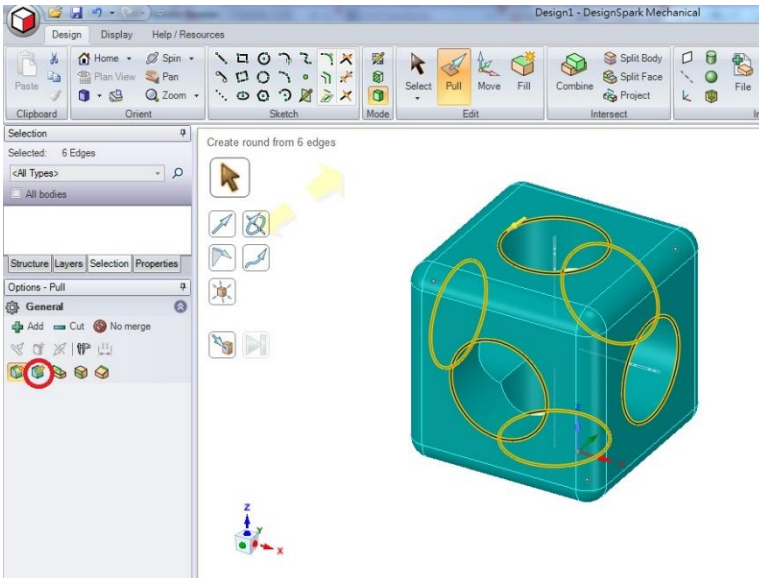

Fig. 2-17 Choose same edge

Step 9: Left click the selected edge, and pull along the direction of the arrow, type 1 (radius), and press Enter.

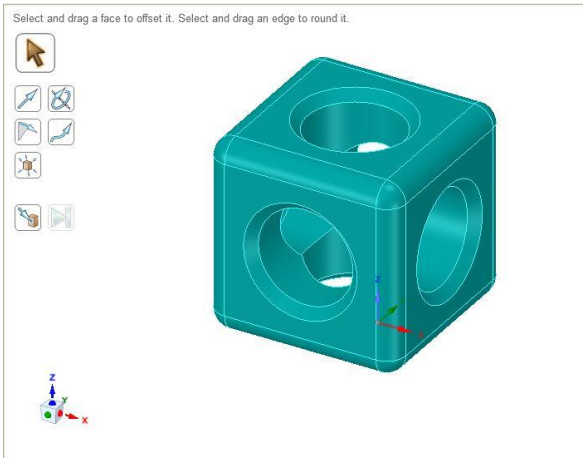

Fig. 2-18 Chamfered complete

Step 10: After drawing is complete, save new file, and name the file as “Hole Cube”

2-4 Cube Assembly

Step 1: Please open pin.rsdoc and hole.rsdoc files that you have already drawn in sections 2-2 and 2-3, the file name can be seen at the bottom of the program after opening.

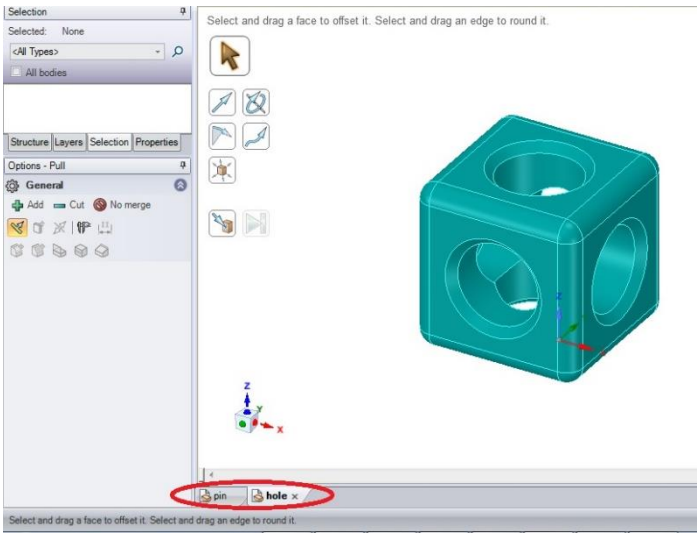

Fig. 2-19 Open the files, the file name is shown at the bottom left

Step 2: Click Hole Cube three times in a row, select the entire entity, use replication at the top left of the toolbar.

Step 3: Click to enter the Pin Cube design, and use Paste (using the Paste command will not modify the original file) to import Hole Cube (it does not matter if the import model and the original model are overlapping). After importing, save new file as “Cube Assembly”.

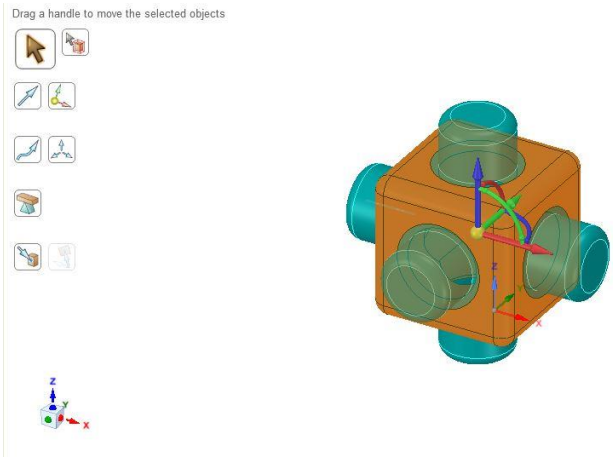

Fig. 2-20 Import object

Step 4: Select Bold in the Edit tool, select Pin Cube (triple-click in a row), a three-axis movable anchor will appear in the model, three-axis representing the x, y, z axis. Select one of the axis and move the object to reposition the Cube. You can move these two cubes away from each other.

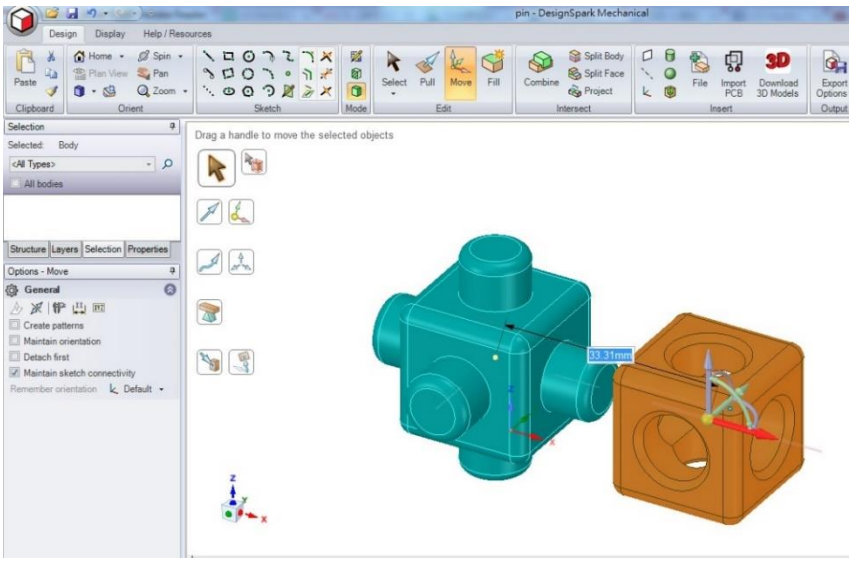

Fig. 2-21 Move one object axially

Step 5: Next, we want to put the axis into the hole of Hole Cube to combine two cubes. You will need to do Goto and Move, these two operations move the cube to the right position.

Make sure you choose the Move function, and select the entire Cube, drag the center point of the mobility anchor, and place it on the surface of one cube that you want to align with another cube. In other words, we need to align one surface of each cube in the same horizontal plane, so please move the anchor to the surface for alignment.

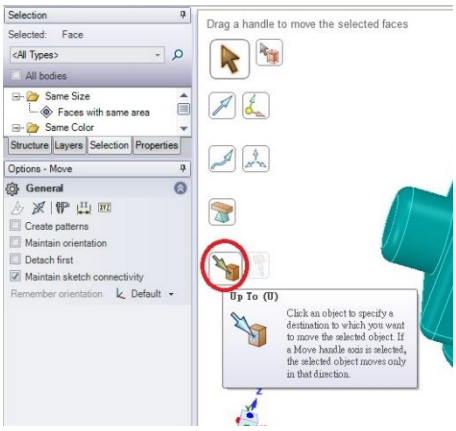

Fig. 2-22 Select Goto

After moving the anchor to the surface we want, click Goto, and click the surface for alignment (you can also click on the point or edge, as long as they are on the very surface), then the two surfaces will be aligned on the same horizontal plane.

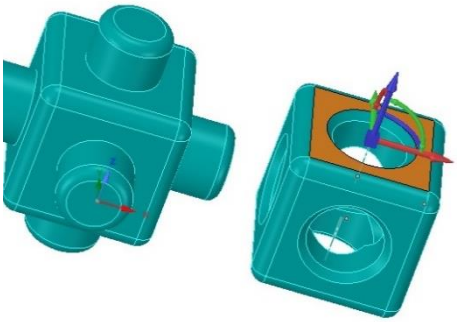

Fig. 2-23 Select the surface you want to go

Step 6: Repeat Step 5 to align the surfaces of the three axis, the two cubes are combined together.

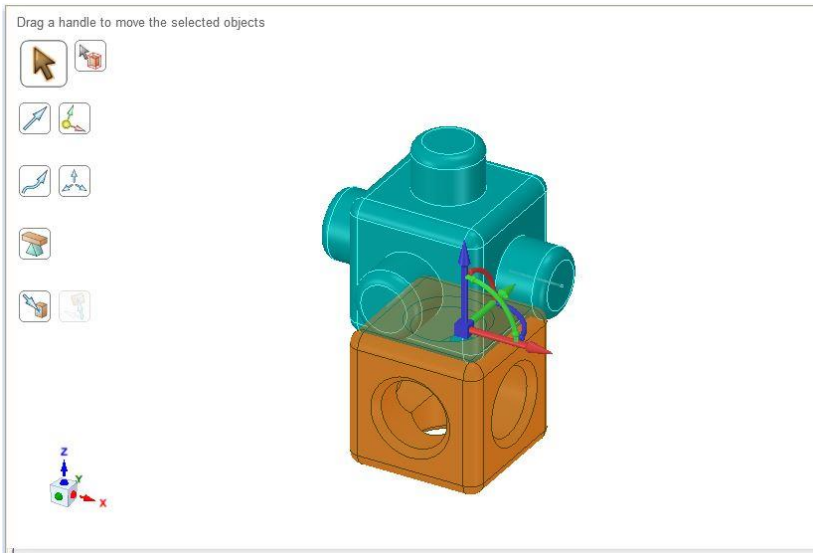

Fig. 2-24 Two aligned components

Step 7: Do some practice. Please duplicate more cubes and repeat the previous steps to combine them together.

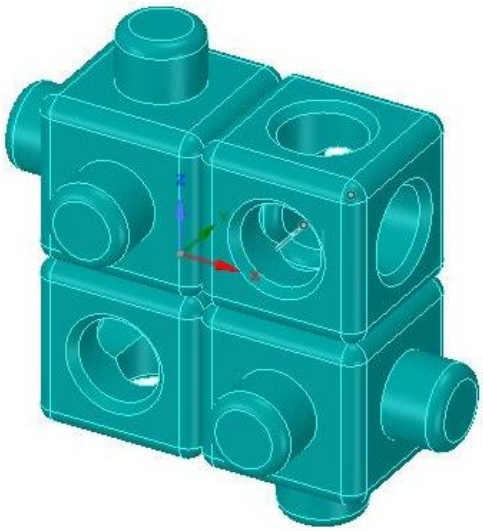

Fig. 2-25 Four combined objects

2-5 Fill

Fill tool is used to remove individual shapes in the model and replace it with a geometric solid form. The fill tool can be used before or after the selection of the shape on the model. In other words, if the model is selected after Fill in the Edit tool has being selected, the fill tool effect can be used continuously. Conversely, if the fill tool is selected after the model selection, the fill tool will deselect automatically after this part is filled.

Step 1: Open the Hole Cube drawing file for practice.

Step 2: Select the fillet. Click any fillet in the Hole Cube to select the surface.

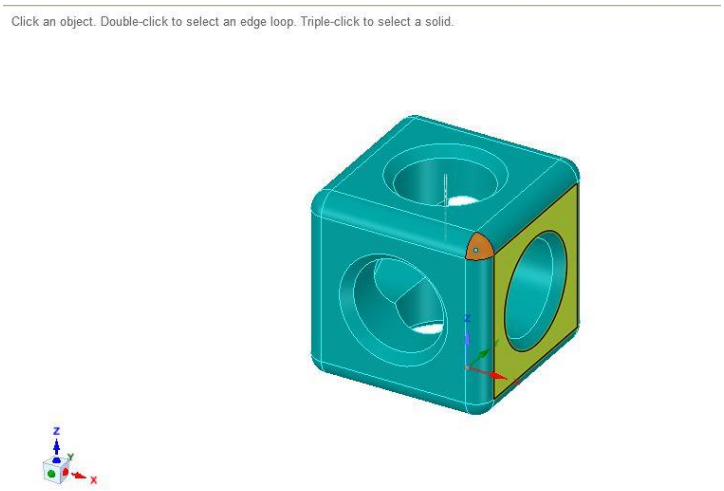

Fig. 2-26 Select fillet

Step 3: Fill the fillet. Choose Fill in Edit tool, you can see the fillet surrounded by three sides of the cube is filled by the extension of the three sides (the Fill function can be checked this time).

Fig. 2-27 Fill

Step 4: Fill the fillet. Then fill the remaining seven angles all at once, first select one of

the rounded corners, left click on Select Items on the left of the program, and select the Drawing with the same area. At this point the surfaces with the same area are all selected, the next is same as Step 3, choose Fill in the Edit tool, fillets are all replaced by edges.

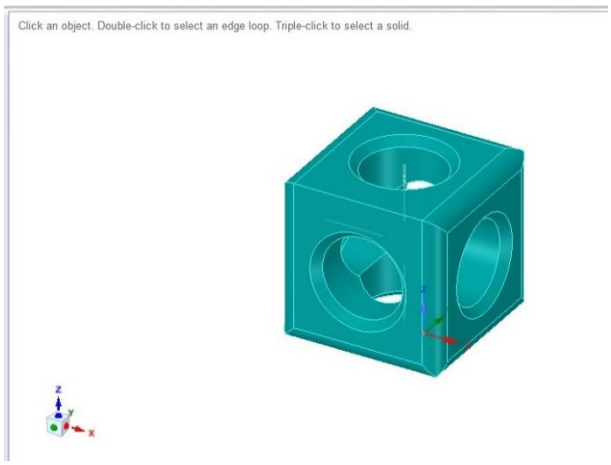

Fig. 2-28 One edge is returned to no fillet

Step 5: Select Fill. Select Fill in the Edit tool again (Be careful not to click on the model first).

Step 6: Fill the edges. Because the model is not selected first, the Fill function is continuously available. Click on any round edges on the center between the two sides of the cube, and press Enter, you can see two planes extend out and replace the rounded edge. Repeat the same procedure to fill all the edges.

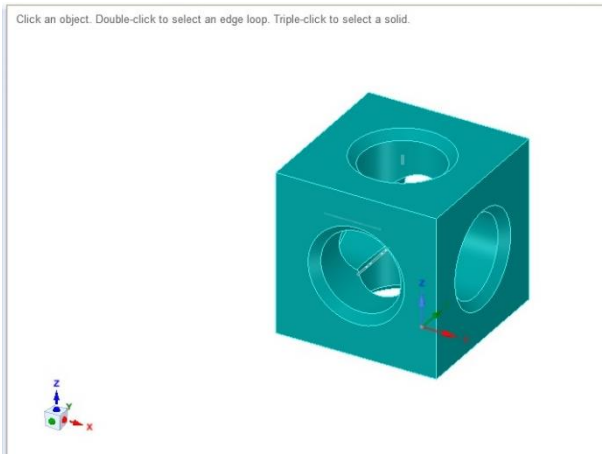

Fig. 2-29 Fill all the other edges

Step 7: Choose the edge of the holes on the cube, and then select “Equal or smaller-sized chamfers” under Selection section on the left-hand side window, and press Enter to fill all the chamfers.

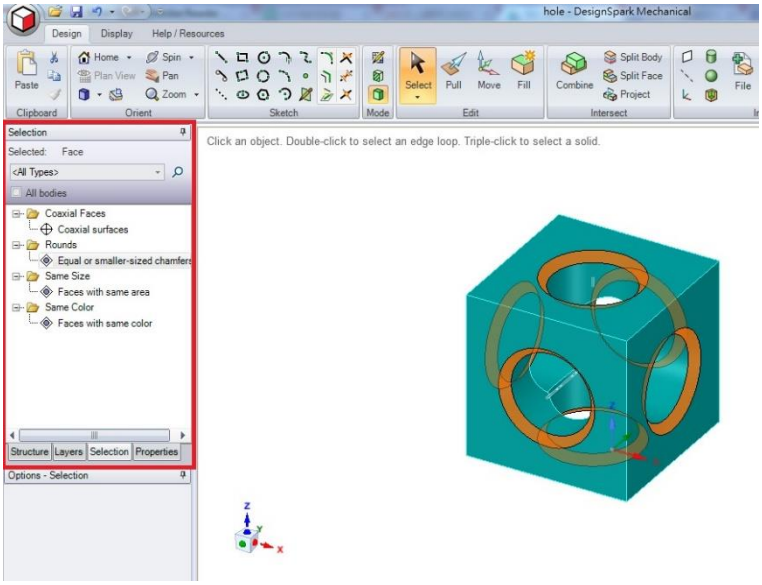

Fig. 2-30 Fill the chamfer

Step 8: Done.

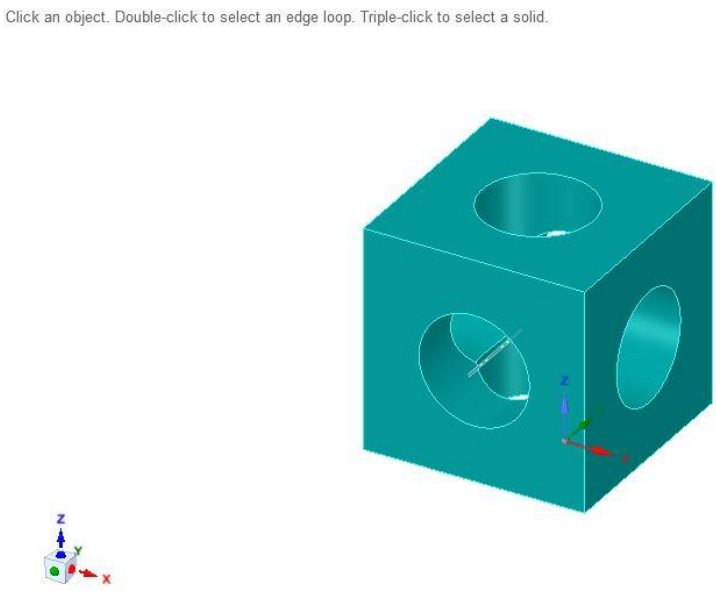

Fig. 2-31 Completed

2-6 Summary

All these projects are nothing more than starting from a simple cube and performing some cutting or extending. In this chapter, we started with the drawing of a simple cube, and used some basic commands such as Move, Fillet, and Combination. Practice and perfect these new skills.

Chapter 3 – Advanced Modeling

This chapter will use several common commands and a few examples to help you understand the use and application of these commands in DesignSpark Mechanical. By drawing a cup and a screw, you will learn how to generate complex facets.

3-1 Drawing a Cup

You will learn how to draw a cup with DesignSpark Mechanical.

Step 1: Open a new design, click on the front view, and start drawing.

Step 2: Draw side view of the cup. Click on sketch lines, and draw side plan view of the cup. There is no limit on length and width, the design can be based on your personal preferences.

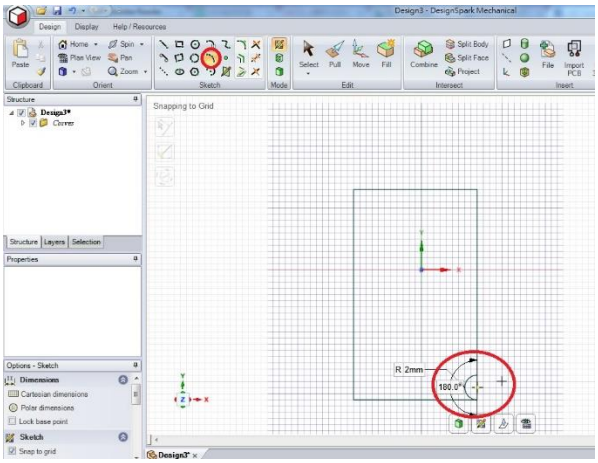

Figure 3-1 Draw contour of the cup

Click the three-point arc in Sketch tool, and draw a semicircle at the bottom of the cup. Please click on both ends of the semicircle, and then select the arc angle, we will use this graph to sweep out the cup body.

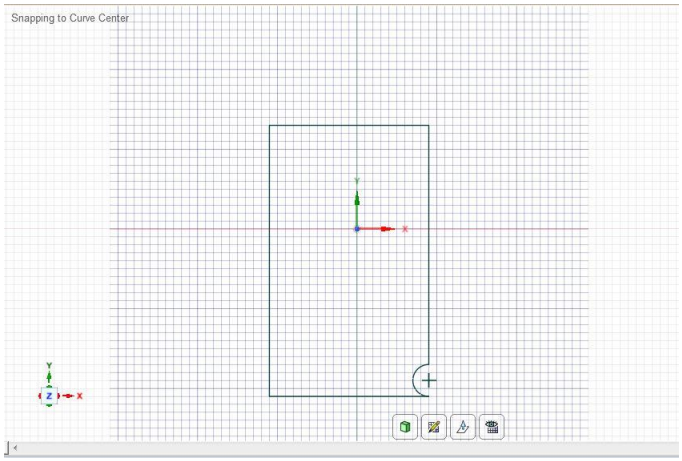

Figure 3-2 Three-point arc

Step 3: Draw main body of the cup. Select the main view, click Pull in the Edit tool,

select Rotation in the drawing area, and click the central axis of rotation, rotate the plane along the arrow to draw the cups main body.

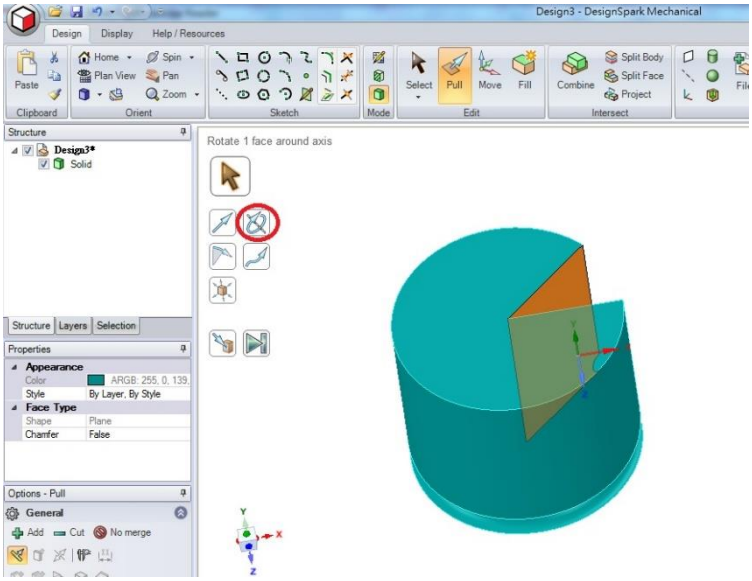

Figure 3-3 Pull and rotate

Step 4: Draw cup thickness. Select Pull in the Edit tool, and click on the top edge of the cup, and then select Copy Edge, pulling inward along the arrow until you produce the desired thickness of the cup. After finishing the pulling or entering a value, press Enter, at this time we can see a circular line at the top of the cup.

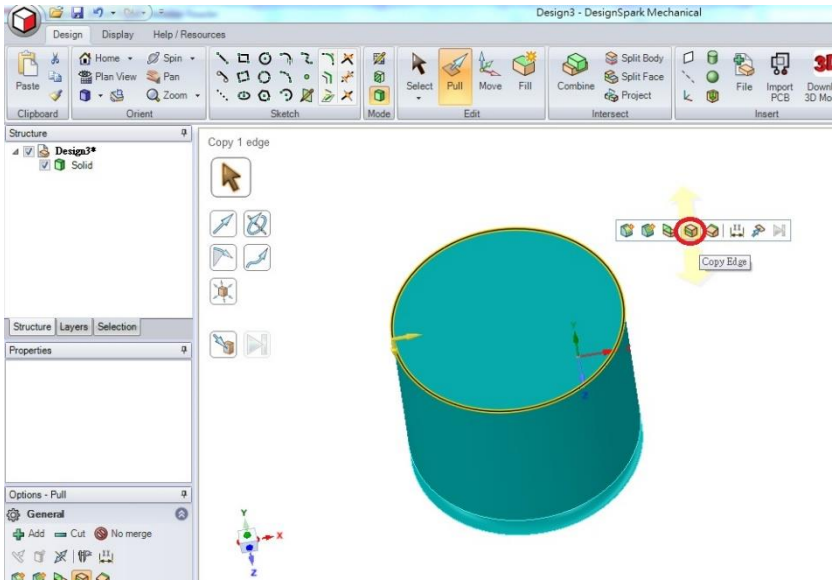

Figure 3-4 Copy edge

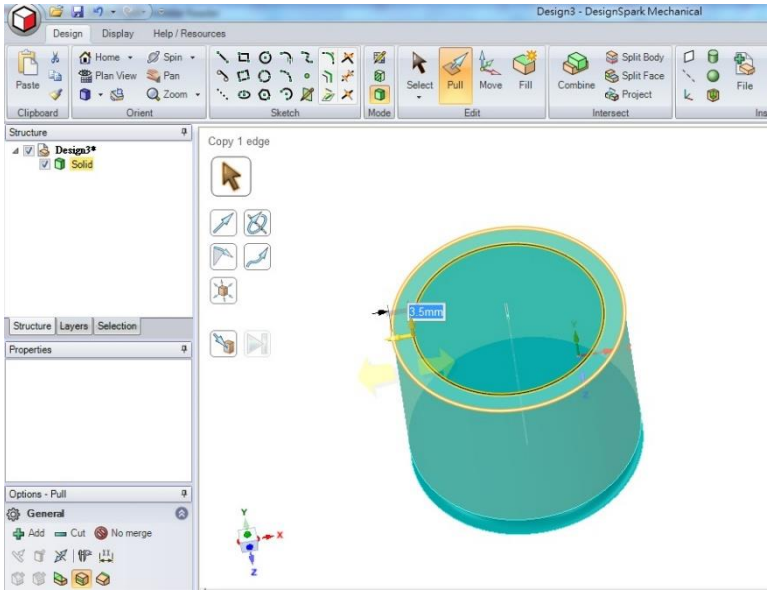

Figure 3-5 Adjust the thickness

Step 5: Establish the internal space of the cup. Select Pull in the Edit tool, click on the circle you have just drawn, pull down along the arrow to the bottom of the cup, and this is the cup depth, which you can adjust. When adjusted to the proper position, press Enter.

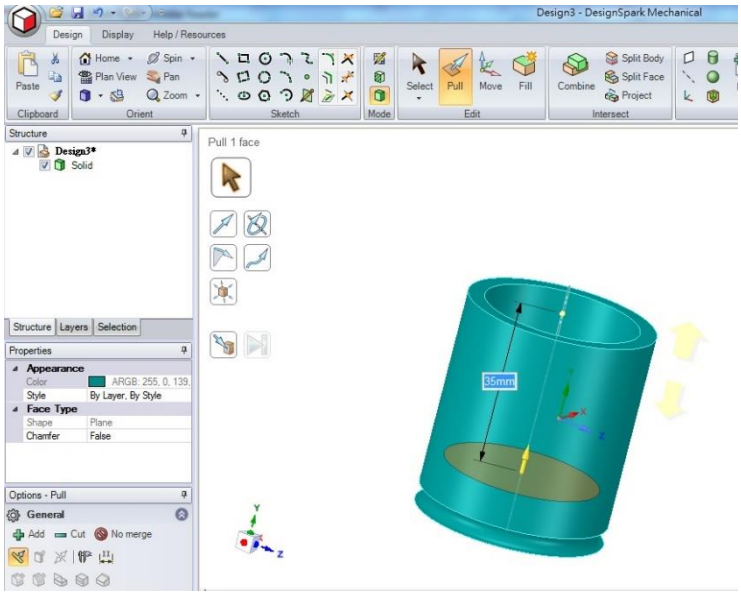

Figure 3-6 Pull cup space

Step 6: Pull the cup outward. Select Move in the Edit tool, left-click the outer edge on top of the cup, then on the top right will appear a menu, select the Pivot Edge, then the edge will have a yellow arrow. Please pull out the edge of the cup along the direction of the yellow arrow, if the direction of the yellow arrow is not the direction we want, you can click again to change the direction of the pivot edge.

While pulling out, it is recommended you directly enter the desired value. It would be more convenient for you to pull the inner ring later, or the thickness might be inconsistent.

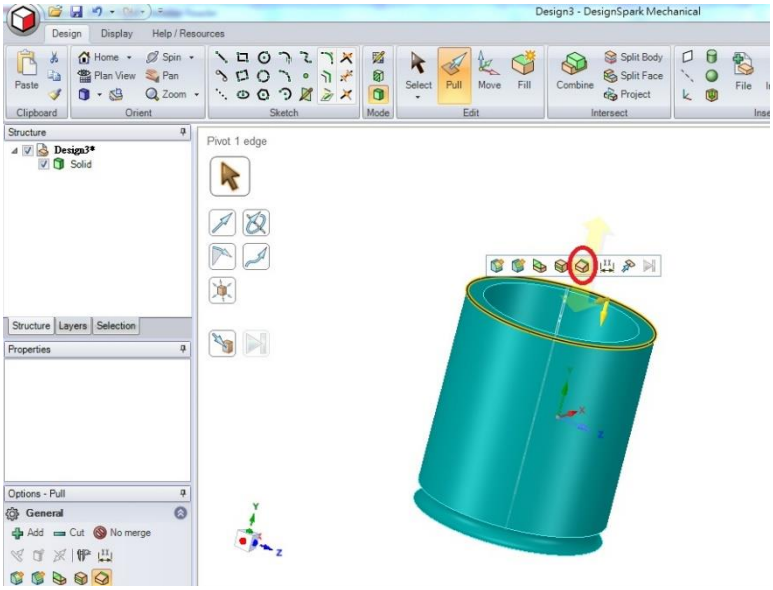

Figure 3-7 Select the pivot edge

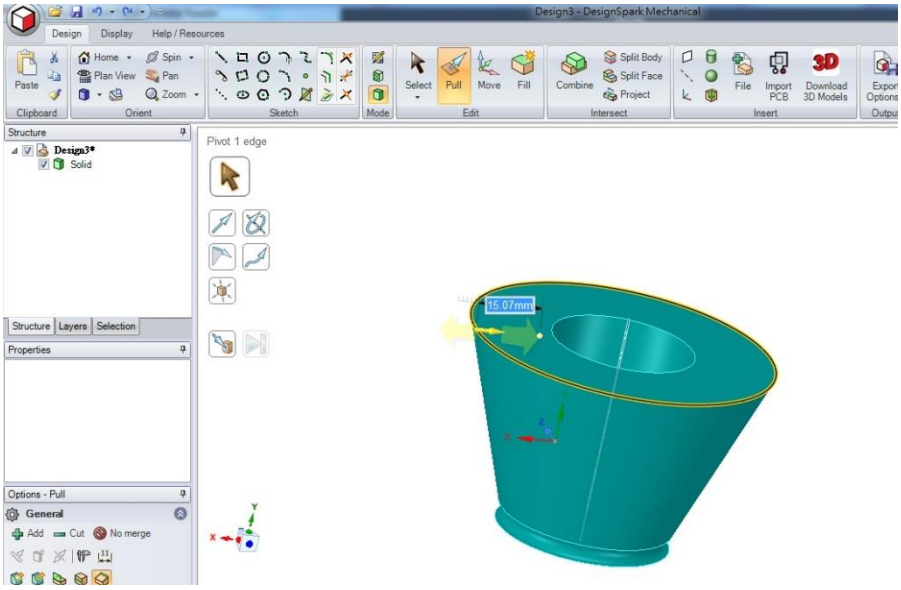

Figure 3-8 Pull out the edge

Step 7: Pull out inner edge. The idea is the same here. Choose Move in the Edit tool, and then select Pivot Edge. We can follow the yellow arrow direction to pull out the edge of the cup, and enter the value you used for pulling out, let the cup wall extend outward

evenly.

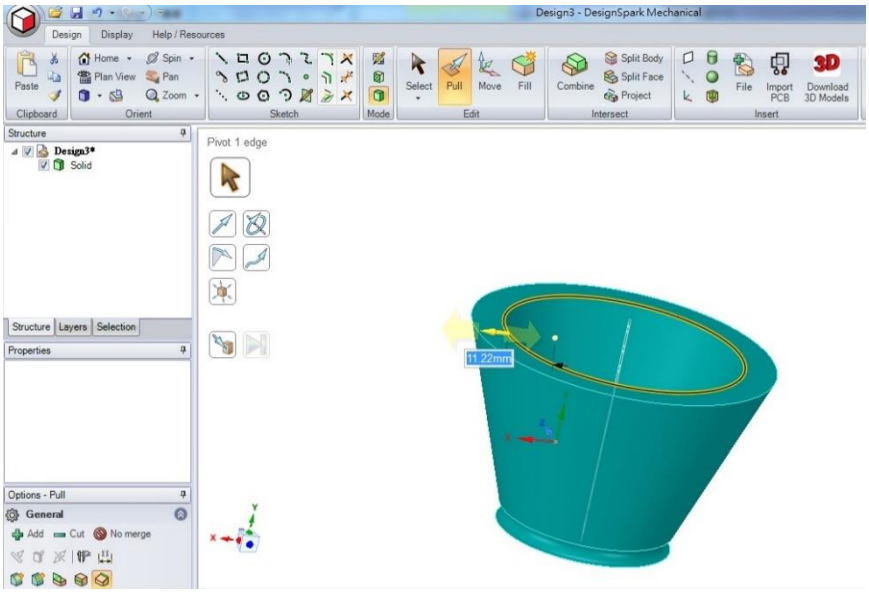

Figure 3-9 Pull out the inner edge

Figure 3-10 Complete pulling wall of cup

Step 8: Draw lines of handle. First, we must first go to the front view (maybe your front view is different from this example, you only need to adjust the view until is convenient for you to draw lines to the side. Select Cloud Curve in the Sketch Tool, and then select the two appropriate locations in the inner wall lines, draw a curve, and press Enter.

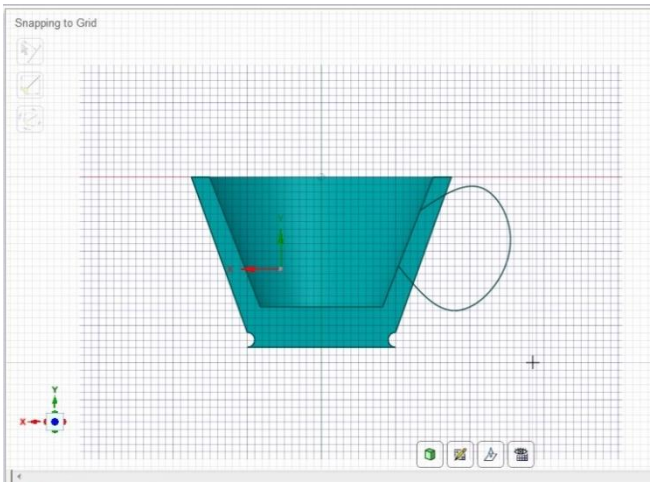

Figure 3-11 Curve of handle

Step 9: Insert plane. We can establish a plane based on the selected object, in order to draw graphics. Click Plane in the Insert tool menu, and then click on the intersection of the inner wall of the cup and handle, an auxiliary plane will appear at the intersection.

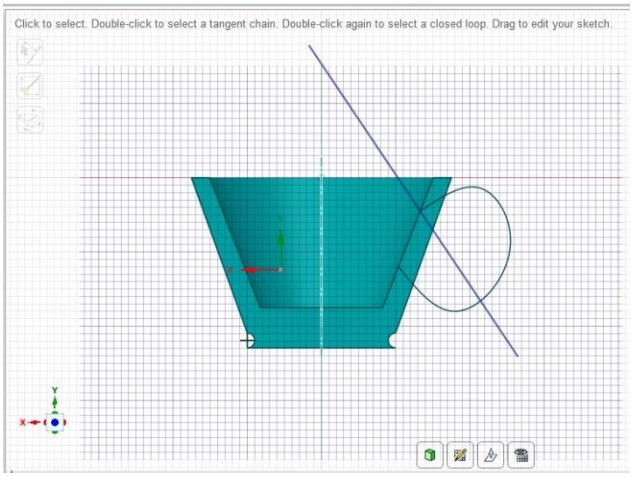

Figure 3-12 Establish an auxiliary plane

Step 10: Eliminate unnecessary checks. In order to facilitate the drawing, we can keep the plane and handle lines in the structure first, uncheck the rest, and adjust to a good drawing angle.

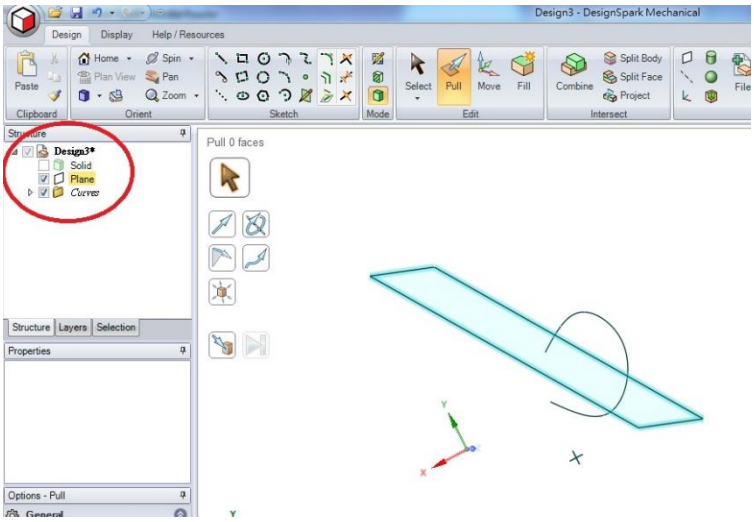

Figure 3-13 Uncheck unnecessary structures

Click Circle in the Sketch tool, and draw a circle of radius of about 1-2 cm at the bottom

line.

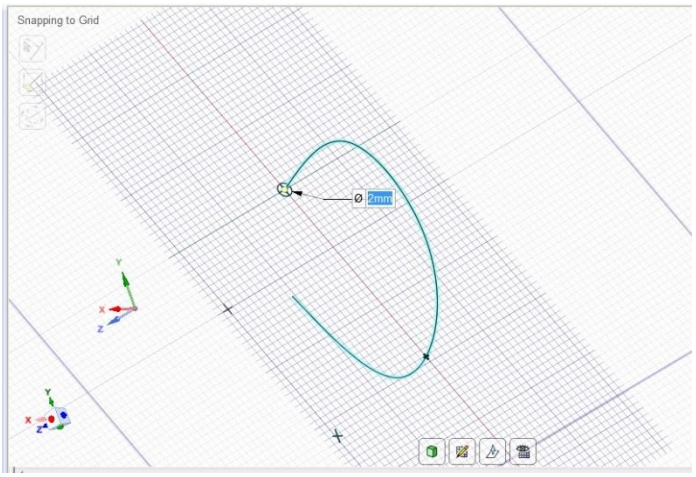

Figure 3-14 Draw the circle

Step 11: Sweep. Next we want to draw the handle through the curves and lines, click Move in the Edit tool, and select Sweep in the left of drawing area. The Sweep command will extend the contour or shape along a specified path to create a solid surface. So after you have selected Sweep, click Circular Planar in the left of the Structure menu, then click lines to sweep (cup handle line), and using the arrow sweep the handle entities.

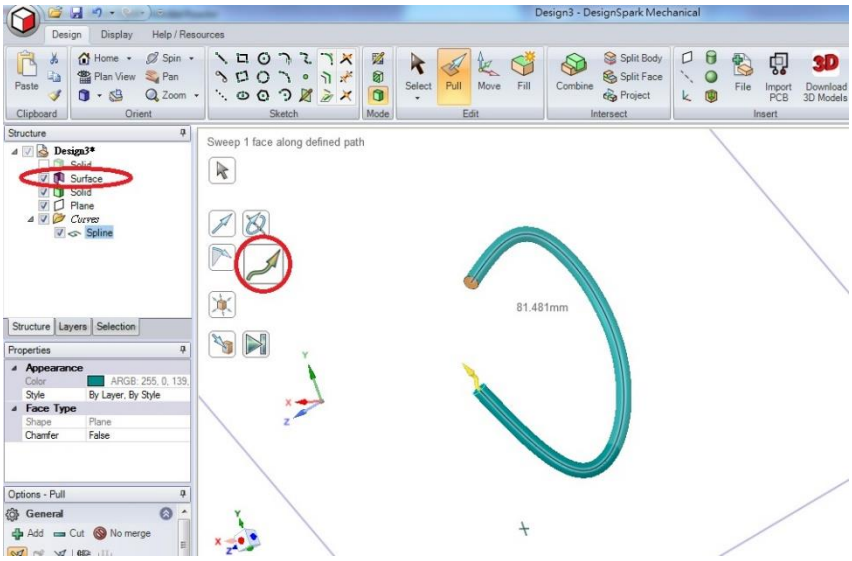

Figure 3-15 Sweep to create cup handle

After the sweep is completed please open Other Objects in Structure, move the screen a little to check if the sweep is a success.

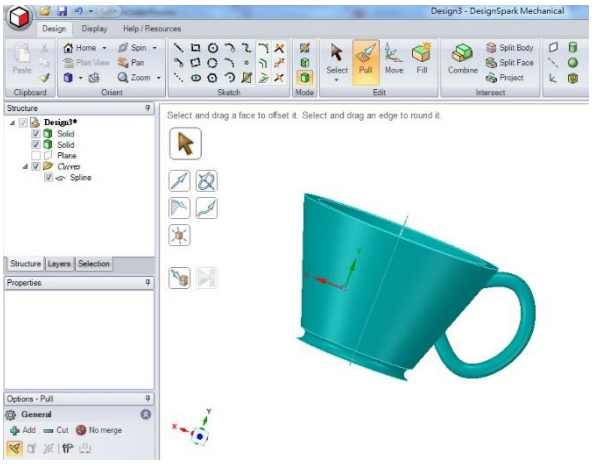

Figure 3-16 Cup almost ready

Step 12: Fillet the top of the cup. Click Pull and press the Ctrl key and select the two

circular lines (inner and outer) at the top of cup, and select Fillet in the Pull mode menu, which is in the left screen, pull out the arrow until both sides contact with each other.

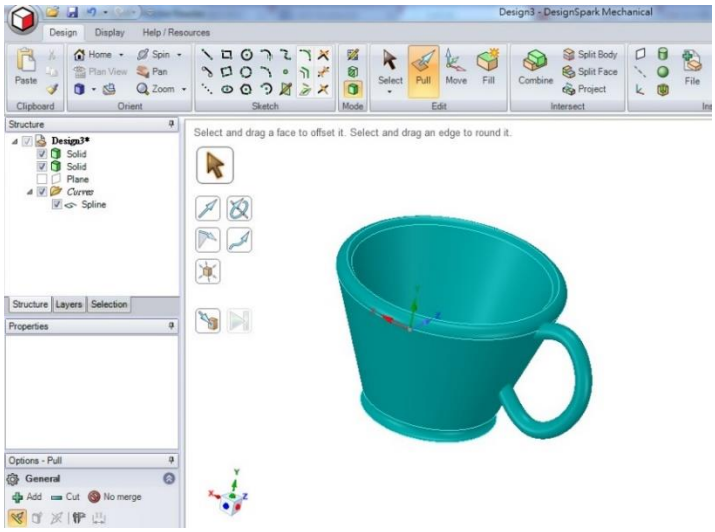

Figure 3-17 Fillet of the top of the cup

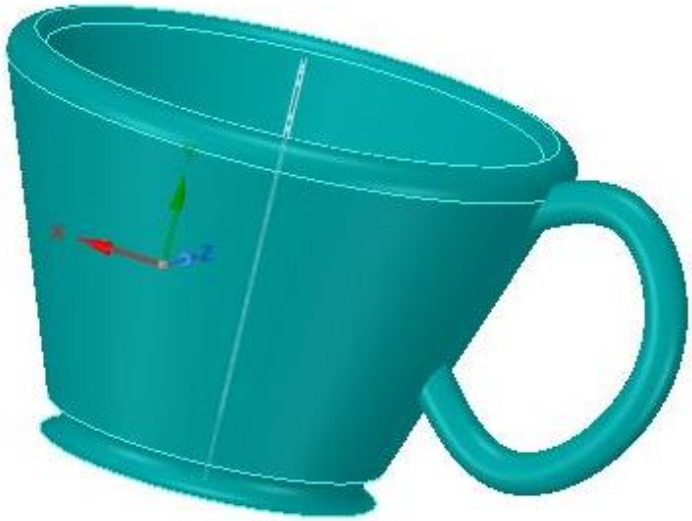

Figure 3-18 Well done!

Step 13: Finally, click Edge in Display tool, and uncheck the tangent and entities.

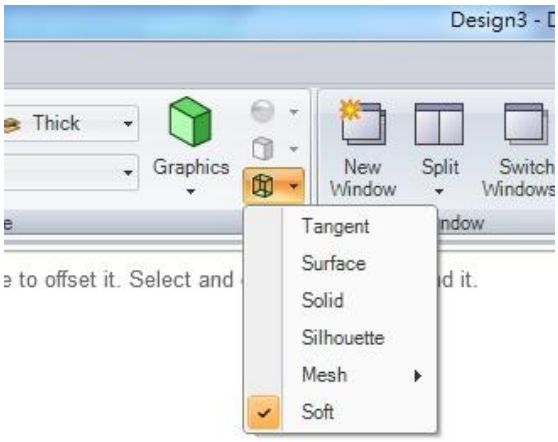

Figure 3-19 Uncheck display

3-2 M8 Screw

In the assembly process of various parts, the screw and nut are indispensable parts, and drawing a screw in DesignSpark Mechanical is not difficult. DesignSpark Mechanical provides some convenient tools that allow us to quickly get this done, this section will teach you how to draw an M8 screw.

Step 1: Establish a cylinder. First, draw a cylinder with diameter of 8mm and length of 11mm, as the lower half of the screw.

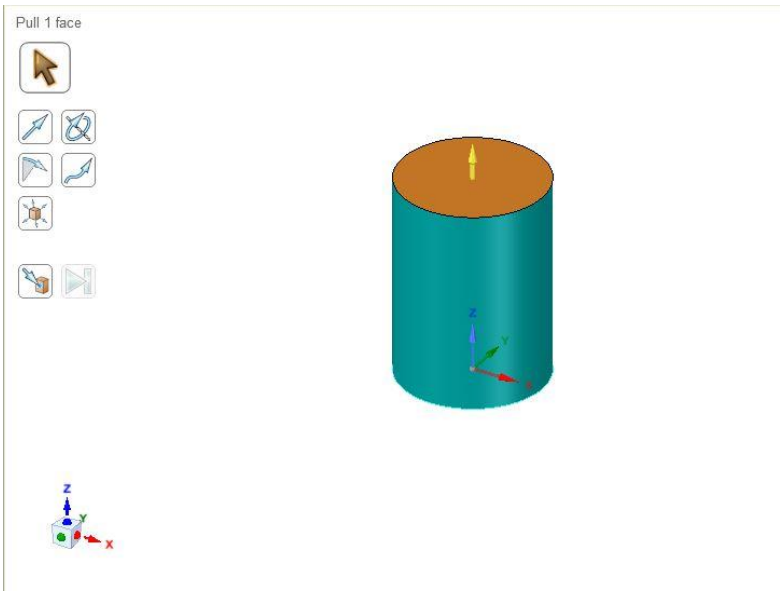

Figure 3-20 Draw a cylinder

Step 2: Sectional mode. In order to quickly draw the thread, you can switch the viewing angle to a cross-sectional view of the cylinder. Please click on the center line of the cylinder first, and select tri-axial index at the bottom left of the drawing area, select the appropriate shaft as the plane of the section, then click Section in Mode tool.

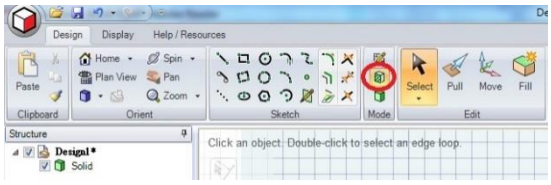

Figure 3-21 Section mode

Step 3: Generate thread. Plot an equilateral triangle with side length of 1.24mm in the flat top of the cylinder (M8 screw has pitch of 1.25mm, but since an equilateral triangle with

the side length of 1.25 cannot pull the spiral, just draw a side length of less than 1.25mm).

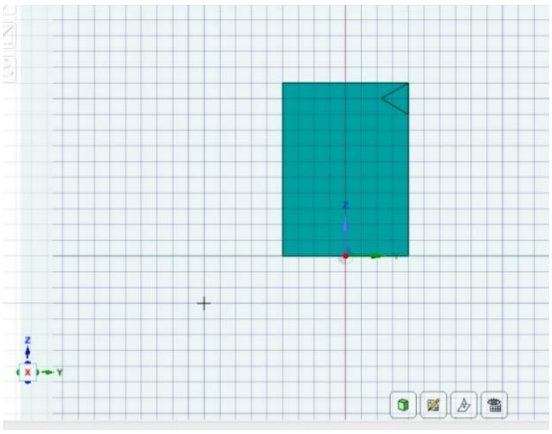

Figure 3-22 Thread

Step 4: Pull setting. After the drawing of an equilateral triangle on a cross-sectional graph, click 3D mode in Mode tool, turn the viewing angle into a perspective view, and then we want to cut the thread by the equilateral triangle you have just drawn.

Select Pull in the Edit tool, then click the plane you have just drawn in the structure tree, which is on the left. After selecting the objects, click Rotate in the left Drawing area, and then click on the center line of the cylinder as the axis for rotation, then the rotate menu appears in the left, select Rotary Screw, and check cut in the pull menu at the top.

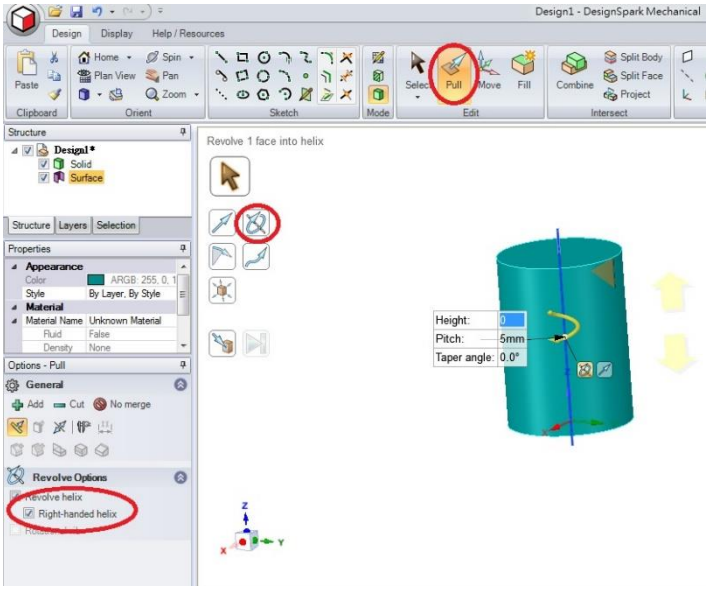

Figure 3-23 Pull setting

Step 5: Pull spiral. After setting is finished, you will see an arrow and a box in the spiral cylinder. Change Pitch to 1.25mm, then left-click on the cylinder and pull downward until the entire cylinder is cut completely.

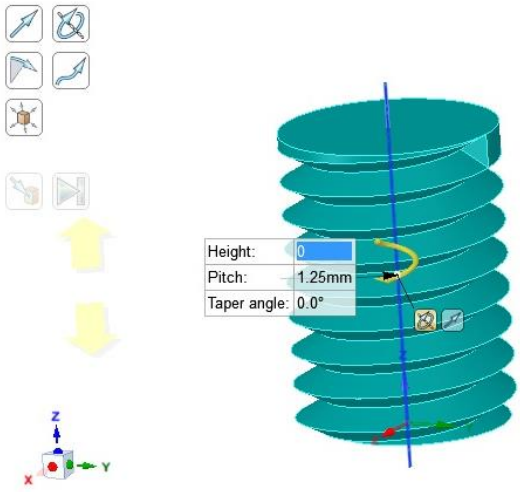

Figure 3-24 Pull spiral

Step 6: Draft mode. Next, we are going to draw a screw head on the top of this cylinder, therefore the plane of the sketch must be set at the top of it. First choose Select in the Edit tool, click on the top of the cylinder, then click Sketch in Mode tool, a grid sketch will be automatically created on the selected plane.

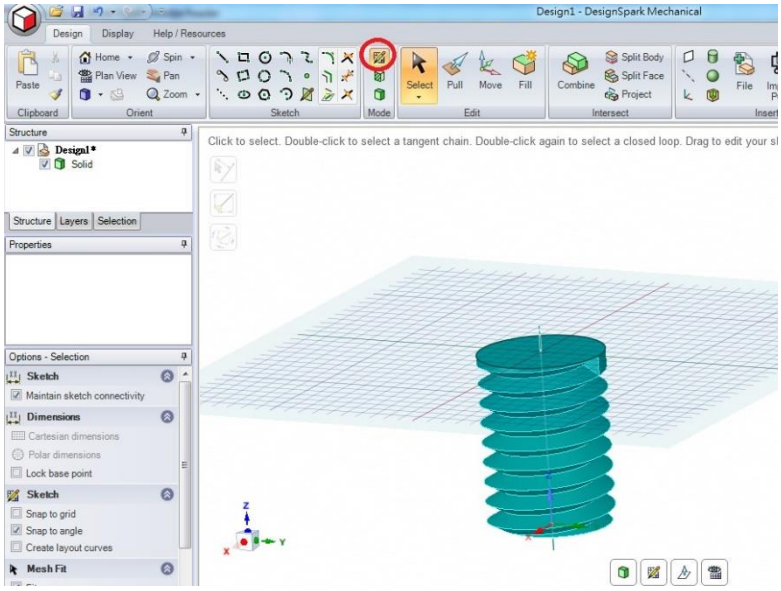

Figure 3-25 Draft mode

Step7: Draw top cylinder. Once you have a sketch plane, click Circle in Sketch, use the center of the cylinder as the circle center, and draw a circle with a diameter of 14mm. After drawing the circle, two circles will appear on the graph (outer circle and inner circle). We need to pull the two circles at the same time to make a cylinder. So click on one of the circles, then press the Ctrl key, and select another circle at the same time. Click Pull in Edit tool, and pull up the two circles to 5.2mm.

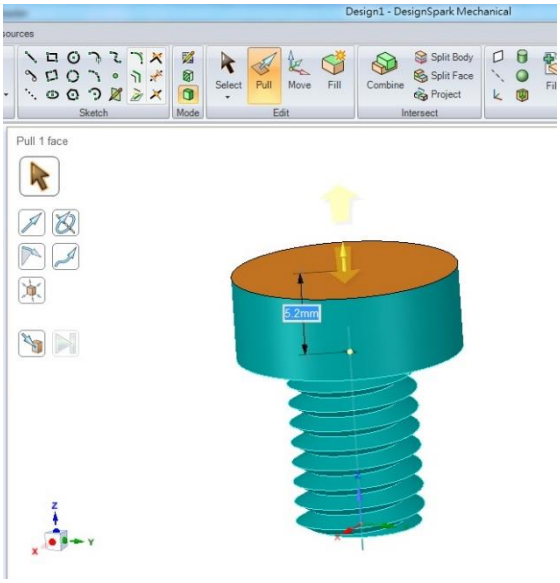

Figure 3-26 Draw cylinder

Step 8: Draw two intersected rectangles. Same as drawing the cylinder of the screw, we must first build a sketch plane on the top surface. Click Select in Edit tool. After clicking the top surface of the cylinder, click Sketch in the Mode tool.

After selecting sketch plane, select Rectangle in Sketch tool, and draw two rectangles of the appropriate size on the top plane.

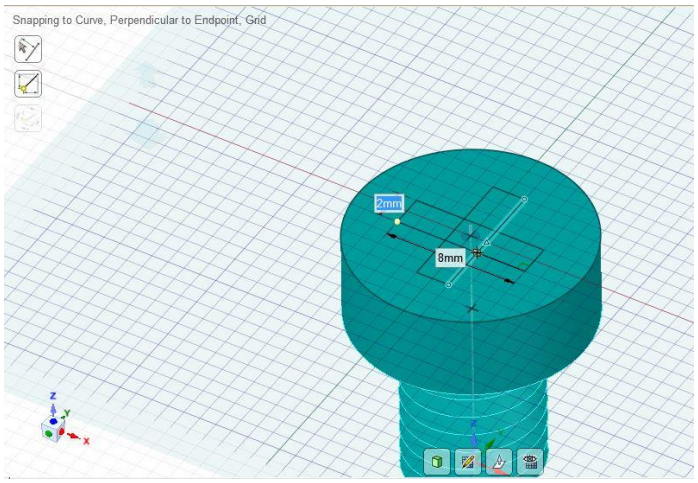

Figure 3-27 Draw two rectangles on top

Step 9: groove. We want to pull the cross-shaped plane formed by two rectangles to cut the cylinder, so delete the crossed lines before pulling, so it becomes a complete cross plane.

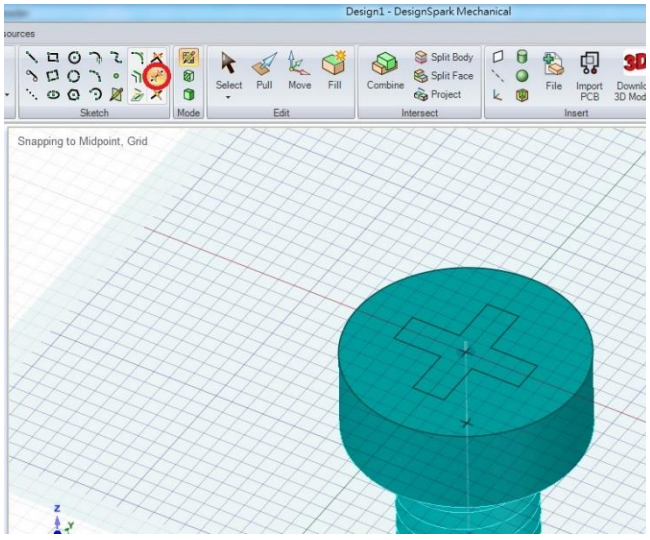

Figure 3-28 Cross groove

After you remove the extra lines, click Pull in the Edit tool, pull down the cross-shaped

surface to cut out a cross groove on the cylinder.

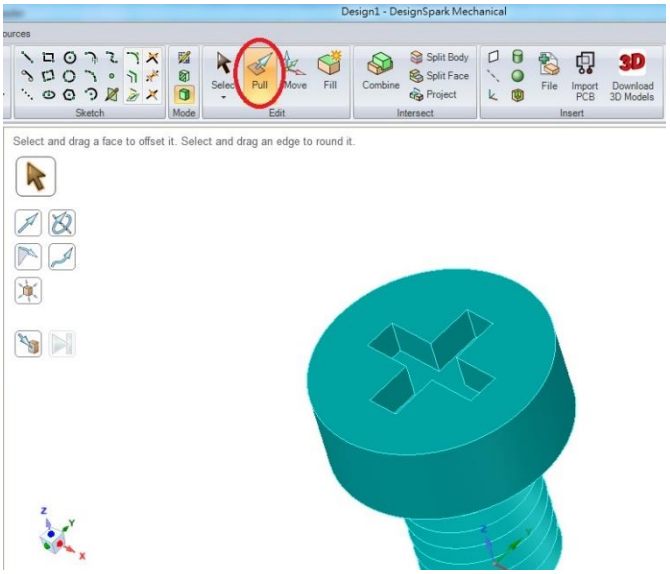

Figure 3-29 Pull out the cruciform recess

Step 10: Fillet. Finally, use Pull as a continuation of the previous step, click edge on the top of the screw head, pull fillet along the direction of the arrow.

Figure 3-30 Create fillet

Step 11: Complete

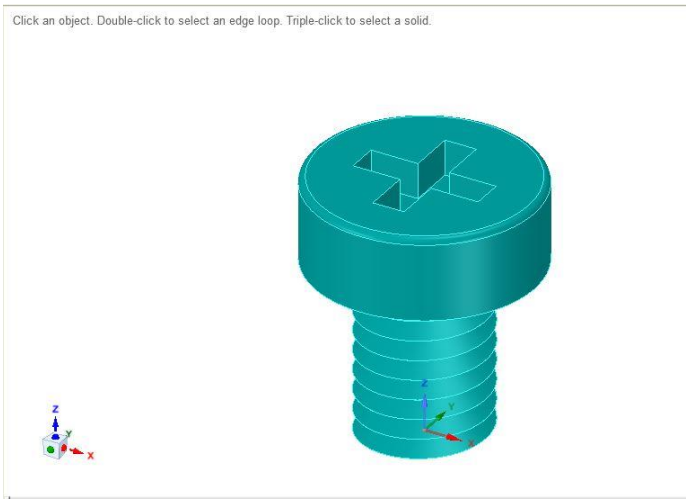

Figure 3-31 Final version of screw

3-3 Summary

Through the examples of cup and screw, we can learn and practice advanced commands and their applications.

Chapter 4 – Import Image Files and PCB Files

DesignSpark Mechanical has a huge 3D graphics library, you do not need to spend a lot of time to redrawing, as you can find suitable graphics online, download and modify them.

In addition, you can also import PCB designs like the DesignSpark PCB that the first half of this book taught you to draw.

4-1 Import Format and Method

DesignSpark Mechanical's default file format is .rsdoc. The online 3D model repository can be found in the online model library of the insert tool in the DesignSpark Mechanical main program, you open the page by clicking on it. A large number of ready-made parts can make it easier for you to complete your product design quickly.

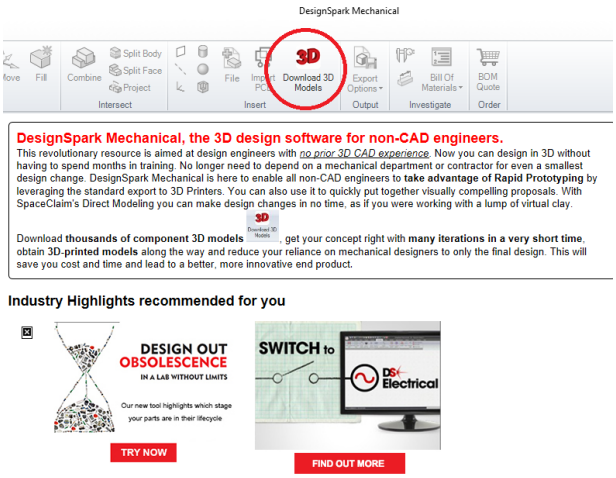

Fig. 4-1 Click the online model library

You can select the desired input pattern types in the left side of the online model library webpage, including various types of electronic parts, mechanical parts and wire. Including parts from RS, we can also find other company's image files of components in the pull down menu on the left, or search by entering the name of the components in "query type" in the middle of the screen.

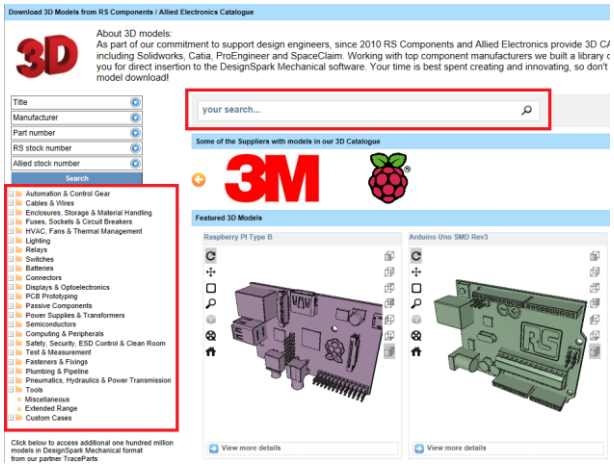

Fig. 4-2 Query Type

After selecting the image file, we can click on "CAD Insert" and directly import files into DesignSpark Mechanical after choosing the file saving path. In addition, we can also click "Download" to download new graphics (save), or "Add to basket", this function is similar to shopping carts in some sites, multiple files can be temporarily stored, and then sent via e-mail or be downloaded at once.

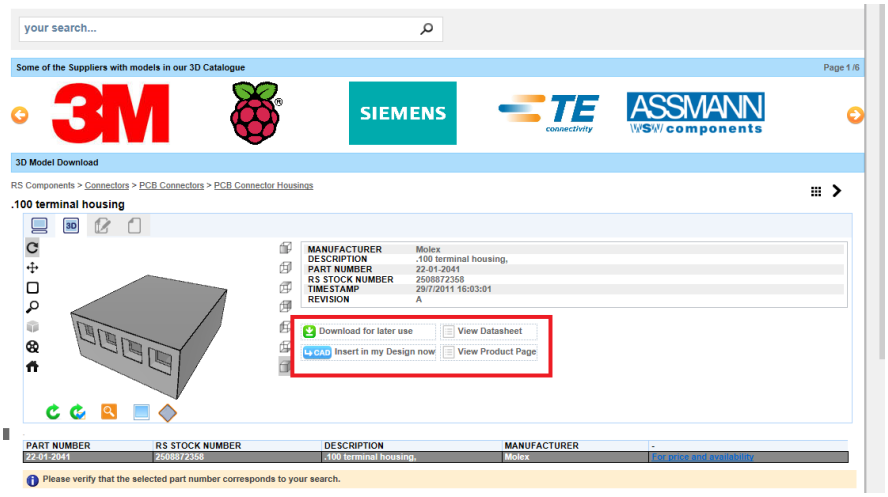

Fig. 4-3 Download file

If you want to import the image files that you downloaded to your computer already, you can click File in the Insert tool, click on the plane to locate the insert position, and after the file is located, you can insert the image. Note, that if you use this method to import the image, when this image file is modified, the original file will be modified as well. If you want to preserve the original file, I suggest you use copy and paste to import files.

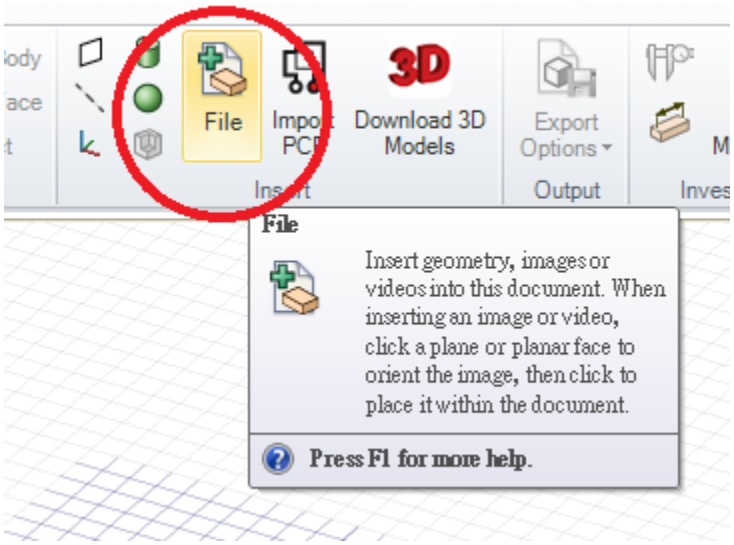

Fig. 4-4 Insert file

Furthermore, DesignSpark Mechanical also supports .skp a file drawn by Google SketchUp, this format is easily to search for in the forums, and can be imported into DesignSpark Mechanical for modification. However, because of the model feature of the .skp file, there may appear some facets in some of the section line's surface. If this problem occurs, you can click on DesignSpark Options under the File menu, find the SketchUp file option, and select Simplify t0 surface option, it should be able to solve the problem, but some models may be too complex to deal with after this option is selected.

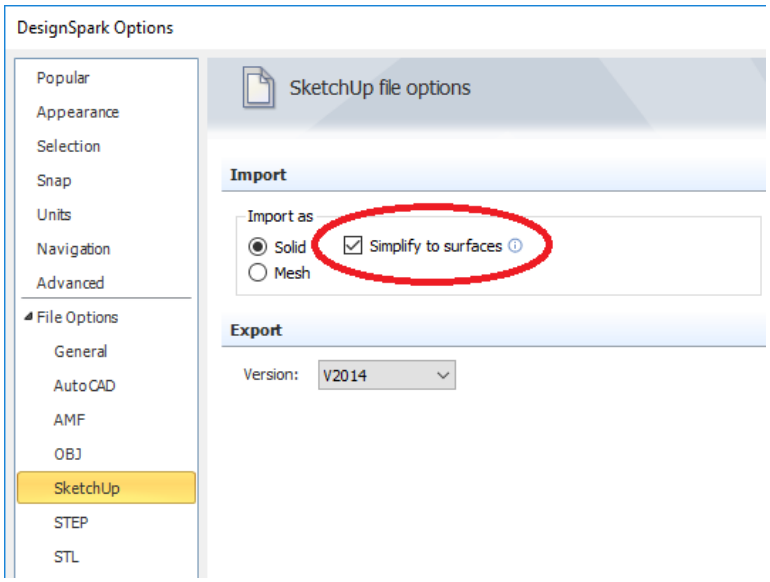

Fig. 4-5 .stl output file settings

Another common graphics file, .STEP file, can also be imported into DesignSpark Mechanical, but this format can only be read-only in DesignSpark Mechanical (the Add-On modules allows you to modify STEP files). In other words, you can import, save, or conFig. .STEP files, but the model cannot be modified in any way after import.

4-2 Combination Tools

Combination tool has two main functions:

1. Use the object shape to cut another object.

2. Merge adjacent or overlapping models into an object.

4-2-1 Cutting Object

In this section you will load an object from the online model library, and cut a graphic through this object, showing that the output of the graphics we draw in DesignSpark Mechanical, can be combined with real objects.

Step 1: Draw a box. First, please create a rectangle with a length x width x height of 100mm x 100mm x 3mm, you can refer to the Chapter 2: Basic Modeling of this book to draw this box.

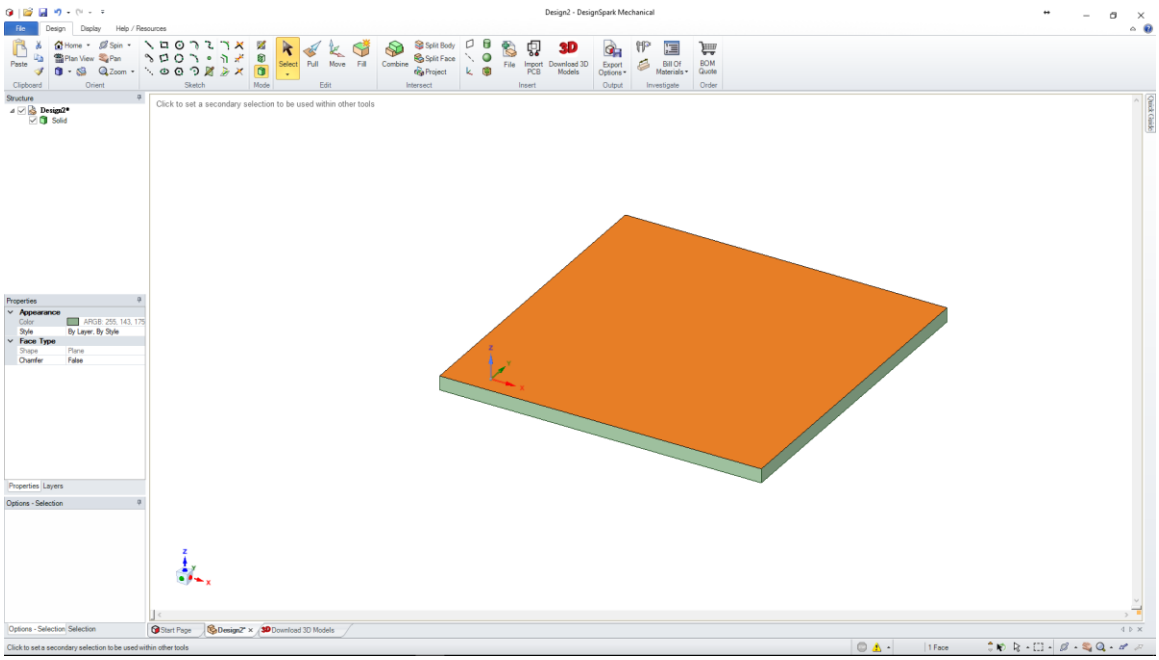

Fig. 4-6 Draw a box

Step 2: Import graphics. Click online model library function under Insert tool, enter "5427949" the RS stock number to find this example: 15 WAY PLUG. Alternatively, you can also use other graphics to do this exercise.

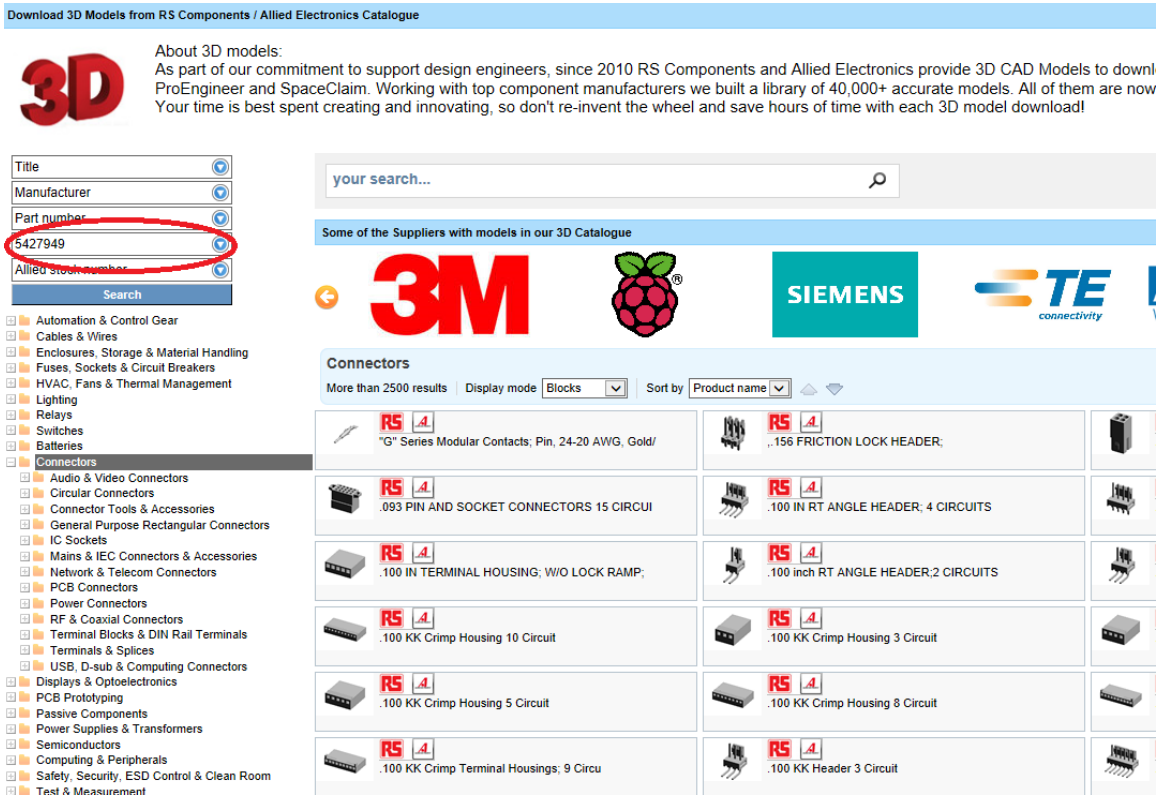

Fig. 4-7 Search part

When you open the 15 WAY PLUG graphics page, you can import it into DesignSpark Mechanical

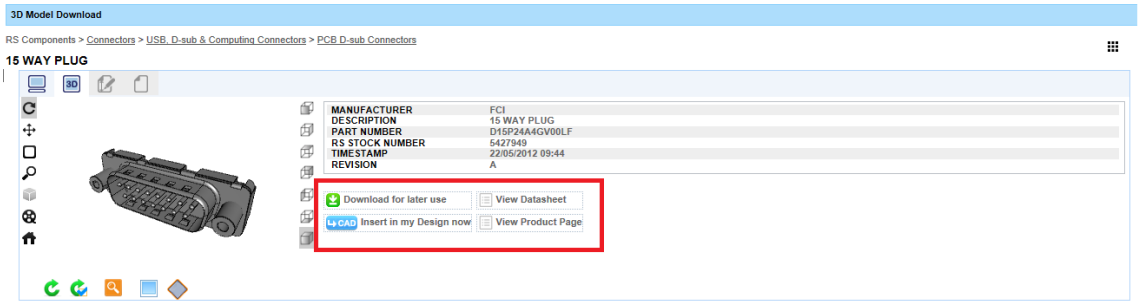

Fig. 4-8 Download part

Step 3: Adjust the position. Once you have imported the graphics, use Move in Edit tool, adjust the position and direction of the connector, put the connector above the box with the contacts facing down.

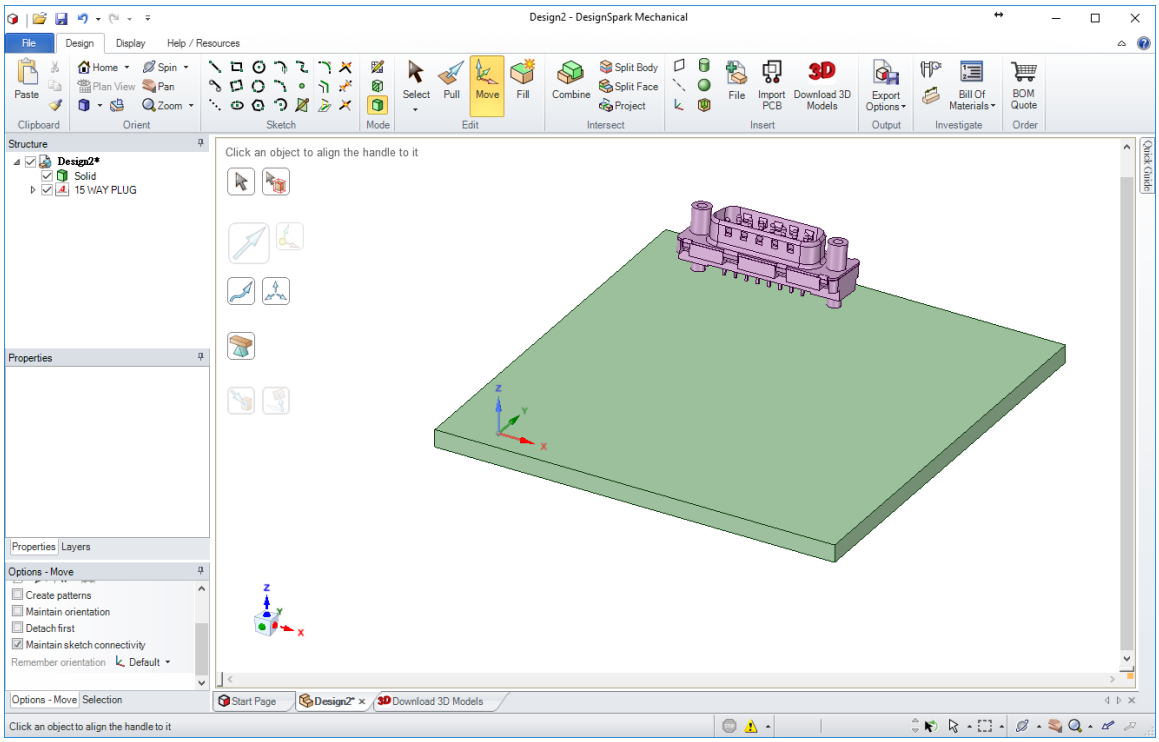

Fig. 4-9 Adjust position

Step 4: Anchoring. Triple click the connector to select it all, press the Z key to enlarge the selected item, select Move in Edit tool, and select Anchor in the left of design window, then you need to pick a plane as an anchor's (Moving anchor) datum plane, left click the inner surface of the connector (brown side in the Fig.), the moving anchor will be placed on that surface.

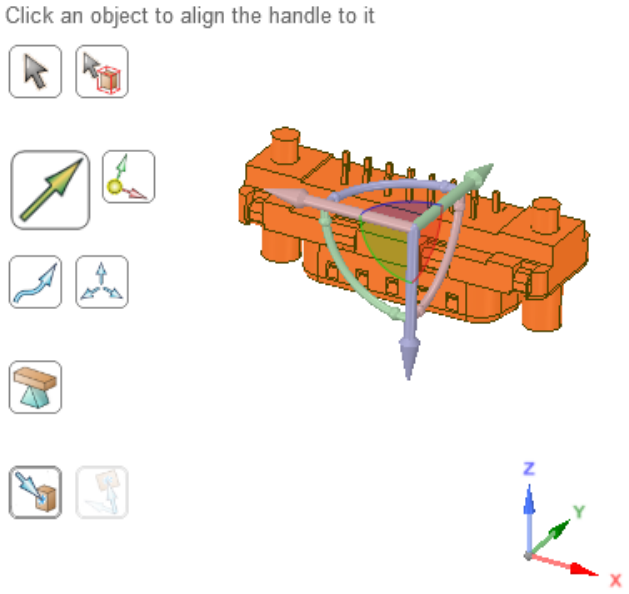

Fig. 4-10 Anchoring

Step 5: Align connector with the surface. After moving the moving anchor to the reference plane, click Goto on the left, and then click on the outer surface of the box. The connector's surface (datum plane) will move to this surface of the box, then click on the blank space to end the move operation.

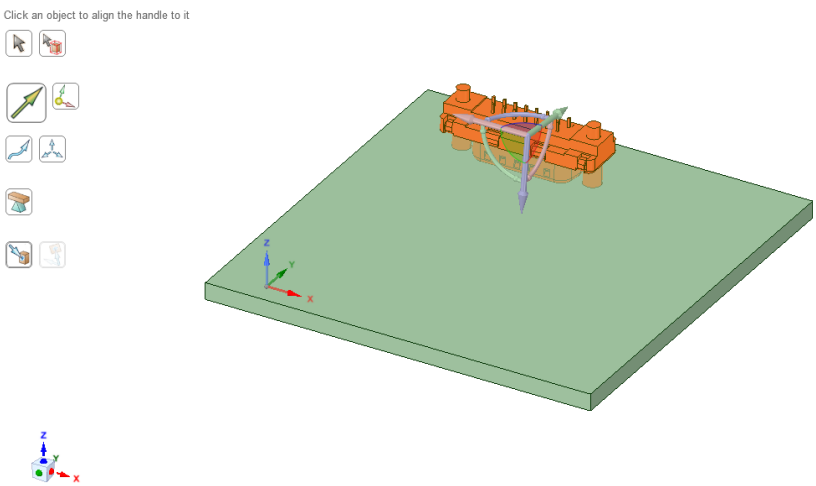

Fig. 4-11 Goto

Step 6: Cutting. Select Combine in Intersection tool, first click on the box as the target object, and then click the connector as a cutting object.

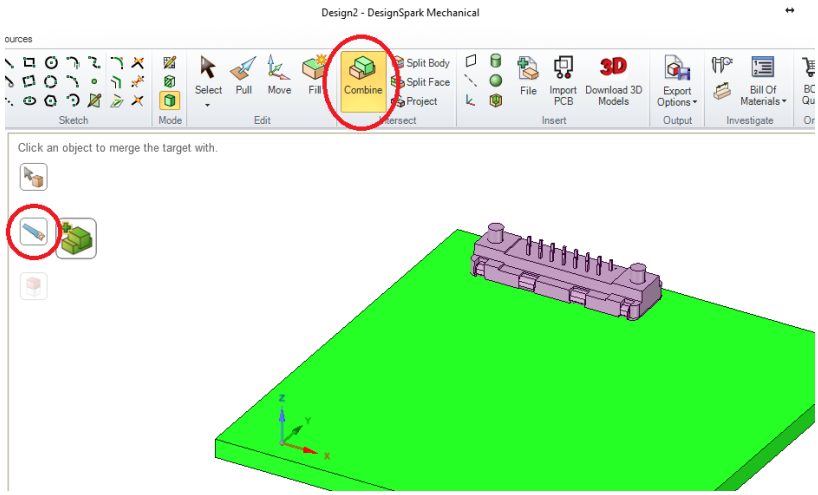

Fig. 4-12 Cutting (cutter)

Step 7: Confirm cutting. At this time the structure tree will have a lot more objects, which

are outer separators, they are new objects cut out of the connector. Please pull down the structure tree and uncheck object 7156, this object will be hidden on the screen, then we can confirm whether the box has cut out a shape of the connector or not.

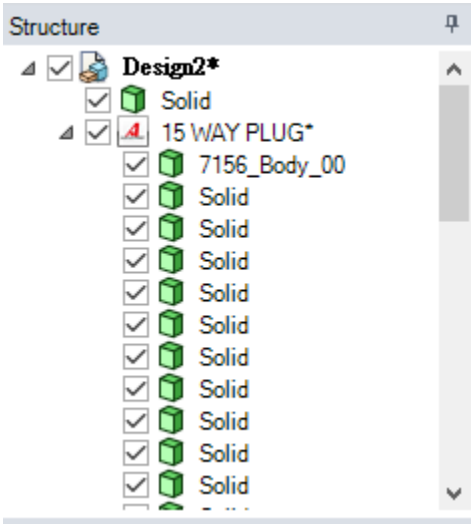

Fig. 4-13 Align plane

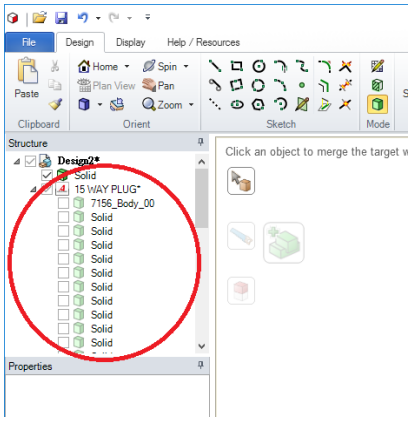

Fig. 4-14 Confirm cutting

Step 8: Delete graphics after cutting. In combination mode, pull out a selection box,

choose all the lines established by the connector housing and delete them, and you will find there are only two objects left in the structure tree, then you can click on any blank space to end the combining operation.

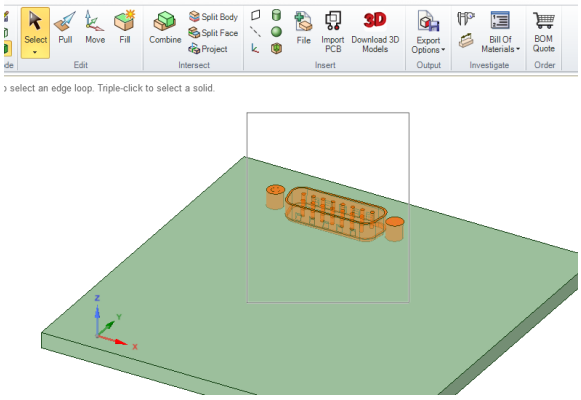

Fig. 4-15 Delete after cutting graphics

Step 9: Complete the cut. At this point you can see the box has been completed where the connector connected to the box through the Combination function, and we finally save this file as Cutting.

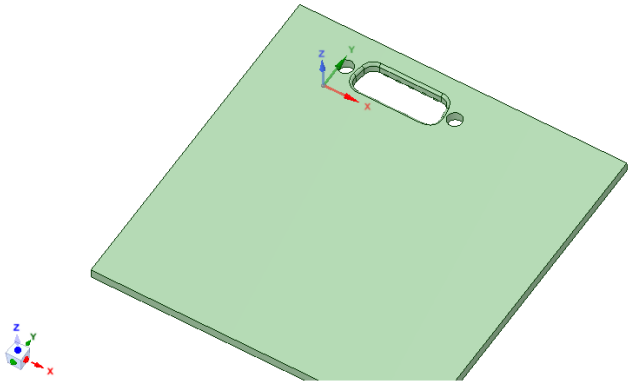

Fig. 4-16 Cutting completed

4-2-2 Combine Objects

This section will teach you how to write three-dimensional text in DesignSpark Mechanical, and we will combine the text and what we have drawn.

Step 1: Establish circle and mark size. First, create a new design (File >> New >> Design), and establish a circle of any radius. After the circle is drawn, click Dimensioning in Investigate, click edge or surface of the circle, and move the mouse to the outside of the circle, select an appropriate location, click the left mouse button, then the program will create an annotation plane, and the diameter of the circle will be marked on the annotation plane.

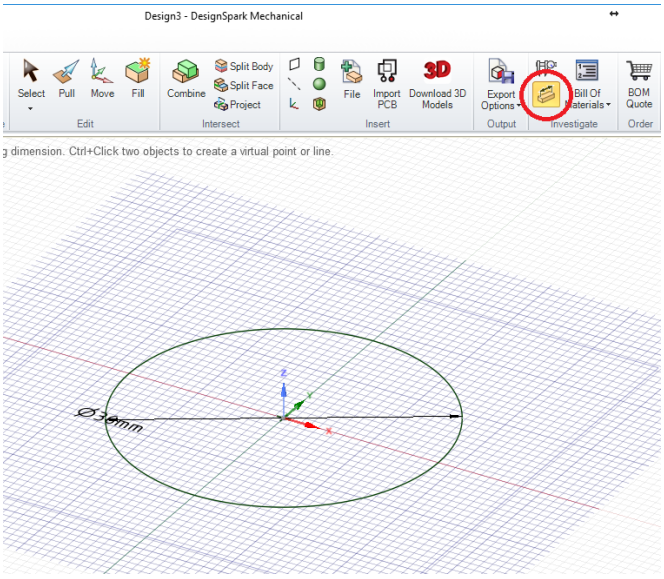

Fig. 4-17 Combine objects

Step 2: Edit Text. You want to edit the comment text into the text you want. Please click twice on the text , edit the text to what you want and highlight to choose this text, then the text edit tool will appear in the upper right corner of the text, zoom the text to appropriate size, then widen text box until you can display all of the text.

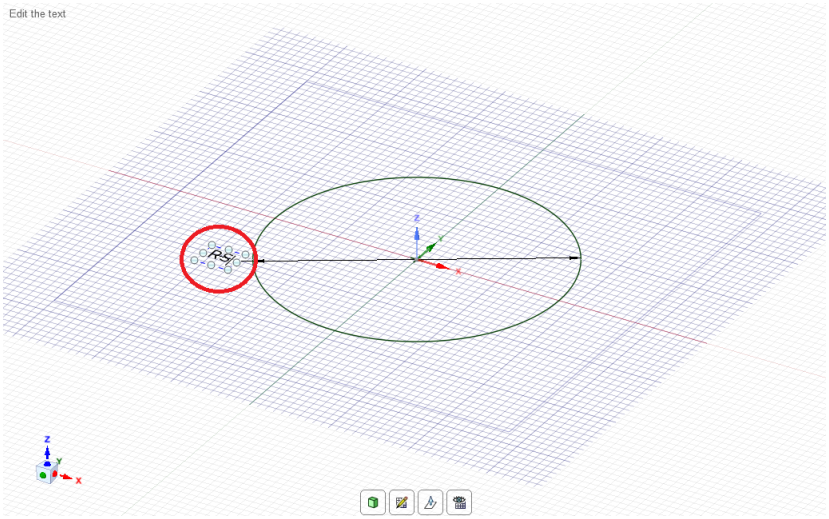

Fig. 4-18 Edit text

Step 3: Project onto the sketch. After the text editing is complete, we want to transfer the text into lines. Please highlight the text, and then click Project to Sketch in Sketch tool. It will project the selected edge or the annotation text onto the sketch (finish text editing before the projection, as it is more difficult to edit text beforehand).

You can see there are a number of lines added in the structure tree after projection, we can uncheck the annotation plan in order to confirm the projected results.

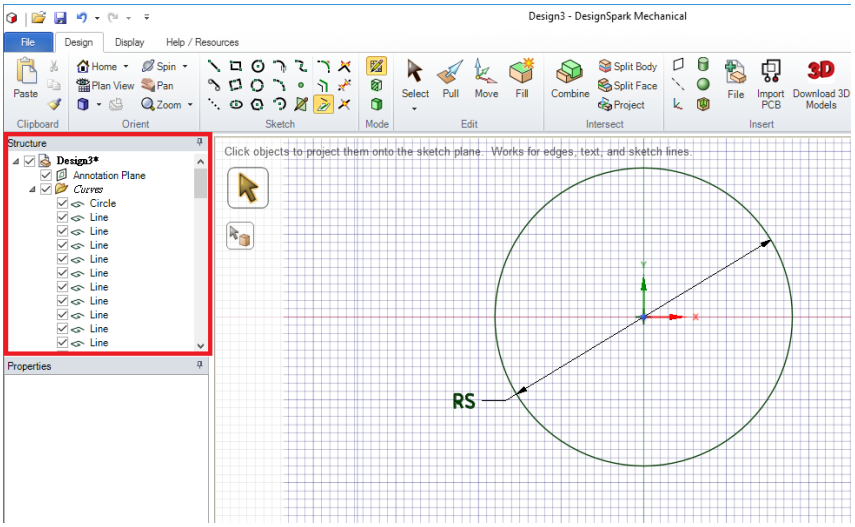

Fig. 4-19 Project to a sketch plane

Step 4: Generate font plane. This file aims to create text, so the previous aid tool can be deleted (circular lines and annotation plane). After deletion, we will mark all the words that are projected on the sketch, click Fill in Edit tool, then you can see that there is only one surface left in the tree structure, and the remaining objects such as lines are all gone.

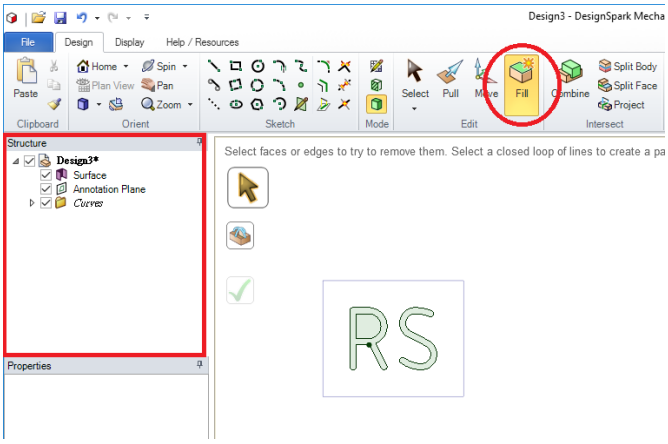

Fig. 4-20 Generate font plane

Step 5: Remove extra surface. There may have some unwanted planes, so we have to remove them before making a perspective view (as the font used in example is relatively simple, other fonts may produce more planes).

Fig. 4-21 Remove the extra surface

Step 6: Making a perspective view of font. Click Pull in Edit tool; pull up the plane 2mm. After pulling, the three-dimensional font is complete.

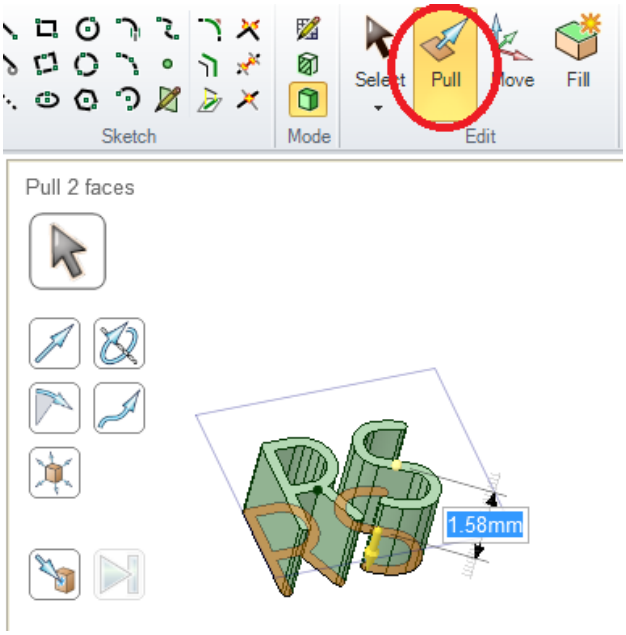

Fig. 4-22 Make a perspective view of font

Step 7: Import font into Cutting file. Open the Cutting file you created in 4-2-1, we can "Insert File", or use "copy, paste" to import file. Please adjust the text's location by using Move in Edit tool after import, so that we can facilitate the operation afterwards.

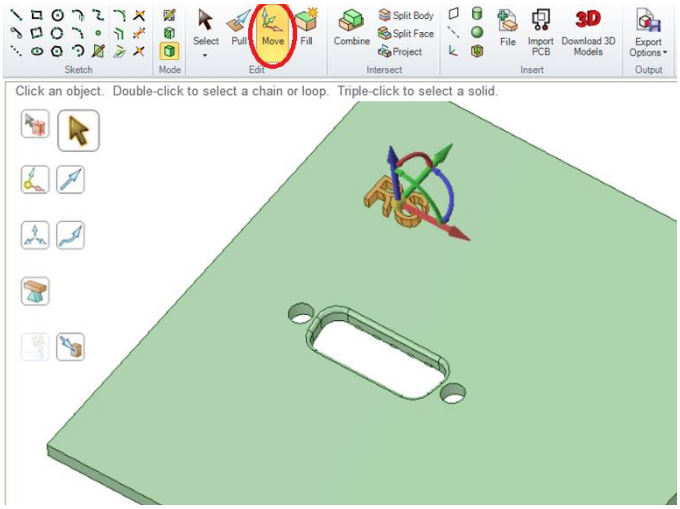

Fig. 4-23 Import the font to Cutting file

Step 8: Move the text to box surface. Next, we are going to connect the perspective view of the text and the box to each other, select all the text and click Move first, and click Anchor on the left, choose the surface that the text is going to connect with on the box, then the moving anchor will automatically move onto the selected surface.

Click Goto, and then click on the box surface, then the text will automatically move to the surface of the box, both of them will connect to each other.

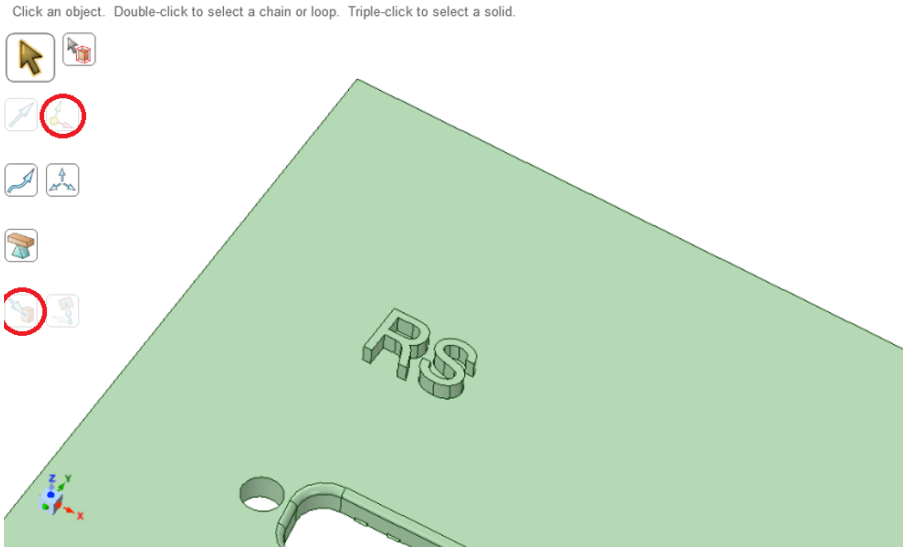

Fig. 4-24 Move the text to box surface

Step 9: Combine text and box. Finally, set the text and the box together, so that the two become one. Please click on the box, and then click Combine in Intersection tool. In addition to cutting that the previous section mentioned, you can also use Combine to combine two items, select the entities to merge on the left, then click two entities (you

can also click entities directly in the structure tree), the text and the box will be combined.

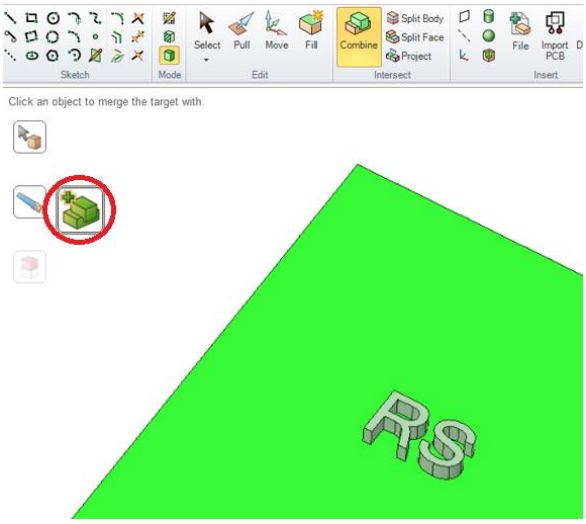

Fig. 4-25 Combine text and box

Step 10: Complete. Now, there are only two entities in the structure: the box and 7156, this is the completed version.

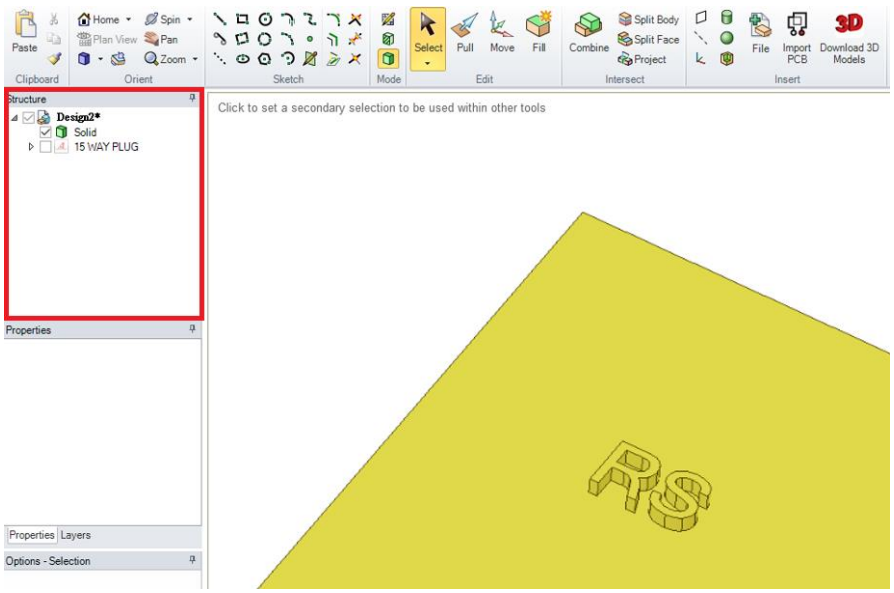

Fig. 4-26 Complete

4-3 Importing a PCB Design

In many cases, transferring the printed circuit board design into the mechanical CAD software is very useful, the most direct example includes checking if the circuit board size is consistent with the graphical shell size. You can use this to verify the position of the mounting holes, components, and configurations of connectors, or you can directly design a new case for your PCB.

4-3-1 Import DSPCB IDF File Format into DesignSpark Mechanical

DesignSpark Mechanical Import intermediary data format (IDF file). This format applies to all circuit board design software, of course, also included is DesignSpark PCB. Typically when exporting an ECAD software file, there will be two separate output files: 1. circuit board file (.idb); 2. Interface definition file (.idl), it is recommended that both files be placed in the same folder to ensure the data integrity.

After opening a new design, click File >> Open, select the pull-down menu at the bottom of the window to open "All Files (*. *)."

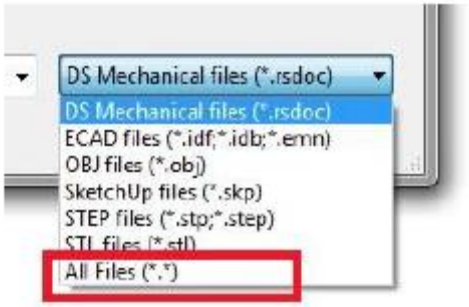

Fig. 4-27 Import files

Choose your .idb files, and import the schematics into DesignSpark Mechanical, this may take some time, which depends on the complexity of the schematic design and file size. The progress will be displayed at the bottom of the DesignSpark Mechanical window.

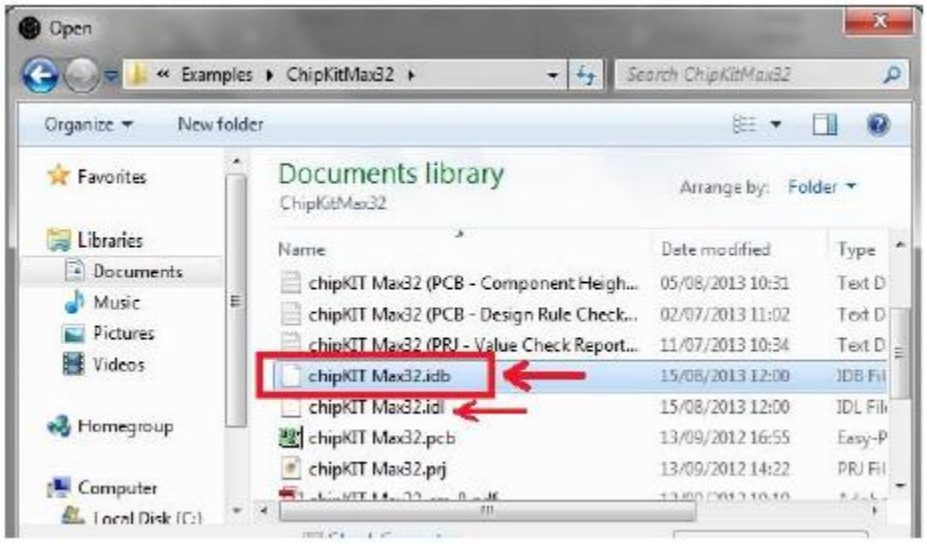

Fig. 4-28 Import .idb file

Now we have imported this circuit board into DesignSpark Mechanical, as shown in Fig. 4-29.

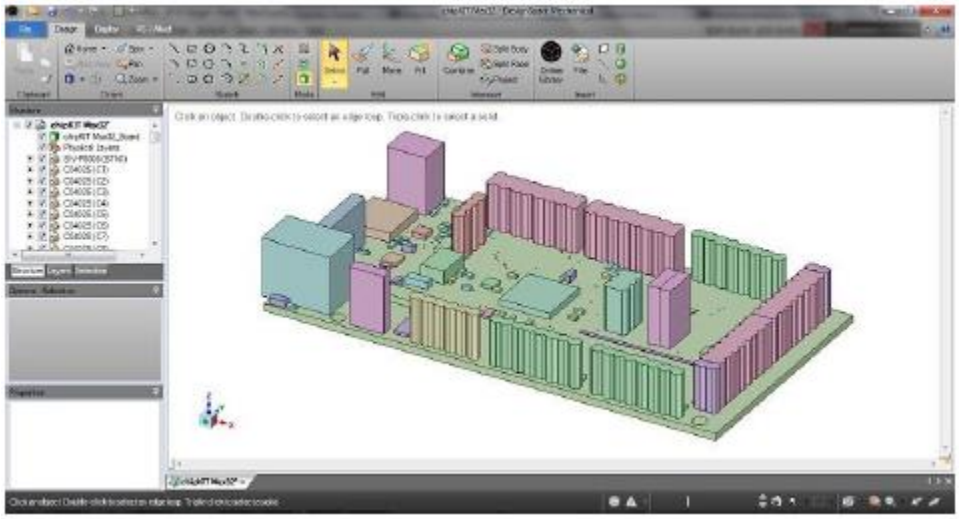

Fig. 4-29 Import complete

4-4 Summary

DesignSpark Mechanical provides a wealth of online resources with many electronic components. These elements do not need to be redrawn; you just need to find the appropriate elements and import them, which can save you a lot of effort in the development process, and also greatly enhance efficiency.

Chapter 5 Assembly, Quotes, Output

In the previous chapters we have learned basic drawing, importing files and other functions, this chapter will take you through designing a case which can hold electronic products inside, and you will learn how to purchase the materials you need online through the RS website.

5-1 Product Design

In this section you will design a customized shell, able to cope with an internal configuration of electronic products. Downloading electronic products from the website can dramatically save you design time.

Step 1: Establish a shell. First, create a cube with a length, width and height of 100mm x 100mm x 50mm, and then click Shell in Insert tool, which can remove elements of the drawing to create the shell, you can specify the thickness of the shell, so please click the command first and then click the surface to remove. Enter the shell thickness after removal.

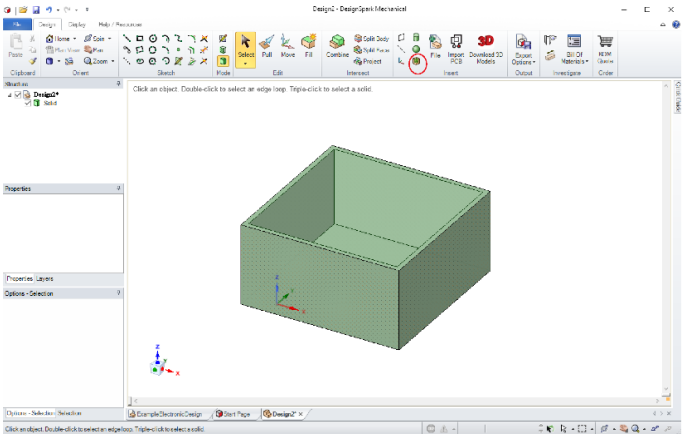

Figure 5-1 Establish a shell

Step 2: Import 3D models. After a rough shell is created, then we need to import the electronic products one by one (This example uses RS Stock numbers # 6741328P, # 140448, # 2740441, # 7763206), move them to the most appropriate location by using the Move function. In this case, if you discover that the shell is too small or too big, you can use Pull in the Edit tool to pull the surface along the direction of the arrows to widen or reduce the size.

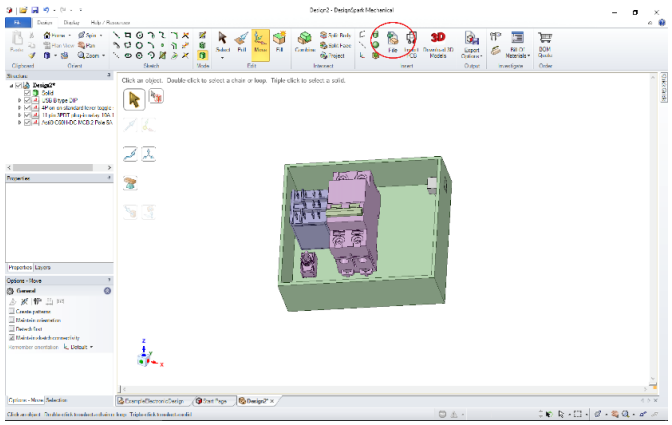

Figure 5-2 Import 3D models

Graphic 32096 in the example, if you want to use two graphic 32096s, then we can click on one of the 32096s and click Pull, we can see the mobility anchor on the graph, then click one direction of the mobility anchor and press the Ctrl key, now you can copy a new graphic along the pulling direction.

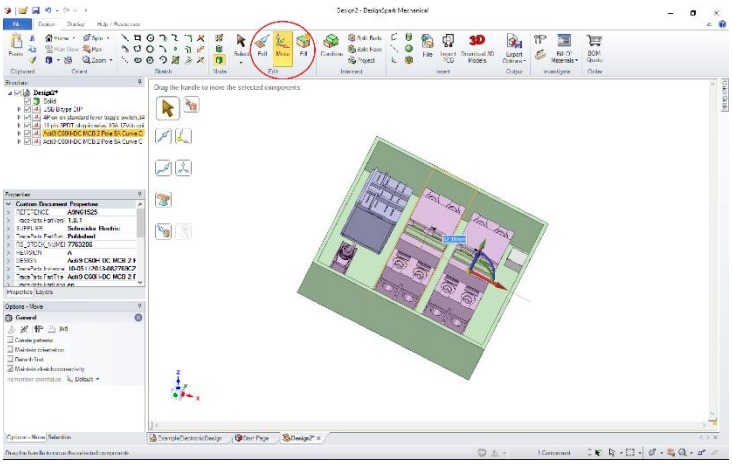

Figure 5-3 Duplicate the model

Step 3: Combine. Take the method in Chapter 4-2 in the book as reference, these electronic parts can be combined with the shell. The electronic parts need to use the function Combine to blend with part of the shell.

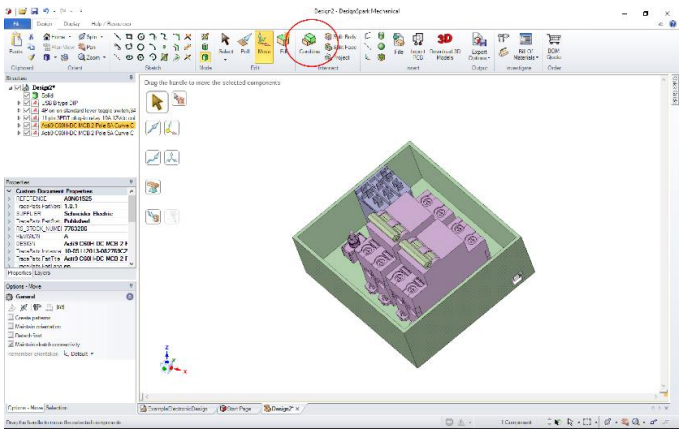

Figure 5-4 Combination

Step 4: Dimension. Dimensioning in the ‘Investigate’ tool can help us mark out dimensions easily.

Choose dimensioning, and click on any plane as annotation plane, the plane's intention is to let all dimension indications integrate on this plane. Or you can also establish multiple annotation planes, marking the size on several individual planes.

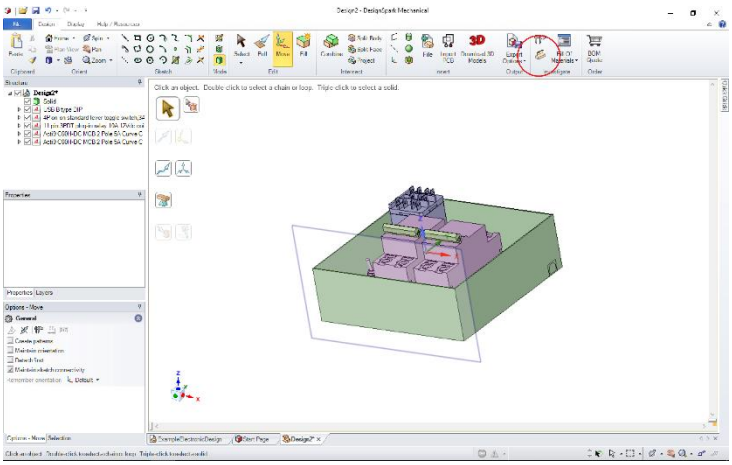

Figure 5-5 Generate annotation plane

After you have selected the plane and generated the annotation plane, we can change it to how you want. Left click on the specified edge and move outward, then there will be a mark moving with the cursor, then left click again in the required place, and then the size can be fixed.

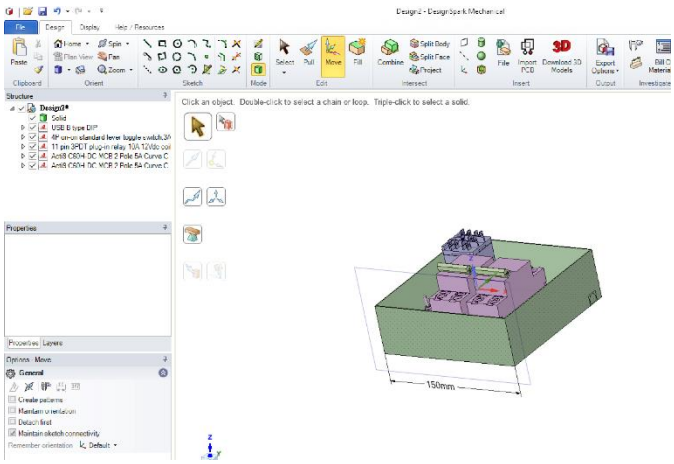

Figure 5-6 Select the place to mark

If you are not satisfied with the default font or size, etc., you can double-click on the font, and the font size modification window will appear.

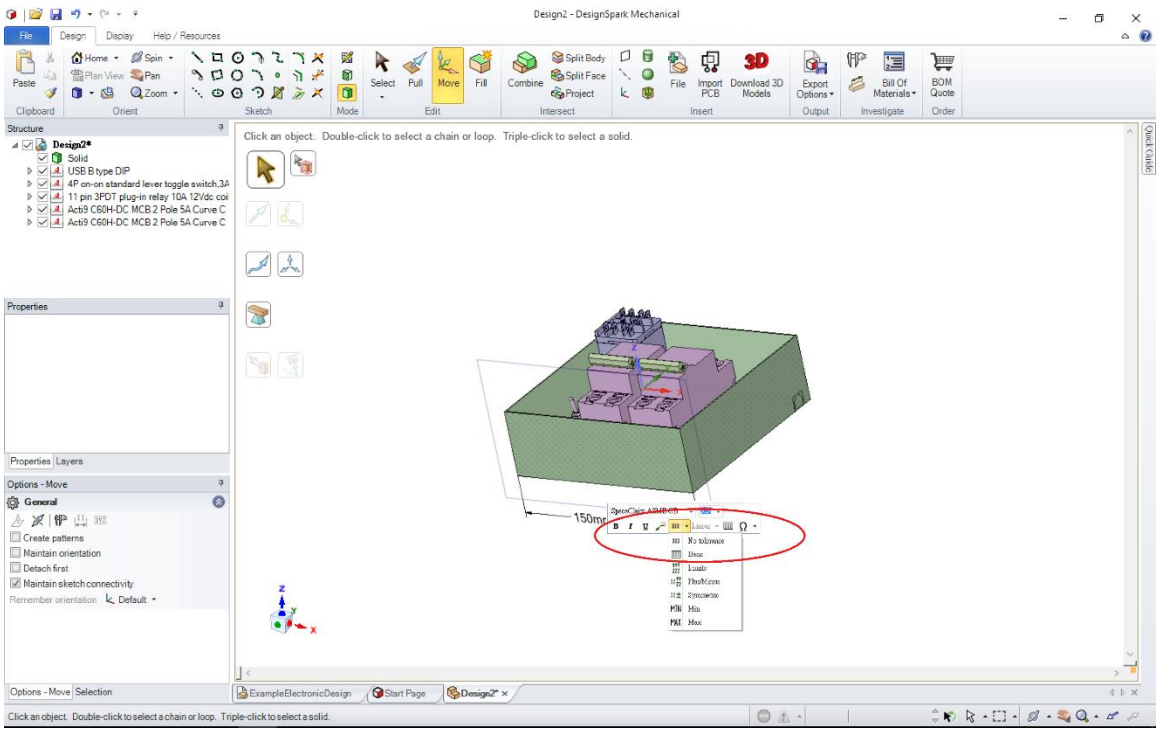

Figure 5-7 Modify the size

It is worth noting that, after the establishment of the size, dimensions are in accordance with the actual size and change when the graphics modify in the future.

Step5: Bill of Material. To be able to communicate with customers or others concerning the products component content, DesignSpark Mechanical provides a handy function: the bill of materials (BOM), which lists all the parts of the product design.

Click on the full details under the options in the Survey tool. And select an annotation plane in the drawing, then place the material and quote from anywhere on the plane.

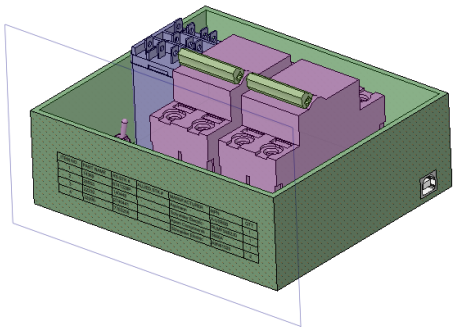

Figure 5-8 Place the bill of materials on specified plane

5-2 Bill of Materials Interface (BOM)

In the 5-1, we will use the bill of materials function, which is a convenient and time-saving tool that automatically lists all of the materials, but whether this list of materials is correct is the most important factor. In order to accomplish this task, there are several points we need to be aware of.

1. The list comes from the bill of materials (BOM) based on the content of the design structure tree.

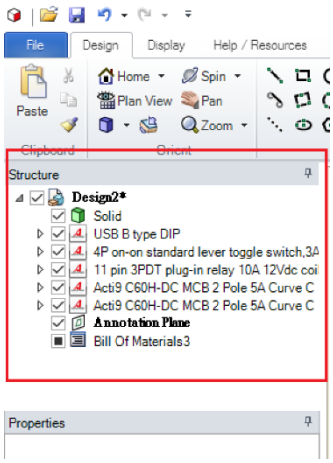

Figure 5-9 Bill of materials (BOM)

2. Each element in the structure tree should contain some entity structure, which will be displayed when you click on "+".

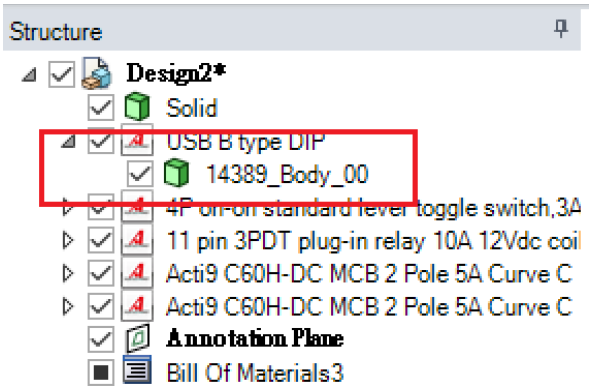

Figure 5-10 Structure tree

If nothing shows when you click on "+", then this is an empty component, it will appear in the bill of materials even if it is not in the drawing. The reason why the empty

components exist is that, whenever a component is deleted in the design window after it has been clicked three times, it will disappear from the drawing, but is still present in the structure tree as an empty member. If you are unsure, please right click on the design file name at the top of the structure tree before the establishment of the BOM and click Delete Empty Components.

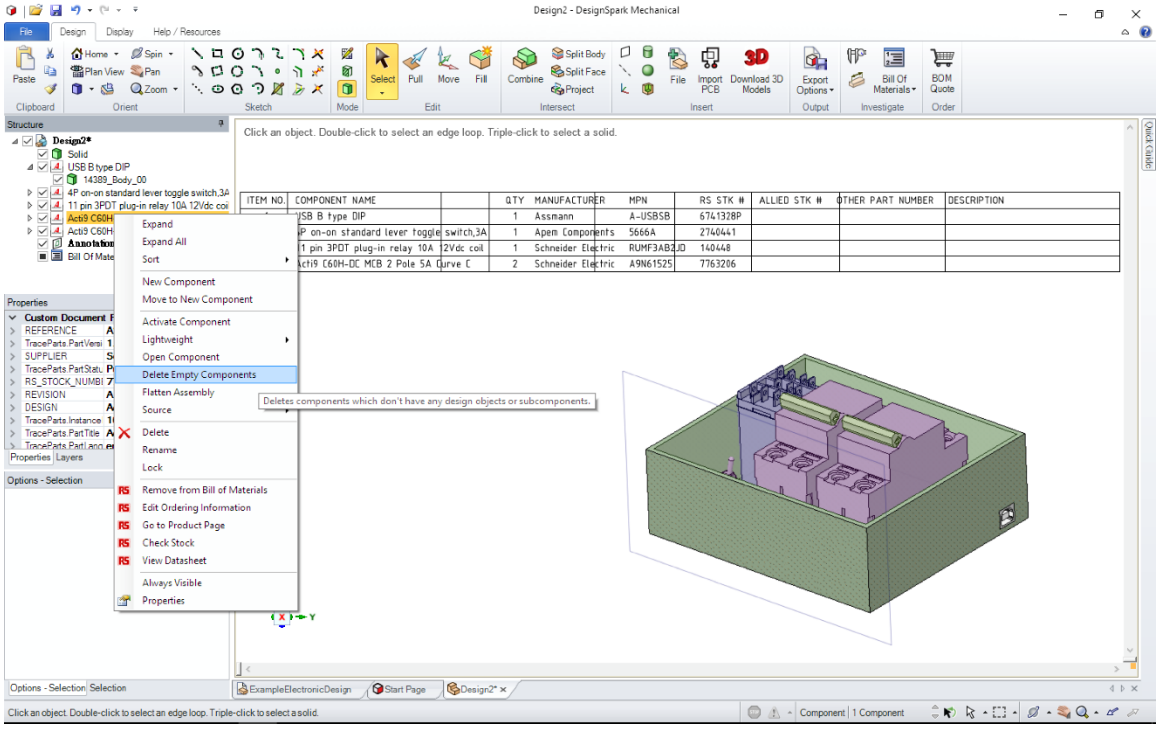

Figure 5-11 Delete empty components

3. The component may have been added to the structure tree (appears on the BOM), but does not appear in the model. For example, you might want to add cable ties, thermal covers or other consumables. To add such components right click the design file name at the top of the structure tree, and then click New Component.

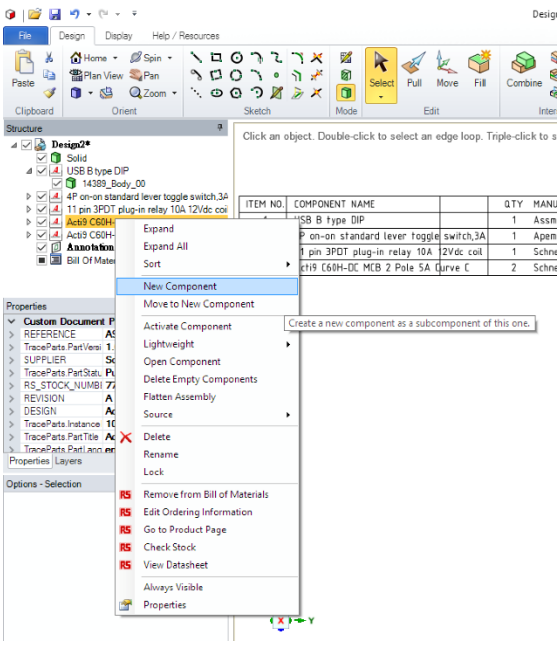

Figure 5-12 Click new component

Then give this new component a new name, it will immediately appear in the BOM, but will not appear in the model yet. After generating a correct BOM table, you can click on the BOM function in the Subscription tool, get a quote and order parts directly from the RS website.

5-3 Export Format Description

After the drawing is done, you often need to output different file types, such a format that you can input to the 3D printer to print out the project, or a format that other graphics software supports. This section will tell you how to manage the output files and the key to exporting different file formats successfully.

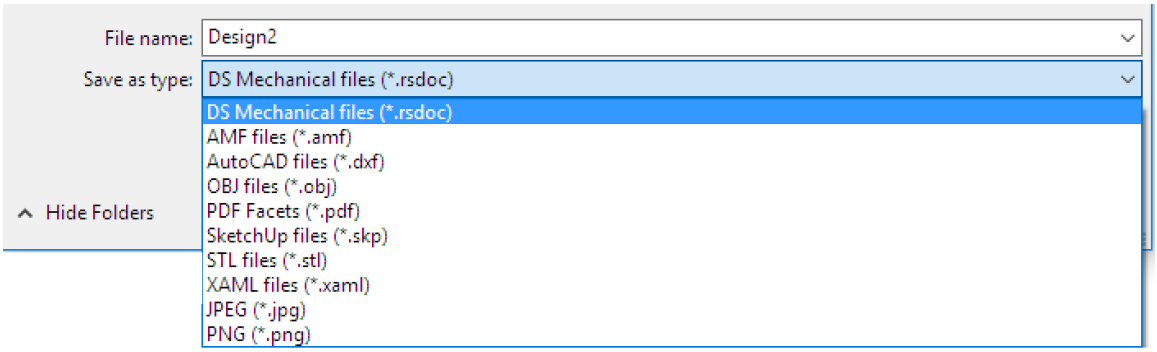

Figure 5-13 Export format

The graphics drawn through DesignSpark Mechanical have many available file formats for saving the file, which can be found in menu under File >> Save as >> Save as Type.

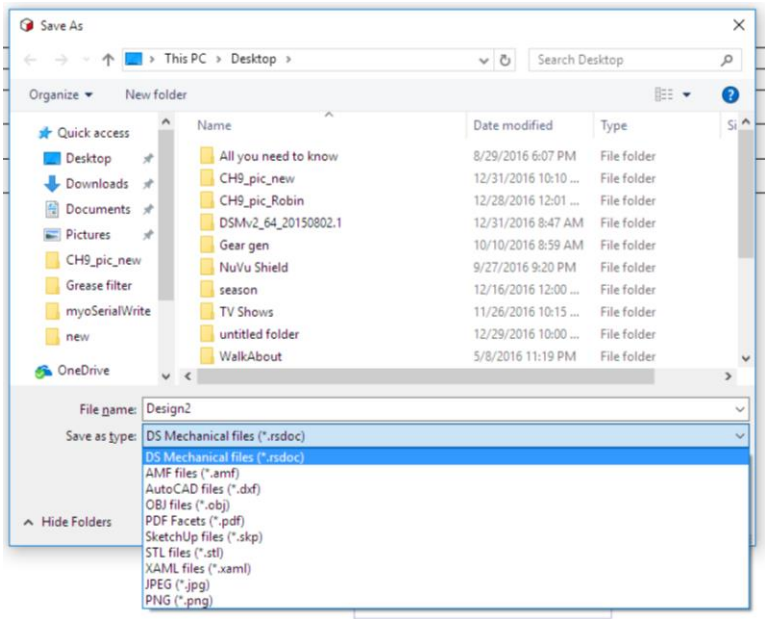

Figure 5-14 Select save as type

The default output format of DesignSpark Mechanical is .rsdoc file. We can also choose other popular graphic formats.

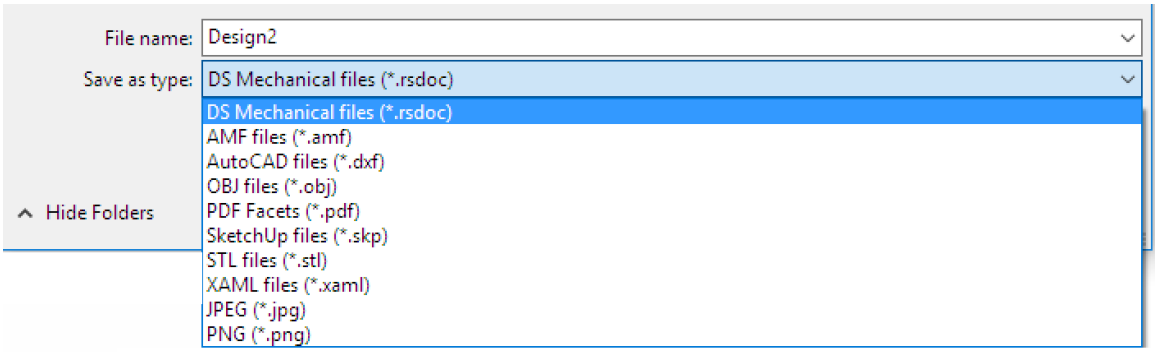

Figure 5-15 The.rsdoc profile

Outputting in different formats is less prone to errors, but in some cases, some small problems are still inevitable, and you can try the following methods to improve them. When you export .dxf file, the view of the model will be converted, saved as a new format and fixed in its direction. Therefore, ensure you've chosen the correct view and perspective you want before it is saved as a new file.

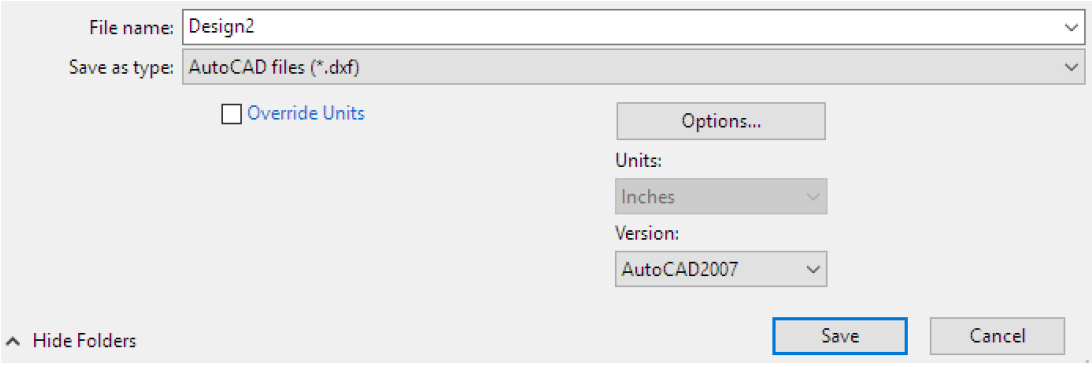

Figure 5-16 Export .dxf file

The resolution setting is set according to the original file when you save a .stl or .obj files. High resolution will improve the quality of output, but the resulting file may be

very large. In general, the resolution is usually sufficient when set to “fine”, if this setting is not suitable for the file, you can try other options, the key is to balance output quality and file size.

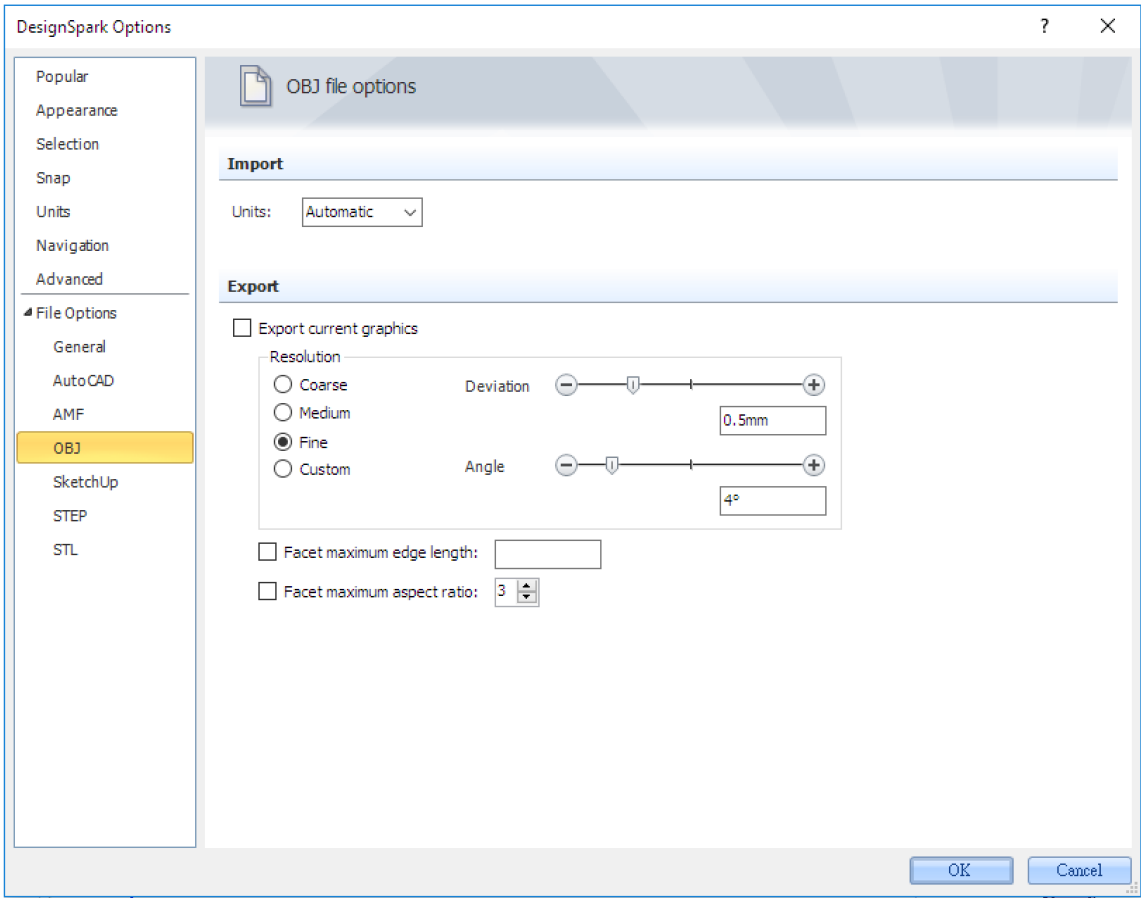

Figure 5-17 Setting parameters for export .obj file

5-4 Summary

DesignSpark Mechanical provides a very convenient online quote and shipping service which is definitely good news for us. You can select the components you want online,

without re-drawing, and also directly adjust the size of components, number. Price and other information that can be integrated in the chart to help us to perform the assessment and communicate with other developers.

As for compatibility issues, DesignSpark Mechanical is pretty powerful, so we do not need to worry about the exported files failing to open in other software.

This book has now come to an end, please practice some more, and use this software to complete your next design project.