

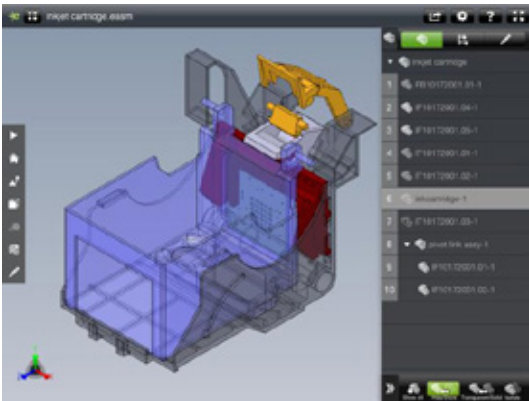


USING eDRAWINGS®

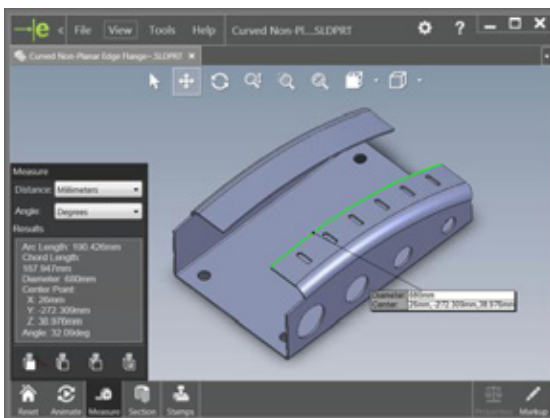


Overview

SolidWorks is one of the main design tools used by Cambridge Design Technology. A very powerful communication tool available to us, allows the ability to share our design output with our clients allowing you to view, spin, zoom, hide/show components and assemblies and measure the 3D CAD models created by us.



This tool is eDrawings®. This is a free download provided by SolidWorks. This gives you the ability to interact with the designs created for you and allows you to immerse yourself into the designs.



You can also open different types of files. You will also be able to open files created in Creo (Pro/ENGINEER), CATIA, Autodesk Inventor, and also manufacturing files like 3D printer “STL” files, 2D DXF/DWG drawing files, and 3D manufacturing formats, “STEP” and “IGES”.

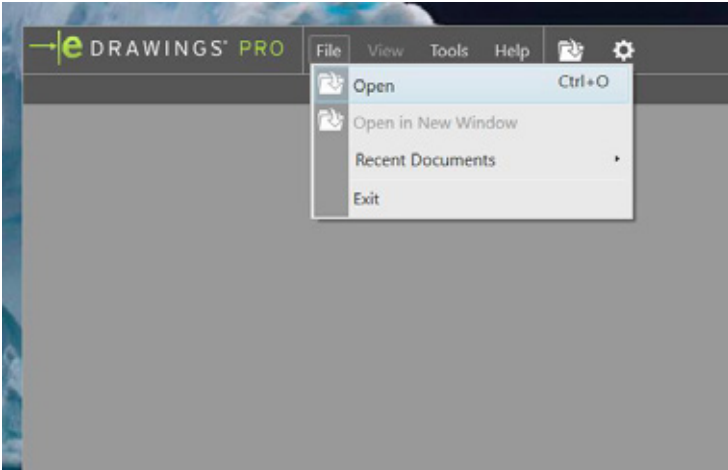
eDrawings® is available for PC and Mac iOS/Android for tablets.

If you don't have access to a PC loaded with eDrawings®, don't worry, we can send you a self-extracting file to help you. All you will need is internet access and we'll do the rest for you.

The following pages walk you through the basics of using eDrawings®

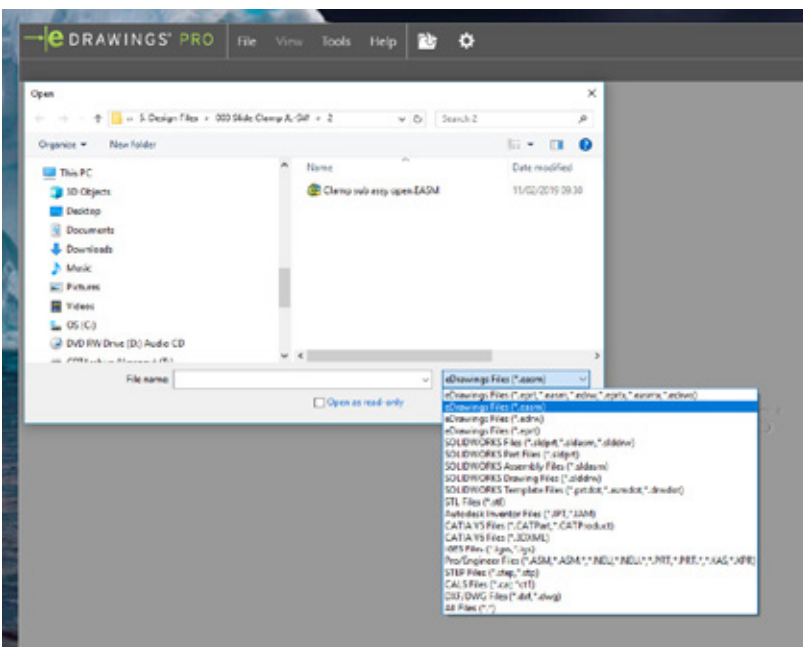
Using eDrawings

To open up a file, go to the <File> menu and click on open.



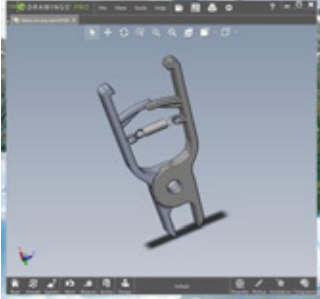
Then using the standard windows dialogue box, browse and click on the file you wish to open.

You can choose which file type to open, but predominantly CDT will either send you an eDrawings file (.eprt .edrw .easm) or the native SolidWorks files (.sldprt .slddrw .sldasm).




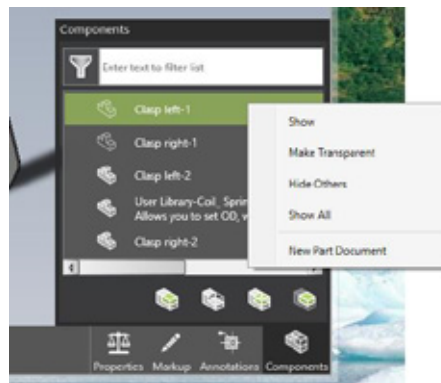
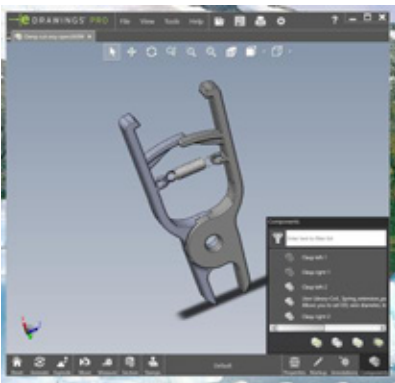
Double click on the file type you require. In this example we are using an eDrawings assembly (.easm).

The assembly will open and display like this:



Model Tree

Click on the components button  and the model tree, showing all the parts in the assembly, will appear. Note two greyed out icons – this denotes two hidden components.



If you click on a component in the model tree, then using the right mouse button, and click (known as a right click), you will see you can do lots of things.

Show – shows the hidden component

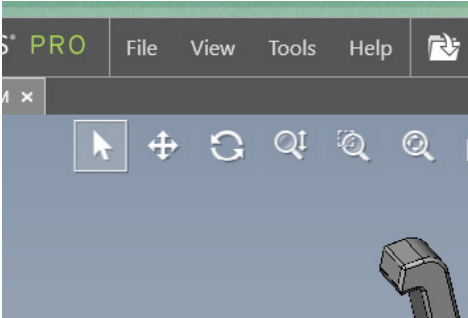
Make Transparent – makes the part transparent

Hide Others – shows the hidden component and hides all others so you can view a part in isolation

Show All – shows everything in the model tree

Once you have shown and hidden the components you want to view lets go onto the next stage.

Manipulating the model on-screen



The model manipulation icons work thus:



Pan – move the model up down, left right on the screen by moving the mouse u-d-l-r



Spin – spin the model around, by just moving the mouse around the screen



Zoom – up and down by moving the mouse up and down



Zoom Window – click the mouse to make a box around the zoom view you desire



Zoom All – click this to get the whole model reset back to the centre of the screen

If you are feeling adventurous, click on the  icon to deselect the any of the above commands,

- use the mouse wheel to zoom up and down
- click down and hold the middle mouse button to spin

Now we will go onto viewing the model in different modes.

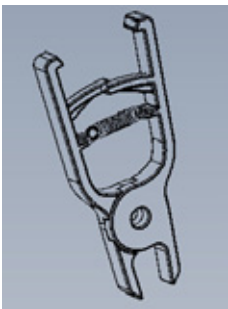
Viewing Modes

You can view the model in shaded mode, wireframe mode or shaded mode with edges highlighted in black. This last view mode is the default.

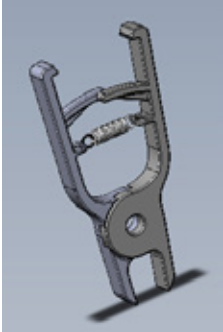
To change the view mode select this icon and click on the arrow next to it.



This icon is wireframe and will your model look like this:



This icon will shade you model with black highlighted edges like this:



This icon will shade the model and remove the highlighted edges:



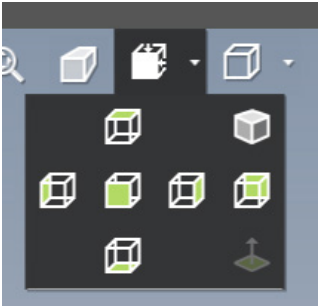
Now we look at the model view functions.



Model View Functions

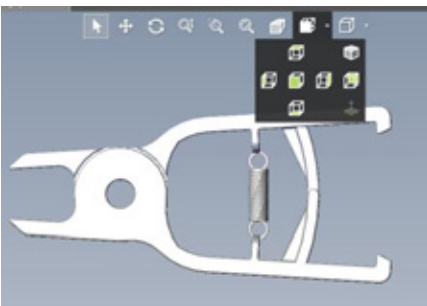
To view the model in any “head on view” click the arrow next to this icon




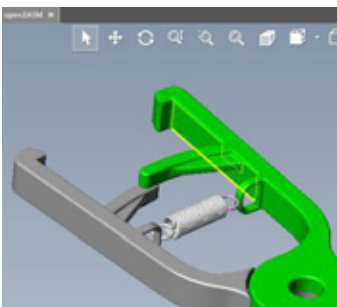
This icon will reveal the size faces front, back, top, bottom, left, right.



There is also an isometric button  to view the model in a default 3D orientation and a view normal to plan icon  meaning you can click on any planar face and view the selected plane head on.



You can select a flat (planar) surface by moving the mouse over the selected surface and clicking. The part of which that surface belongs, turns green, and the selected surface will be highlighted in yellow. Then click the  icon



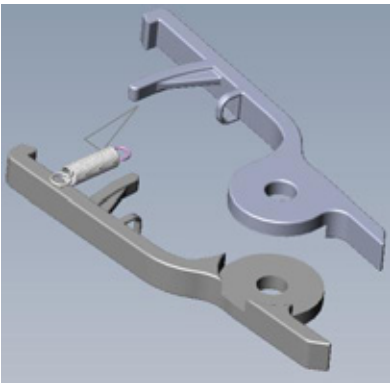
Now we will look at some of the analysis function.

Analysis



Animate – this automatically animated the model into each of the six orthogonal views, F, B, T, B, L & R. Try it to see the effect. It basically is an automation of the steps on the previous page.

Explode – explodes the parts apart. This is useful to allow you to see the build-up of an assembly.

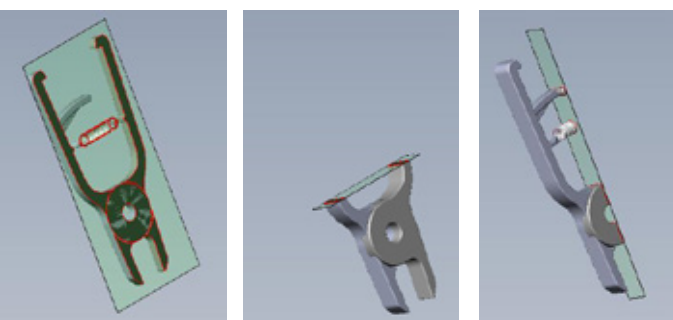
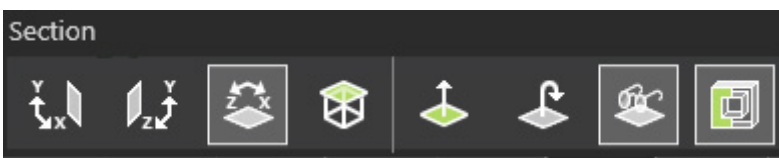


Move – by clicking and holding on a component, this allows you to drag one component out of the way to see where it assembles in to. You move by dragging, by moving in XYZ, translating from a point to point or by spinning it out of the way. To reset any move operation, explode, then unexplode the model by pressing on the explode button.



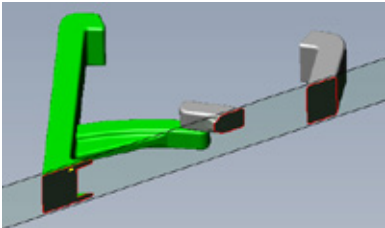
Measure – we will address this in the next section

Section – by clicking on the various icons you can section in various ways

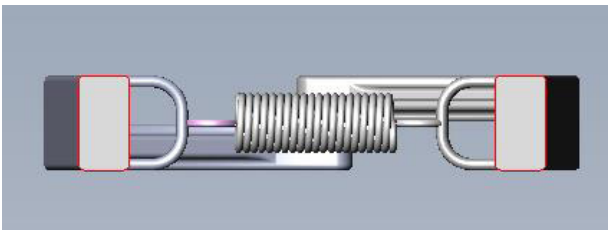


With the other icons, you can:

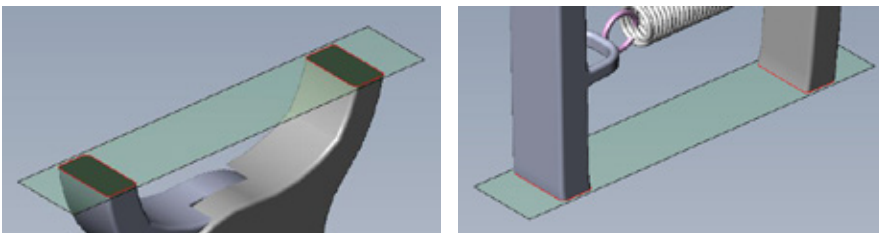
- select the plane on the model that you want to cut through



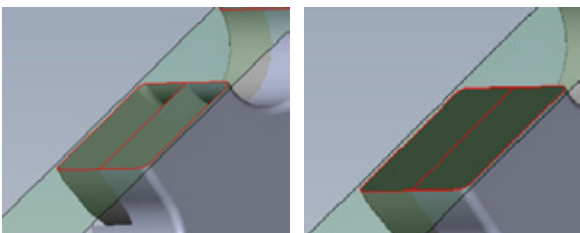
- view normal (face-on) to the section plane



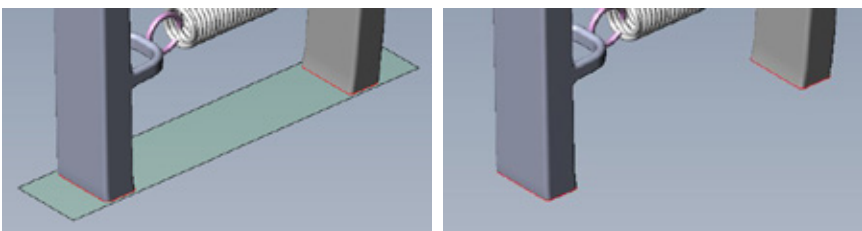
- flip the direction of viewing each section




- cap or uncap each section



- hide the section cutting plane



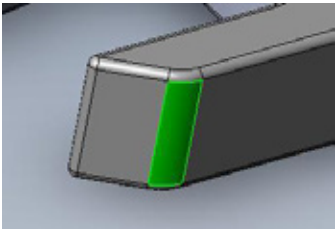
Play with each of these buttons to see how the various controls allow you to view the model in various ways. To cancel any of these, just press the section button  to exit Section mode.

Let's learn how to measure.

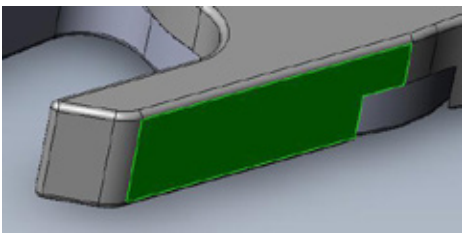
Measuring

First we need to understand any geometry in eDrawings. Each model is made of....

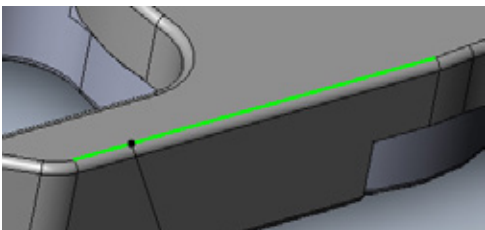
- Surfaces (non planar or curved)



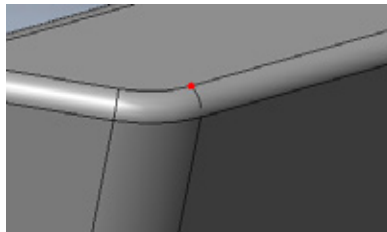
- Planar surfaces (flat)



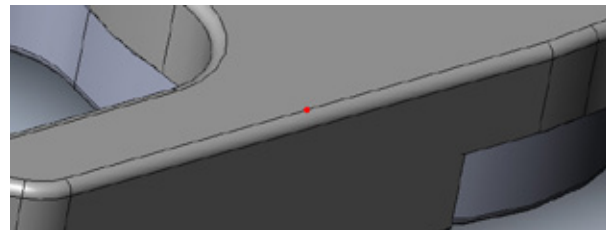
- Each surface has edges




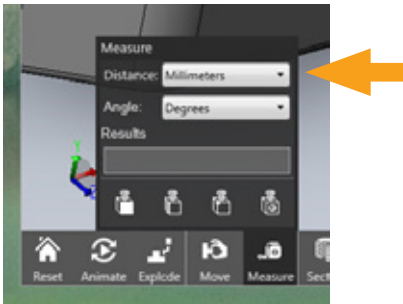
- These edges have two end points (singularly know as a vertex)



- And a middle point (mid-point).

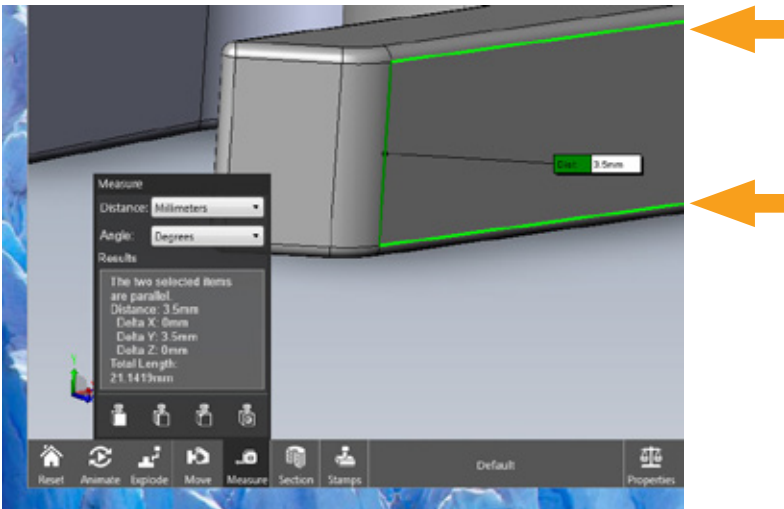


Click on measure button  and the measure function appears.



If you wish to measure distance in other units, drop the menu down by clicking on the black triangle to the right of the units to select mm, cm, m, ft, in, or ft'in" and degrees or radians.

To be able to measure you will select any combination of planar surfaces, edges and vertices and midpoints. As you select each entity the results box will guide you. Once you have selected a pair of valid entities it will display the results. To deselect, click anywhere on the background and reselect.




Experiment, clicking between pairs of planes, vertices, edges or any combination of these, and see what results you get. Note if you select two planes which are not parallel the result will display an angle between the two planes.

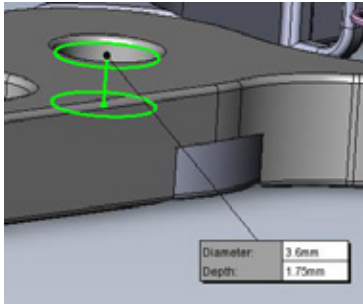
If you want to only select a vertex, edge or plane, then you can use the three filter icons



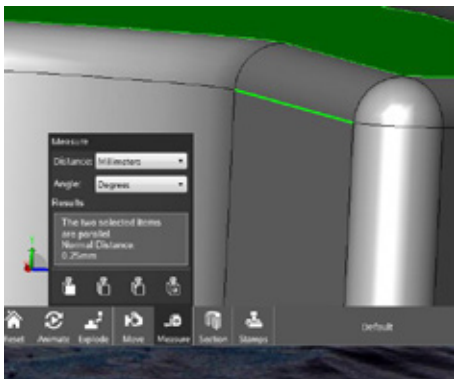
to exclusively select either a plane, edge or vertex.


To measure a hole, click this icon  and click on the circular surface of the hole.

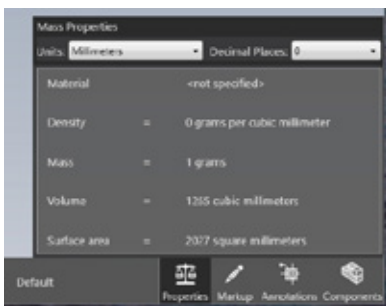
The results will display the hole diameter and the depth of that circular surface.



To measure a radius is a limited function, and is a work around, only allowing accurate measurement between faces which are perpendicular (inside or outside). To measure this radius, we will have to measure the opposite edge and planar face.



The final analysis tool is the properties function.  This allows surface area mass and volume to be specified, however, the mass can only be calculated if the material properties have been applied to the model by the designer at the SolidWorks level.



This is not done by default, but if you decide you want to calculate mass, call **01223 662300** and we will help you with this, apply the materials to the model and supply a new eDrawing file.

The final part to the app is mark up, add annotations and put stamps on the file.

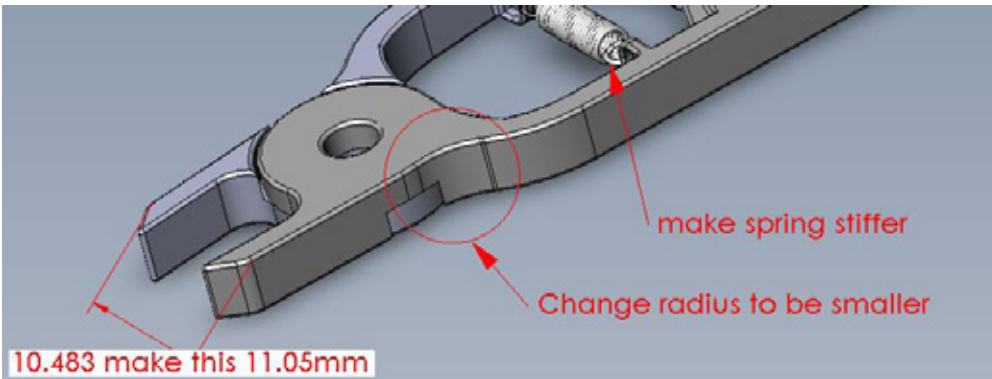


Mark-up allows you to sketch your own notes on the model.

Using the shape, text and dimension notations



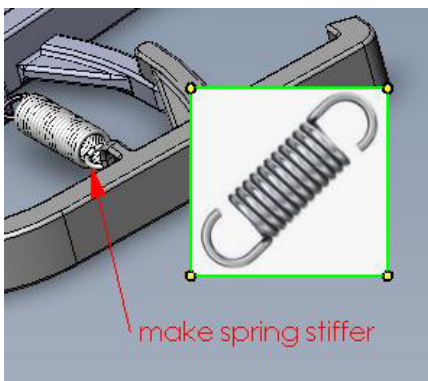
you can do the following:



Using the insert picture



you can add images to supplement the mark-ups.




Using these two buttons

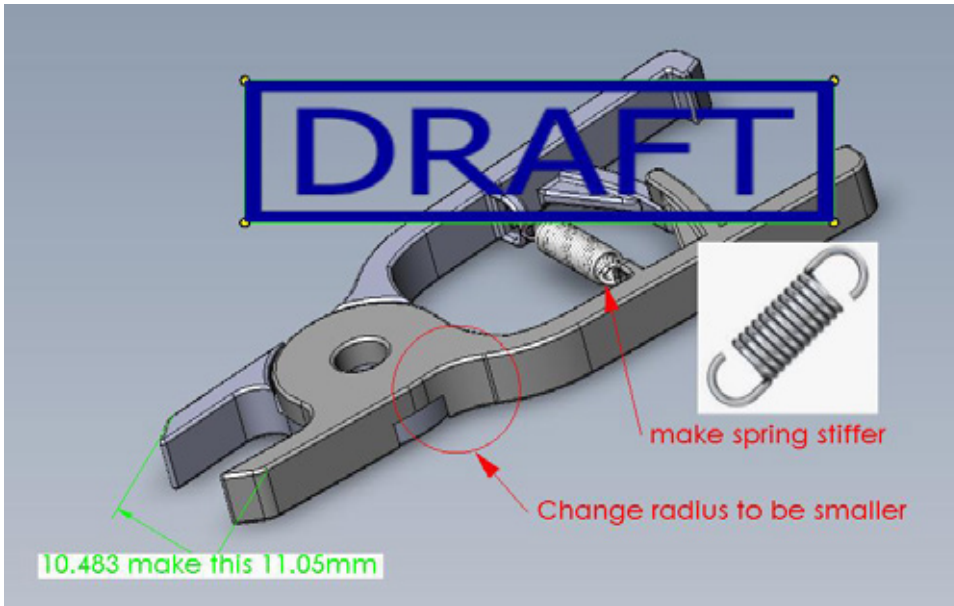


you can:

- save your mark-ups
- change settings giving you control of
- system, background and mark-up colours
- line weights
- contact details of the person making the mark ups

Finally when you have finished reviewing and want to watermark the eDrawing for archive records, passing to another party or department or returning to the designer for modification, you will need to click the  icon.

Select the appropriate stamp, and plant on to the drawing. You can move and resize accordingly.



Once you have finished with the eDrawing, click on save, print or open a new eDrawing to review.



Remember, this example is for an eDrawing assembly, but you can open up an eDrawing part, eDrawing drawing or a SolidWorks part, assembly or drawing.

Congratulations!! Now you can use eDrawings® you will be able to review, understand and comment on the design files created by Cambridge Design Technology at your own desk or on your own tablet.

We hope this document has given you enough information to start using eDrawings®.

If you have any queries or problems, do not hesitate to call us on **01223 662300** or email us at **info@cambridge-dt.com** or visit **www.cambridge-dt.com** for more information on how to use eDrawings®.

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