
Visual Bookmarks and Features Control

This Web training task introduces the Visual Bookmarks feature. This exercise will focus on how Visual Bookmarks can be useful in controlling the visibility of different features on a part in thinkdesign.

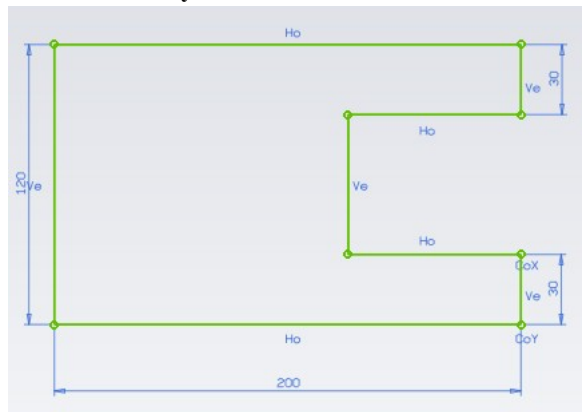
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1. Step1 : Prepare to Create a Visual Bookmark

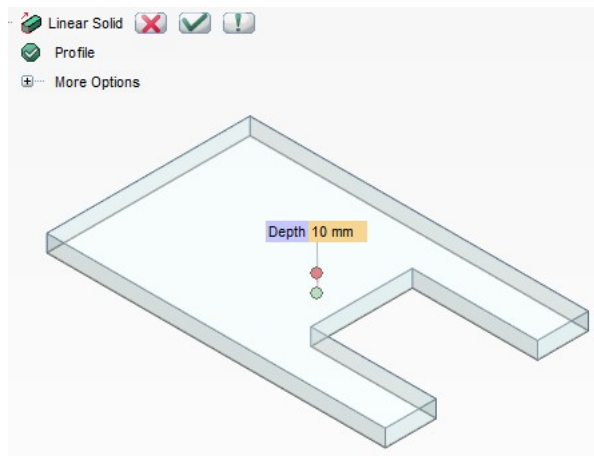
Let's start by building a simple 3D Model.

- Open a **New Model**
- Click the 2D Profile tab on the bottom left of thinkdesign window
- Start the **Polyline** command and sketch a Profile as shown below




Let's now Sweep this profile.

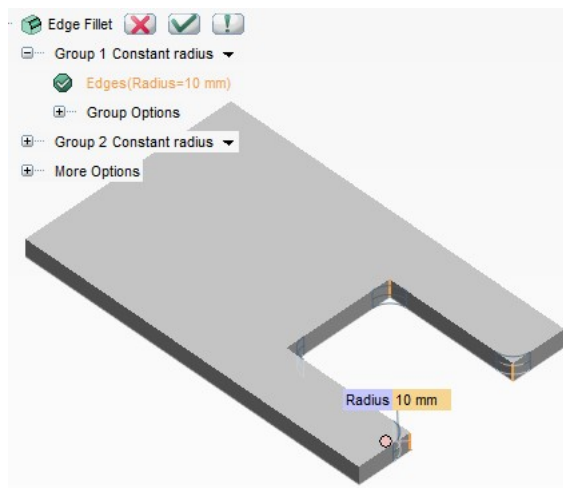
- Start the **Linear Solid** command
- Select the  Profile and set Depth10



Click  OK





Now let's insert Fillets on some selected edges.

- Start the **Edge Fillet** command
- Select the  Edges as shown below
- Set the fillet radius value to Radius10

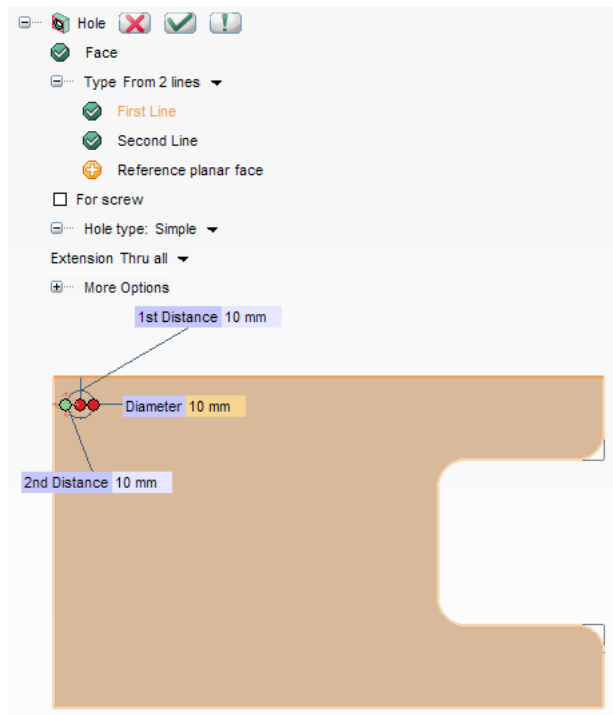


Click  OK




Now insert a hole on one corner and mirror it to the other corners of the solid.

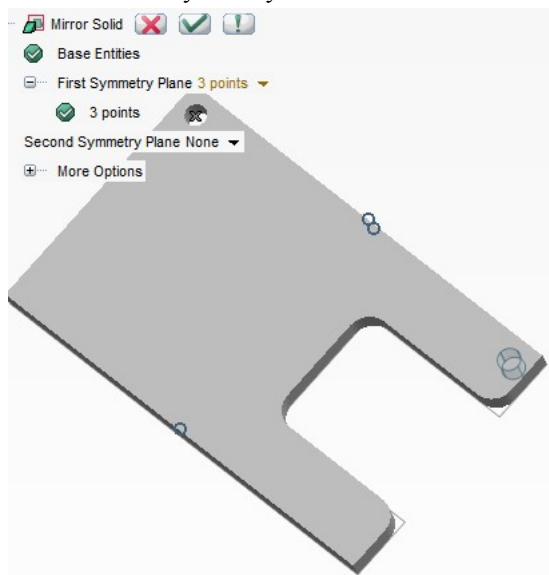
- Start the **Hole** command
- Select the top face as the  Face, and set  Type From 2 lines
- Select the perpendicular edges on the part as the  First and  Second lines
- Set Diameter, 1st Distance and 2nd Distance equal to 10 mm.



Click  Apply...

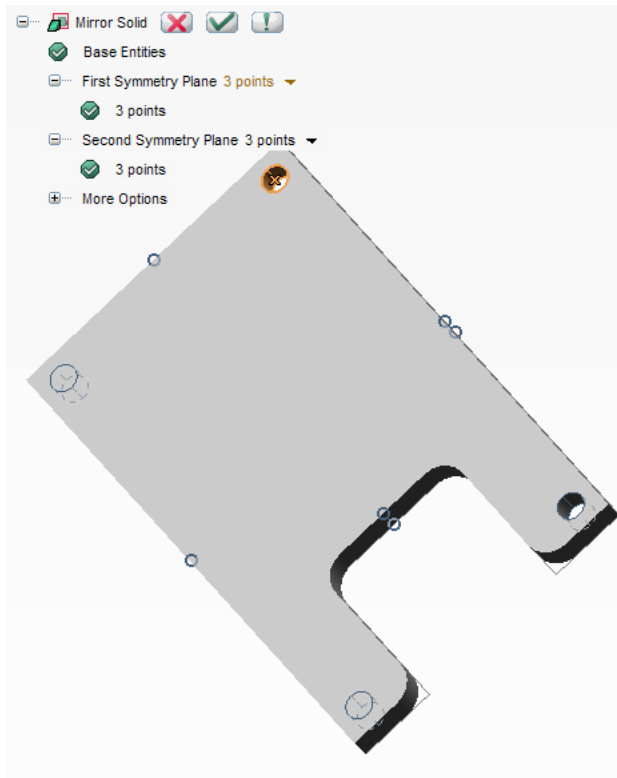


...then click  Cancel.

- Start the **Mirror Solid** command
- Select the hole for  Base entities
- Set  First Symmetry Plane to  3 Points and select the mid-points of the edges of the part as shown below



- Set  Second Symmetry Plane to  3 Points and select the mid-points on the perpendicular edges of the part as shown below
- We are mirroring this hole feature about two separate Symmetry Planes - using the **Mid Point Snap** tool to select the mid-points if necessary.



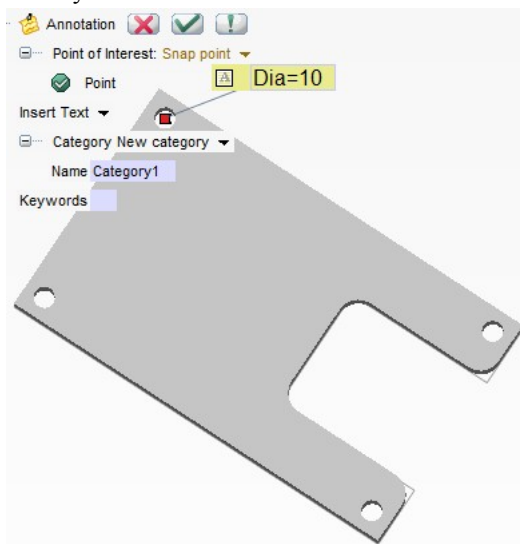
Click OK to create the holes at the 4 corners of the part. Use the **Top View** command to re-orient the part.

We are now ready to create our first Visual Bookmark.


2. Step2 :The First Annotation and Visual Bookmark

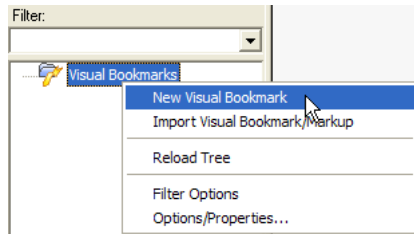
Let's add an annotation, before creating a Visual Bookmark.




- Start **Insert** **Annotation**.
- Select the centre point of the hole for the Point.
- Add text as Dia=10mm.
- Say OK.



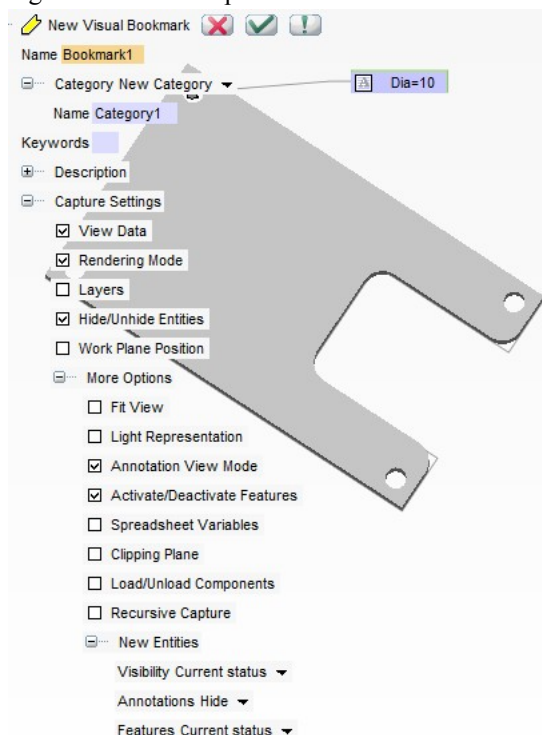
We will create the first Visual Bookmark and set a few parameters to be captured in the process.

- Click the  Visual Bookmarks tab -- located at the bottom-left corner of thinkdesign window
- Right click on top of the Visual Bookmarks tree and click **New Visual Bookmark**.




- Default Name of the bookmark will be Bookmark1.
- Under  Capture Settings, check these items: View Data, Rendering Mode, Hide/Unhide Entities.
- Under  More Options, check Annotation View Mode and Activate/Deactivate Features.
- Under  New Entities, set Visibility as Current Status and Annotations as Hide, as shown below.

The Creating/modifying a Visual Bookmark document explains how we can create a Visual Bookmark by listing the whole set of parameters.



Creating/modifying a Visual Bookmark.

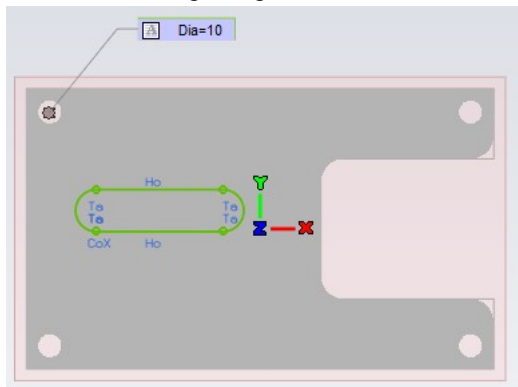
Setting the Annotations option to "Hide" in New entities option enables us to define the status of the new features, that will be added after the Visual Bookmark creation. We can choose from the following options: Current status : Keeps the current status Activate : Features are activated Deactivate : Features are deactivated

Click  OK to create the first Visual Bookmark.

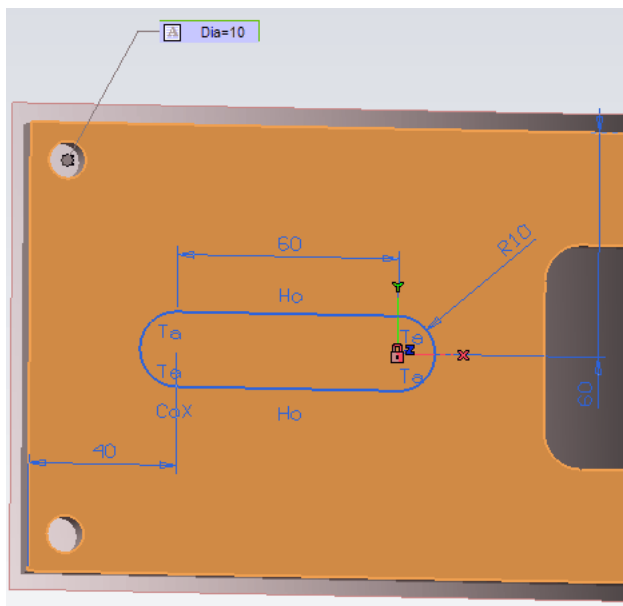
You can see the Bookmark appear in the Visual Bookmark tree.


Let's continue...

- Double-click on the front face (the one used to create the **Hole** feature above) to move the Work plane there.
- Activate 2D Profile.
- Start the **Polyline** command
- Sketch an elliptical profile as shown below






- Insert dimension constraints as shown below

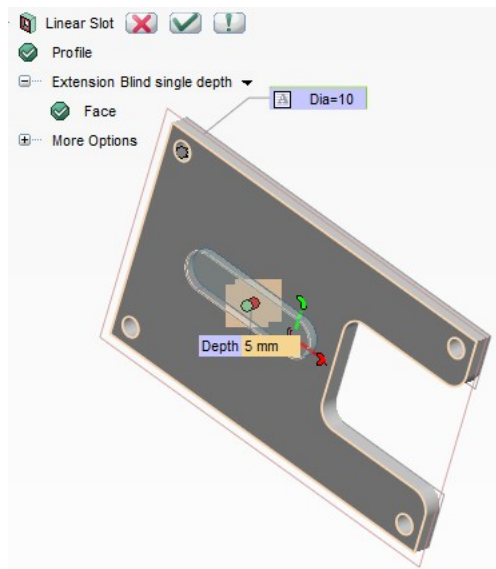


Click  Cancel when finished.

Let's now make a slot using this Profile.

- Start the **Linear Slot** command





The  Profile and  Face are picked automatically. Set the  Extension Blind and Depth5.

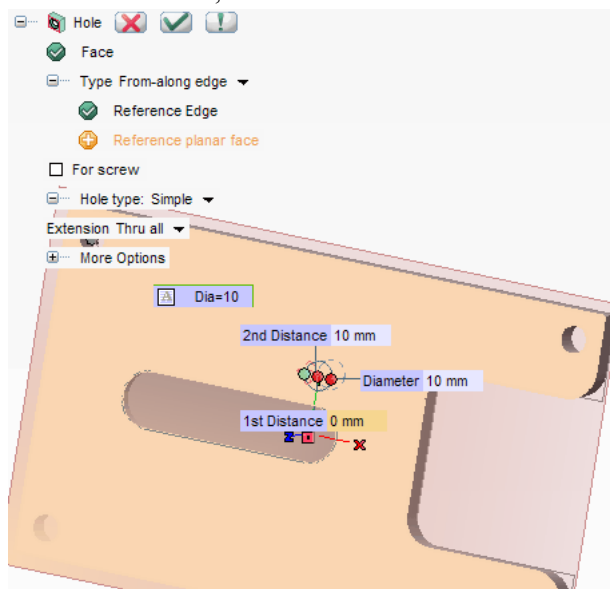



Click  OK.

- Right-click on the slot feature and click Hide Dimensions




Let's insert another hole.

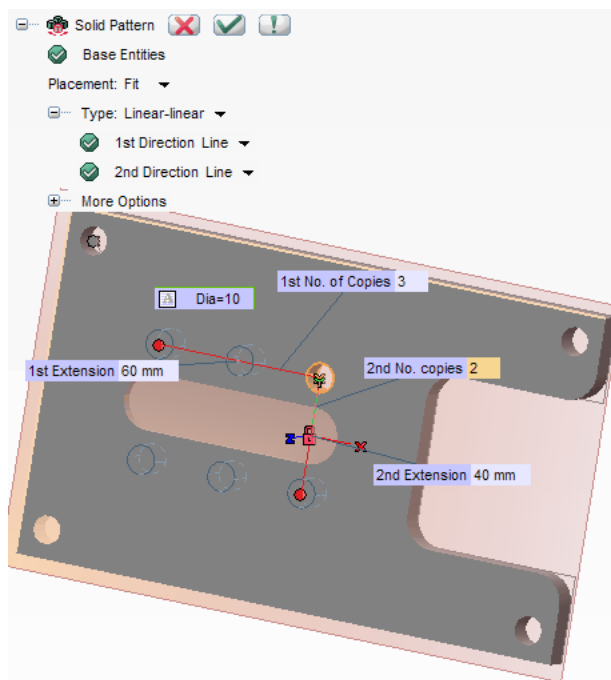
- Start the **Hole** command
- Select the same face sketched on in the previous step as the  Face
- Set  Type From-along edge and select the upper horizontal edge of the slot for  Reference Edge.
- The  Reference Planar Face gets selected automatically (If it does not, don't worry. It does not have to be selected to complete this step.)
- Set Diameter 10, 1st Distance 0 and 2nd Distance-10



Click  OK to create the hole.

Now let's pattern this hole.


- Start **Pattern Solid** command
- Select the hole as  Base Entities
- Set  Type: Linear-linear
- Select the outer perpendicular edges on the part as the  1st and 2nd Direction Lines (see image below)
- Set 1st Extension 60 and 1st No. copies 3, 2nd Extension 40 and 2nd No. copies 2
- Set Placement: Fit

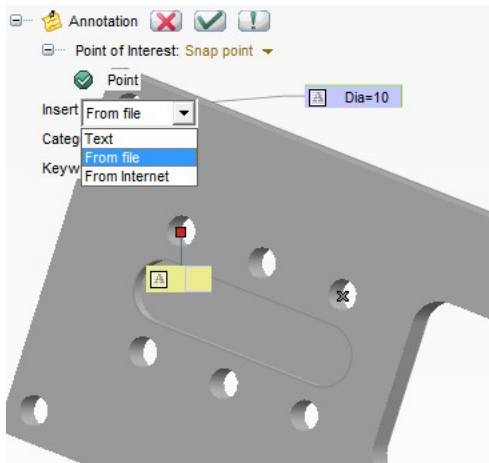


Click  OK to create the hole pattern, then use **Hide Entities** to hide the pattern dimensions and the slot profile.

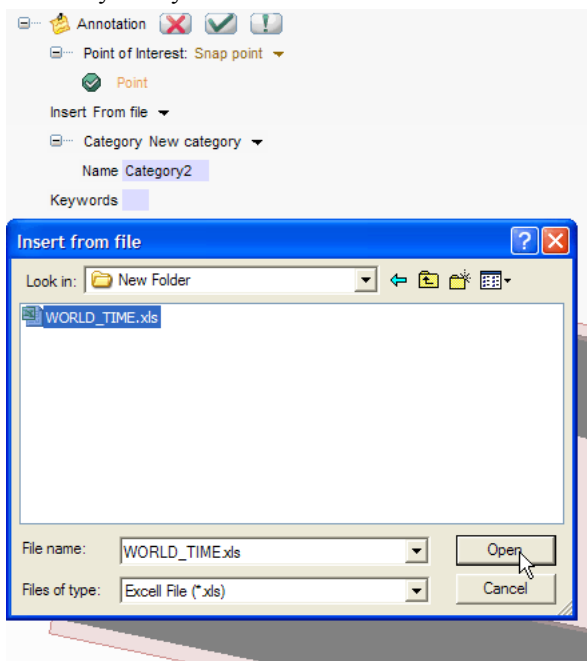
3. Step3 :Visual Bookmarks in Drawings

Let us add an external file as annotation.

- Start **Insert**  **Annotation**.
- Select the centre point of the hole as shown.
- Change the option from Text to "From file".




Insert any files you want to.

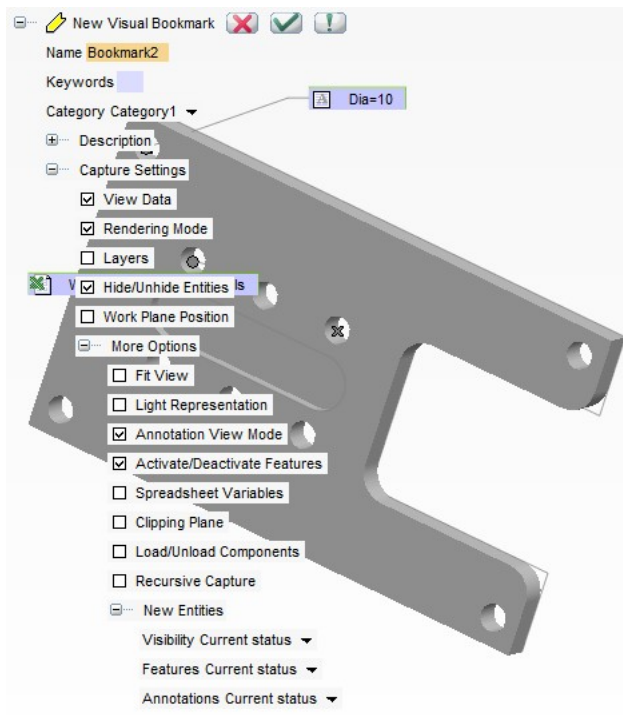



Click  OK.

In this step we will add more Visual Bookmarks and activate them in a drawing to get drawing views.

Let's add another Visual Bookmark now:

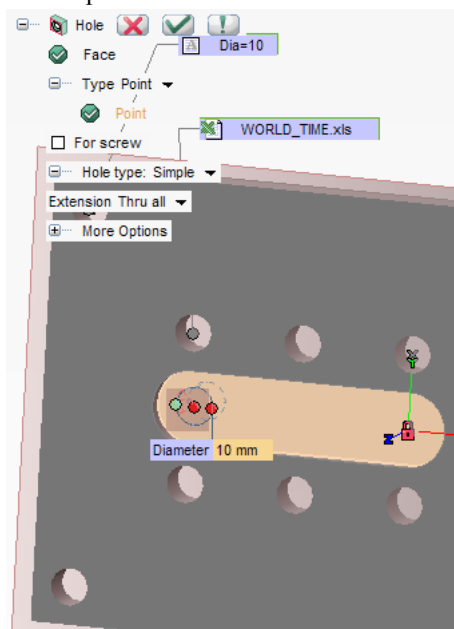
- Orient the part as desired in the Graphics area
- Click the  Visual Bookmarks tab
- Right-click the Category1 and click **New Visual Bookmark**.
- Now the default name will be Bookmark2 and keep all other settings similar to Bookmark1, except the Annotations as Current status.



Click  OK to create this second Visual Bookmark.

Let's add some additional features to our part:


- Insert two new **Hole** (s) as shown below -- use these settings: Diameter10 and Extension Thru all
- Snap to the arc centers of the slot to choose the center for holes.

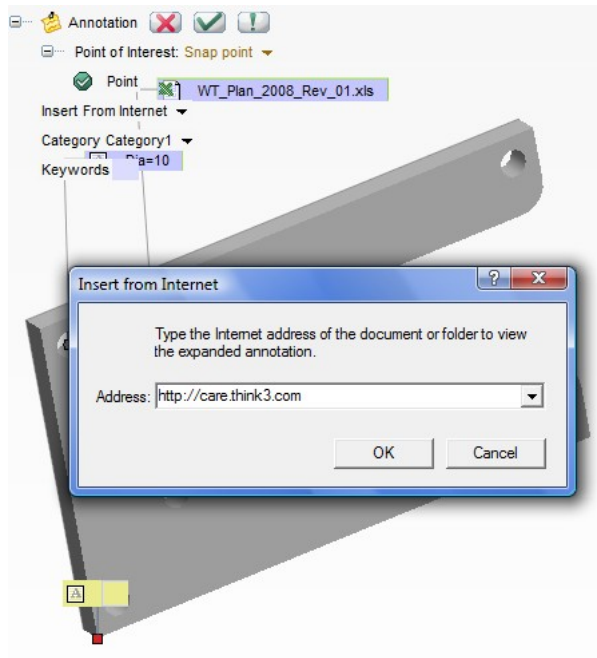


At this stage your model should look like this:

Let's insert the last annotation from the Internet.



- **Insert**  **Annotation** .

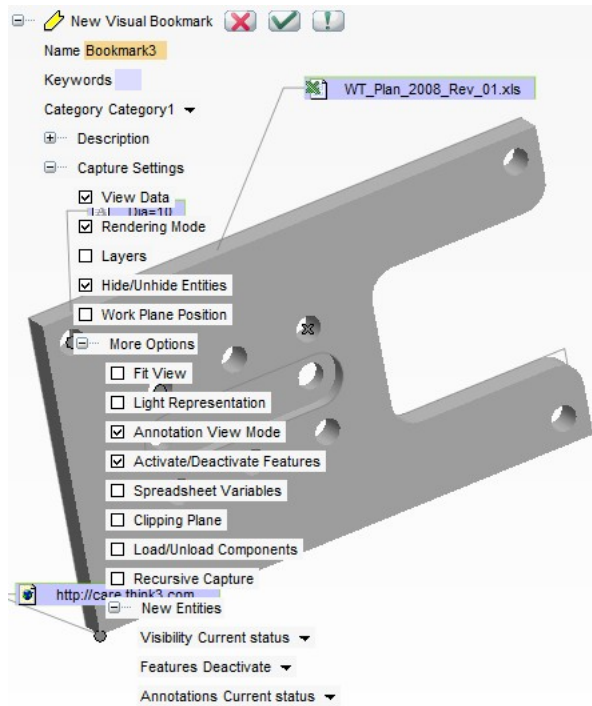
- Select one of the corners of the model for the  Point.
- Change the option Insert to "From Internet".
- Type the website address as www.think3.com.
- Click OK.



- Click  OK in selection list.

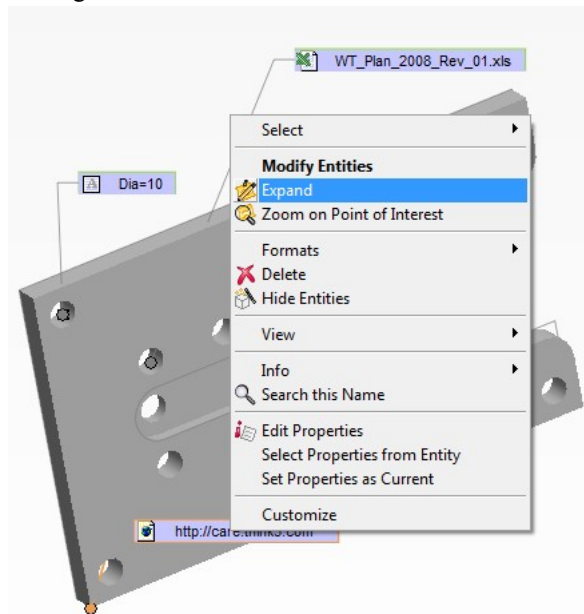
Let's add another Visual bookmark.

- Click the  Visual Bookmarks tab
- Right-click the Category1 and click **New Visual Bookmark**.
- Make the capture settings as shown below and click  OK.




Now let's see how this annotation works.

- Right click on the last annotation we have created and say Expand.



It opens the web page as shown.



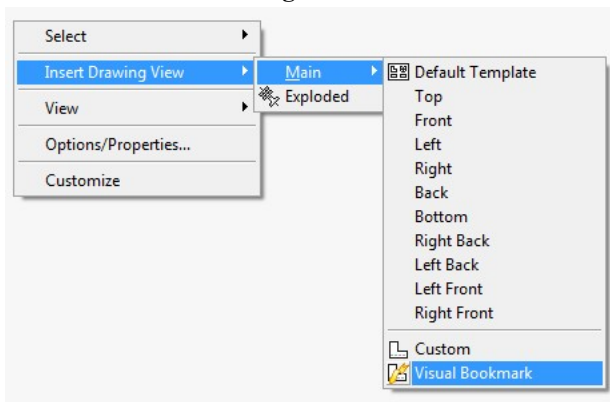
Click  Close Window.


Save this part as Model2.e3.

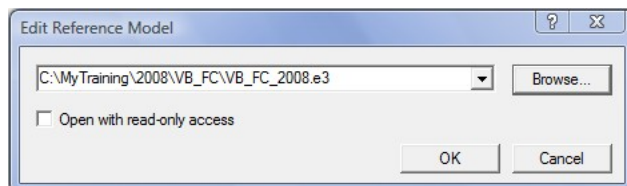
Now let's use the Visual Bookmarks to create Drawing Views -- letting the Visual Bookmark settings control what is visible.

Open a **New Drawing**.

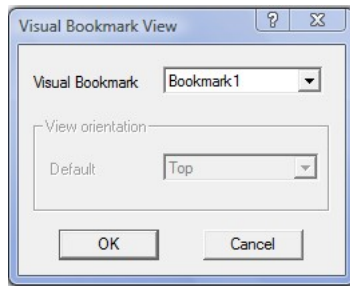
- Click on **Insert** → **Drawing View** → **Main** → **Visual Bookmark**.



- Browse to the Model file and select it
- Click  OK.

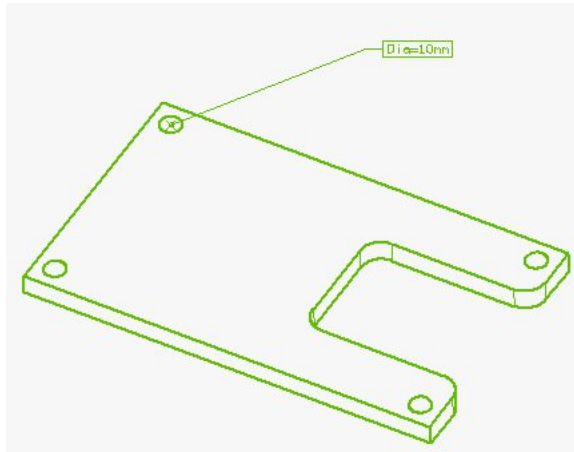


- Set Bookmark1 for the First View.



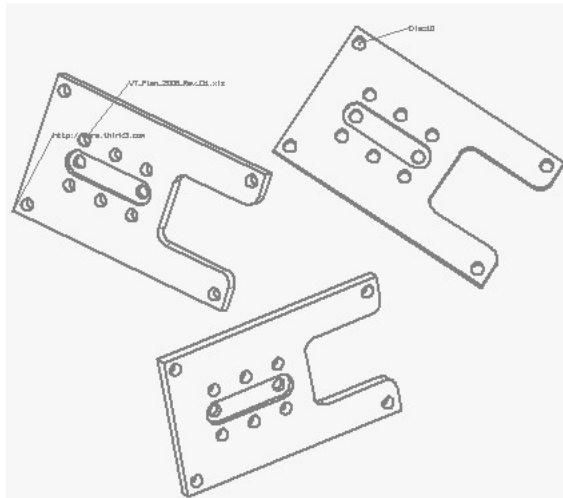
- Click  OK.

The Drawing View appears as per the capture settings for Visual Bookmark Bookmark1.



Similarly, bring in two more views with Visual Bookmarks Bookmark2 and Bookmark4 driving the feature visibility.

The views appear as shown below:



Make a note of the differences in the feature visibility in each of these views. Visual Bookmarks are a quick and easy way to capture feature visibility in both 3D and in 2D.

This brings us to the end of this task.

A.

- An overview on Visual Bookmarks.