BEACON Newsletter - January 2025

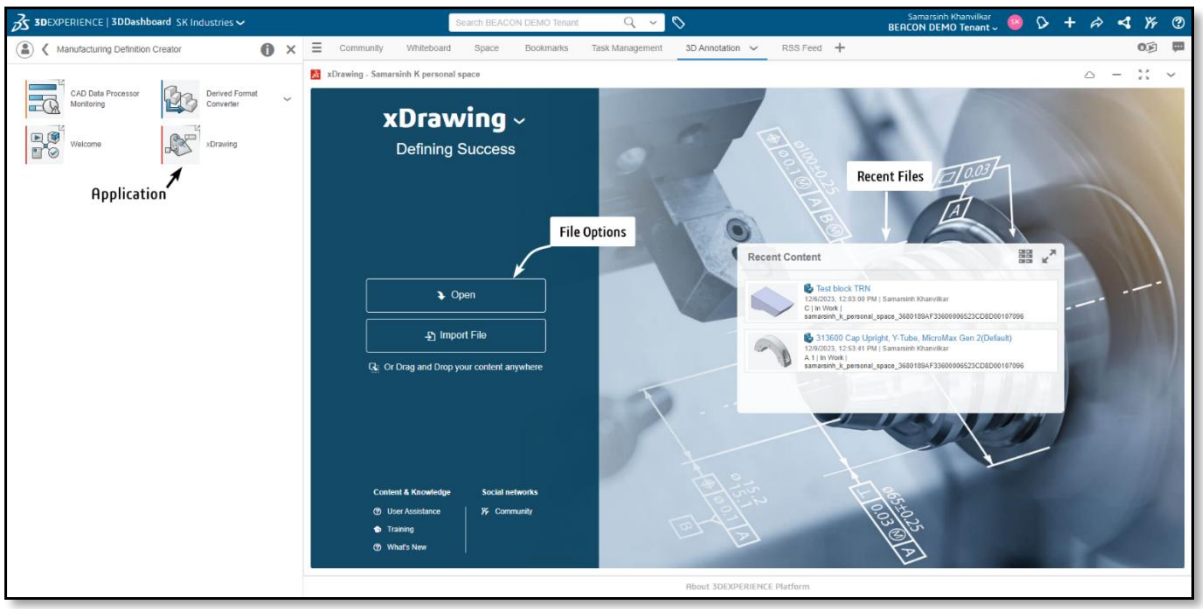
Getting Started with Manufacturing Definition Creator

In the realm of the 3DEXPERIENCE Platform, a recent addition includes the introduction of the "Manufacturing Definition Creator" role. This role is specifically designed to assist users in seamlessly generating 2D drawings and 3D annotations directly within the platform. This enhancement makes the creation of on-the-fly 3D annotations and drawings a straightforward and user-friendly process, contributing to the overall ease of use on the platform.

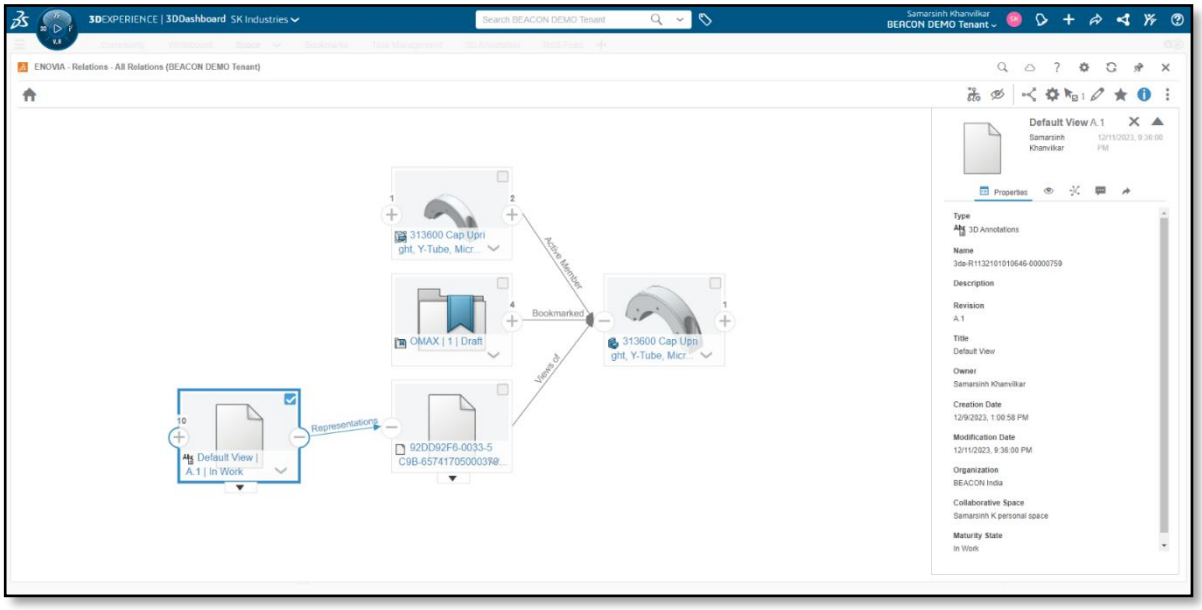
Let's explore the role step by step....

Accessing the role

Accessing this role is straightforward using the "Compass" feature. Click on Compass and search for "Manufacturing Definition Creator." Subsequently, open the xDrawing app, which will prompt the appearance of the following window.

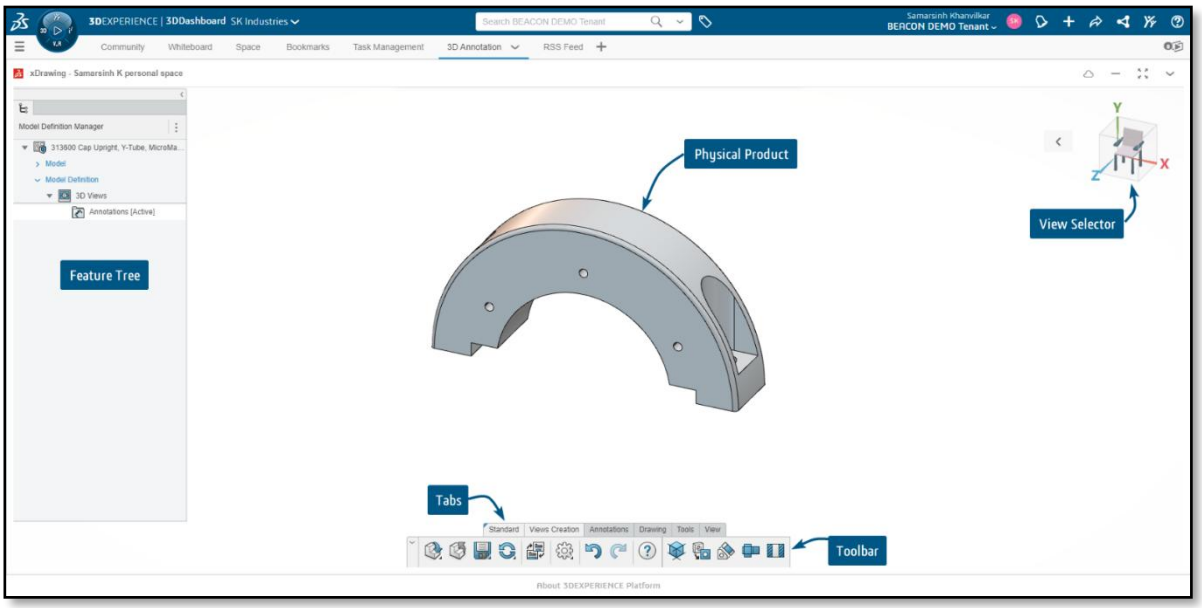


Initiate the creation of a new drawing by selecting "Open" and specifying the physical product to which annotations will be applied. Subsequently, you'll be prompted to establish a new "Annotation Set" for storage on the 3DSpace. This action generates a file on the 3DSpace linked to the designated physical product, accessible and viewable within the relations.



User Interface

After all the above steps, we can start creating the drawing views and annotations with the help of the tools and commands available. Let's first understand the user interface.



1. **Feature Tree:** Model, 3D annotations and respective views and drawing sheets will be visible in this window with their tree structure.
2. **Toolbar:** Commands related to different zones will be visible here inside the respective tabs

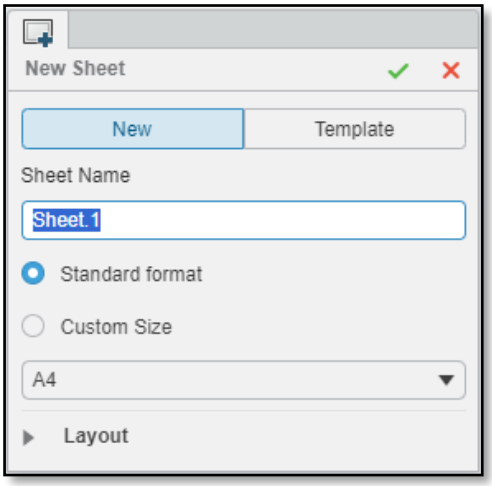
3. **Tabs:** All commands are neatly segregated as per their use and their types. Some commands may be visible or utilized only when a suitable environment is maintained.
4. **Physical Product:** Your 3D part or the sheet you created will be visible here in this window.
5. **View Selector:** You can use the selector to navigate through the different views.

Creating 2D Drawings and 3D Annotations

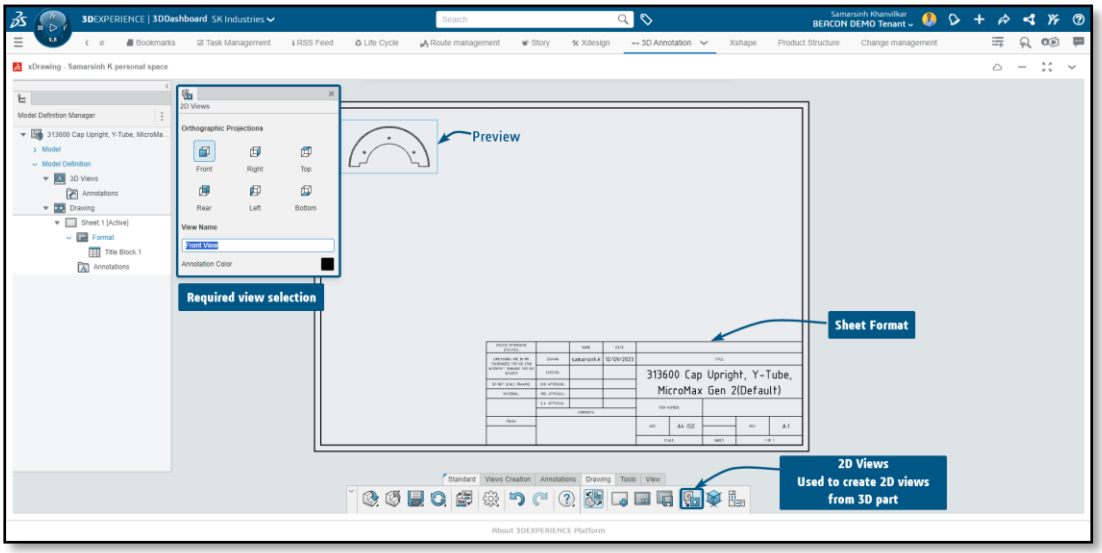
Proceed step by step to initiate the annotation creation process. Typically, annotations and GD&T (Geometric Dimensioning and Tolerancing) are implemented at the drawing level, a familiar workflow for many companies. This workflow proves advantageous as annotations crafted within these drawing views are automatically integrated into the 3D physical product. Moreover, they can be observed in each distinct view for a comprehensive examination.

Now let's create a drawing of a part. Go to the "Drawing Tab" and click on "Drawings Environment" 

1. Selecting sheet: The dialogue box will appear for selecting the drawing sheet and give options to select the template, sheet size and orientation.

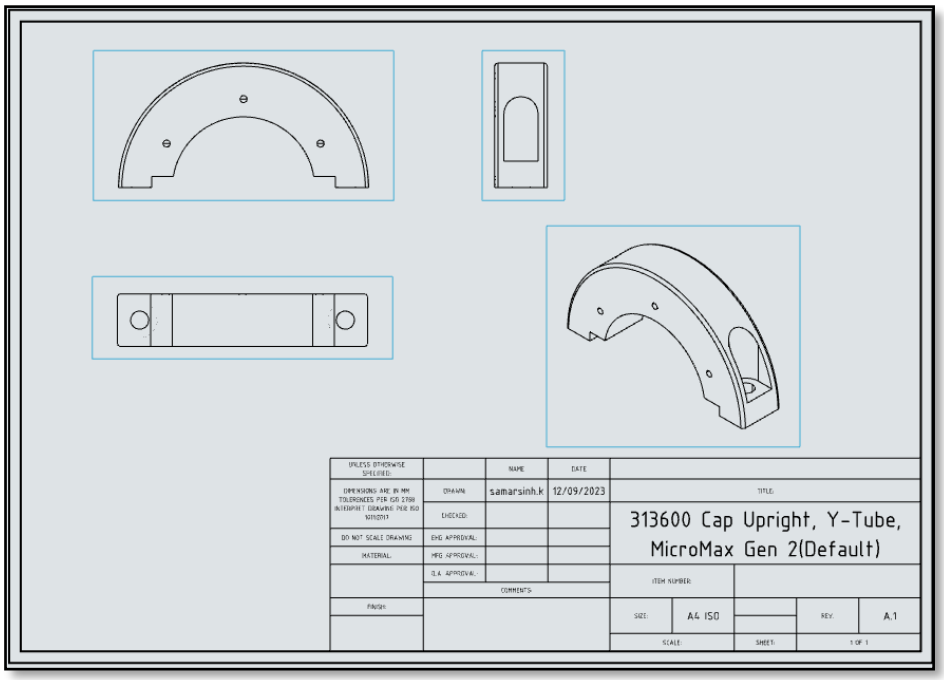


2. 2D View Creation: The views are created from the 3D product. And "2D Views" command is used to do so. Once we activate the command, following window appears:

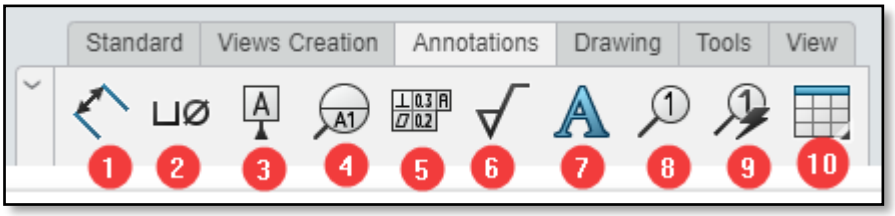


By using this we can create many simple views in the drawing sheet to complete the drafting. Also, the projected views can be added to the sheet using the projected view command: 

3D views can be added using a command called "3D Views"  . After placing all the related views, the sheet will look something like this:



3. Dimensioning and Annotation: Annotations tab consists of all the necessary dimensions and tolerances required to make the drawing finalized and can be found in the toolbar.

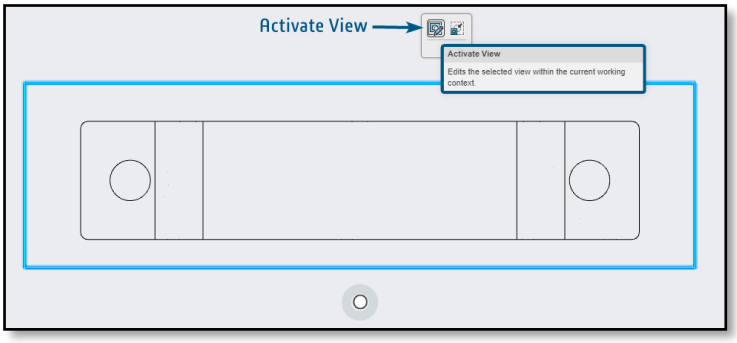


Kindly follow the below one-by-one explanation:

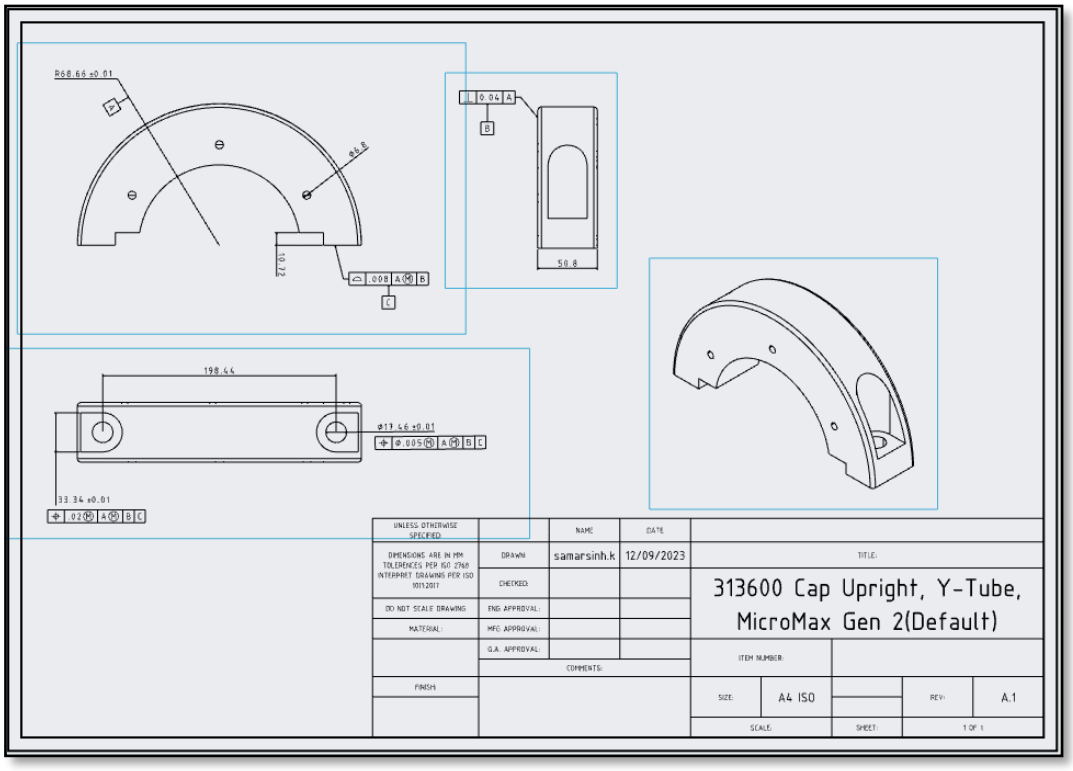
1. Dimension: Adds dimensions to the model. Linear, radial, angular and more can be added.
2. Hole Callout: Adds hole callout to the holes.
3. Datum: Adds the datum to the model.
4. Datum Target: Adds the datum target to the model views.
5. Geometric Tolerance: GTOL can be added with various symbols and schemes.
6. Surface finish symbol: Adding surface finish symbols to the faces and models.
7. Notes: Add descriptive notes to the view.
8. Balloon: Manual addition of the balloons to the parts.
9. Auto Balloon: Automatic ballooning of the part files or assembly files.
10. General Table: It consists of tables such as Regular, BOM and Cut list to be added to the drawing sheet.

Note: All the dimensions and tolerances applied in the 2D views will be automatically replicated on the 3D View.

Adding dimensions and tolerances is the same as SOLIDWORKS. But first, there is one step here we need to take and that is "Activating the View". When we click on any view the Context toolbar appears which has an "Activate View" command. (see the image below)

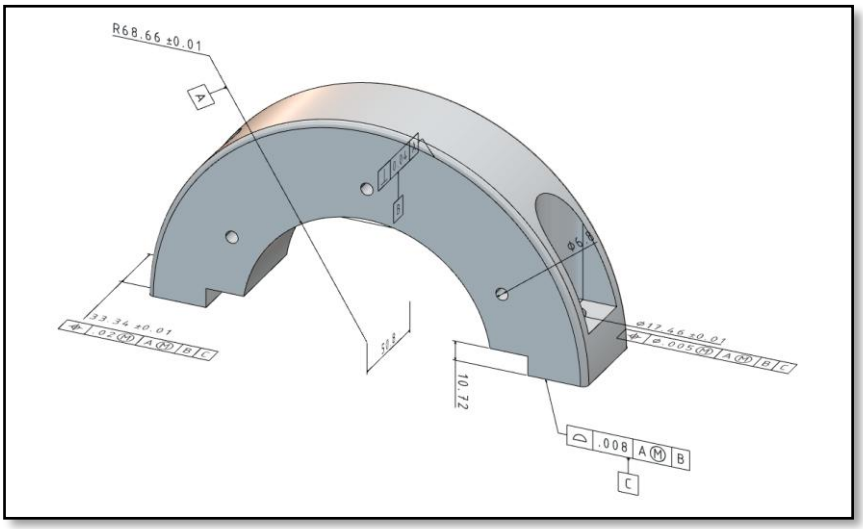


After activating the view, we can complete our dimensioning and tolerancing. And the final sheet will look something like this.



As we mentioned the Dimensioning and Tolerances created in 2D Views will be automatically transferred to 3D views. This will save repetitive steps and make the UI more user-friendly. Engineers today always do their dimensioning in 2D, but to accommodate Model-Based Dimensioning or in the simple term "Annotation sets in 3D" is now become easier by this automated method.

This is how the same dimensions look in 3D view:



Conclusion:

In conclusion, the introduction of the "Manufacturing Definition Creator" role within the 3DEXPERIENCE Platform marks a significant advancement in streamlining the process of generating 2D drawings and 3D annotations. By providing a user-friendly interface and seamless integration with the platform's tools and commands, this role simplifies the creation of comprehensive manufacturing definitions. With features such as automatic replication of dimensions and tolerances from 2D views to 3D models, it not only enhances efficiency but also promotes a smoother transition towards Model-Based Dimensioning. Overall, this role empowers users to create accurate and detailed manufacturing definitions with ease, contributing to enhanced productivity and collaboration within the manufacturing workflow.

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