



Inventor Professional 2024



DESCRIPTION

An introduction to Inventor and the beMatrix Plugin to efficiently create stand designs and more.

LEARNING OBJECTIVES

- Learn how to set up Inventor to work with the library and all the automated tools
- How to start and manage projects
- The beMatrix Library
- Custom made parts based on library parts
- Custom made parts from scratch
- Calculate Connectors
- Review and change automated connections
- 2D Plans
- Placing Panels
- Rendering
- Export your design
- Tip & Tricks: Ideal and structured ways of working

SKILL LEVEL

A basic knowledge of Inventor should be expected such as placing and manipulating parts and assemblies.

ABOUT ME

My name is Kevin Hendrickx, I've been working as a CAD Designer at beMatrix since 2018. During this time, I've made about 2000 technical drawings for clients all over the world. Most of you have probably already been in contact with some of the drawings that I've made.

Besides designing projects, I also create all the drawings for custom made parts which are either produced by our own production facility or purchased at local suppliers. In both cases highly detailed drawings are required to make sure that all parts are made to the required specification, tolerance, functionality and stability.

Our number one concern is to always provide our clients with a design which matches their idea as close as possible, but still make sure that everything is 100% stable and safe to build. We try to think along as much as possible on how to build and try to foresee any problems that might arise.

If you have any questions after this session or need help and assistance, feel free to contact me.

Let's go!



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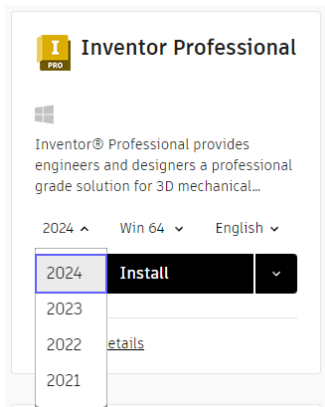
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Inventor Setup

Installing Inventor

At the moment of writing beMatrix has a library which is compatible with Inventor 2024. We generally keep using this library for 2 years before updating to the latest version available of Inventor. When we update our library the current users will get a notification that an update is recommended.

When purchasing an Autodesk Inventor license, be it standalone or included in the production suite, you should have the ability to choose which version of Inventor you wish to install up to 3 versions back.



It's possible to have multiple versions of Inventor installed on 1 machine, but we highly recommend to not do this as different version might create conflicts.

The beMatrix plugin can only be installed for 1 version at any given time.

Please follow the installation instructions provided by Autodesk for the installation of Inventor on your machine.

[Autodesk Inventor Professional Help](#)

If you require any help installing the Autodesk software we recommend contacting the Autodesk Customer Service.

Installing the beMatrix plugin

It's possible to request our beMatrix plugin from our website. <https://bematrix.com/eu/en/becad-inventor>

Our plugin is completely free and gives you access to a synchronized beCAD library in the cloud. This library is constantly updated with our latest items as well as preset assemblies containing different parts already in the correct position.

Using our plugin will also enable you to calculate connectors automatically just like on the beCAD Online tool.

Next to the automated placement of connectors there is also a wide range of automated functions that will assist you to generate designs more efficiently, correctly and quickly.

To install the plugin, do the following steps:

- Startup Inventor for a first time and **press CTRL + F11** to trigger the installer for the VBA Plugin needed to install the beCAD Inventor plugin.
- Execute the .exe file that you received from beMatrix by mail.
- After the installation, check if the following folders have been created at **C:\Users\[Username]\beMatrix**

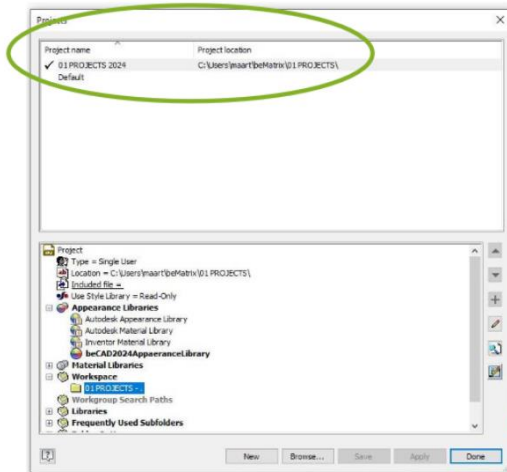
OS (C) > Gebruikers > maarten > beMatrix >



- Startup Inventor and you should receive a pop-up. Unblock the following 2 Add-In's
 - **beMatrix File Manager**
 - **Inventor Automation**

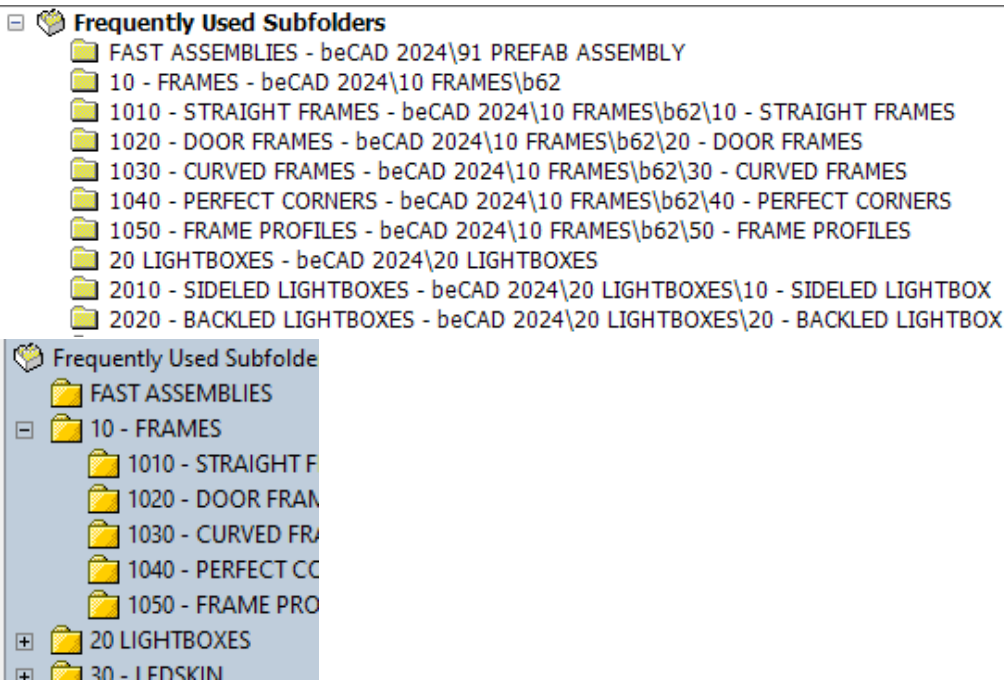
Setting up the Project files

- Go to **Projects**, where a **01 PROJECTS 2024** should have been created already.



- Check if the appearance and material libraries are correct

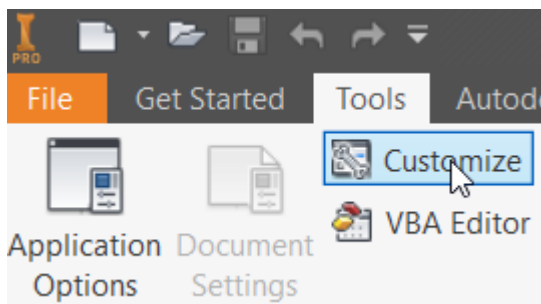
If required, you can also add Frequently Used Subfolders to have shortcuts to specific paths in the library



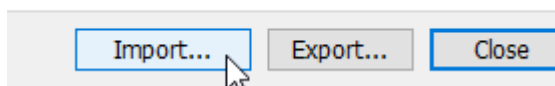
Setting Up The UI

In order to speed up your efficiency it's very handy to have several commands and features in fast accessible locations. It's highly recommended to customize the toolbar, marking menu and hotkeys to improve how fast you can build your designs.

Go to the **Tools** tab and choose **Customize**



Again click on **Import**



In the window that pops up, browse to the folder included in the library:

C:\Users\[Username]\beMatrix\beCAD 2024\99 Templates\20_Config

and select the *beCAD_Customize_Ribbon_KeyBoard_MarkingMenu.xml* file to import the settings file containing all the UI settings to maximize the efficiency with the beCAD Add-In.

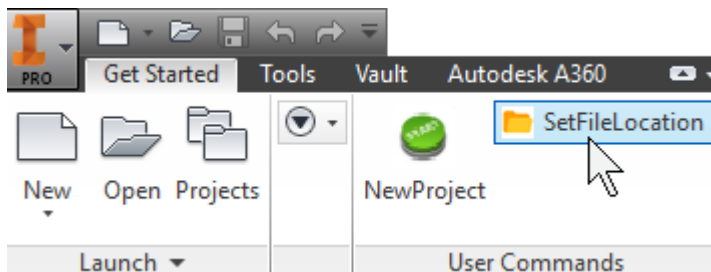
Overview of hotkeys:

Q	Gridsnap
W	Change Element
CTRL + SHIFT + X	Flip X-Axis
CTRL + SHIFT + Y	Flip Y-Axis
CTRL + SHIFT + Z	Flip Z-Axis
TAB	Create Subassembly
SHIFT + TAB	Move to Subassembly
F6	Home View
M	Measuring Tools
ALT + V	Toggle Visibility
ALT + T	Toggle Transparency
CTRL + H	Replace Selected Part
CTRL + SHIFT + H	Replace All Occurrences Of Selected Part

Setting File Locations

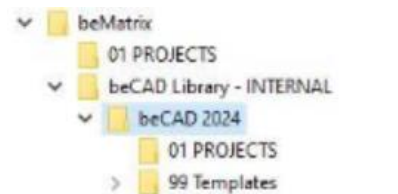
For the functionality of several beMatrix specific commands we need to direct Inventor to the specific folders.

Use the **SetFileLocation** button



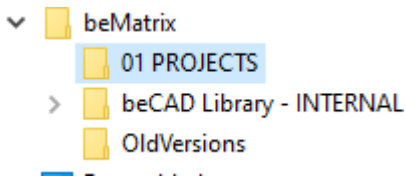
First select the location of the beCAD folder:

C:\Users\[Username]\ beMatrix\beCAD Library – INTERNAL\beCAD 2024

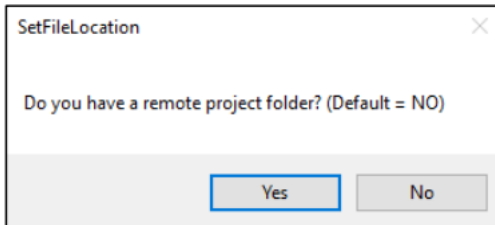


After this select the folder where you will store your projects:

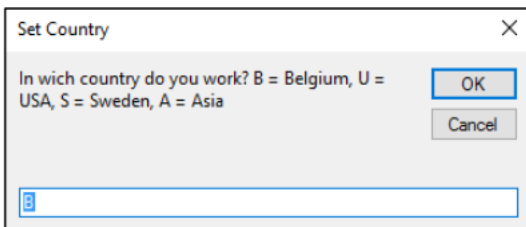
C:\Users\[Username]\ beMatrix\01 PROJECTS



Unless you are using a remote project folder (on a network drive) select **NO** when asked:



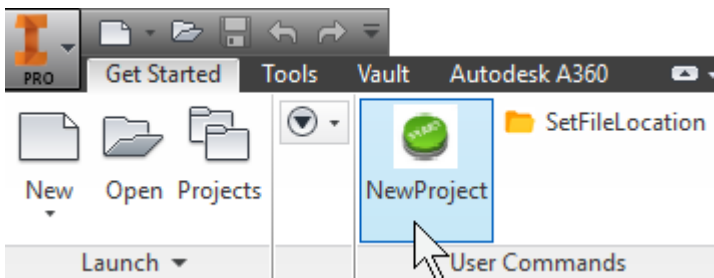
The region select should be **Belgium**



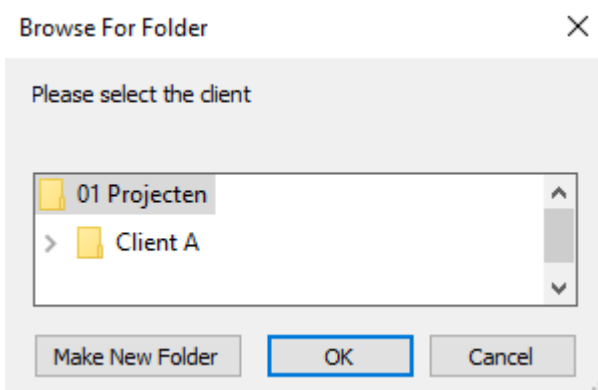
This should conclude the initial setup of Inventor for beCAD. To make sure that everything is working as it should we will test it using a dummy project.

Starting a Test Project

Startup Inventor and try the **New Project** button



You should see the following pop-up asking to select the client. If the client does not yet exist use the **Make New Folder** option to create a new folder for the new client.





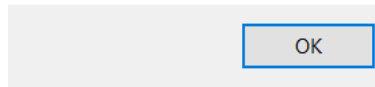
If all went well Inventor is now completely setup for use with the beCAD plugin.

Troubleshooting

- *if the NewProject button isn't visible, this means that Inventor can't find the VBA file containing the macros or the VBA plugin isn't installed yet. Please check if you have the following path in the application options under the Default VBA project line*
**C:\Users\[Username]\beMatrix\beCAD 2024 for clients\99
Templates\05_Default VBA project\beMatrix 3.1.ivb**
- *If instead of the Browse for Folder popup, you receive the following message*

Autodesk Inventor Professional ×

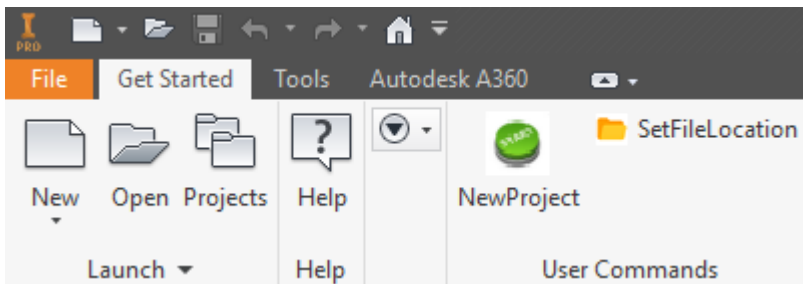
Please Activate the correct project



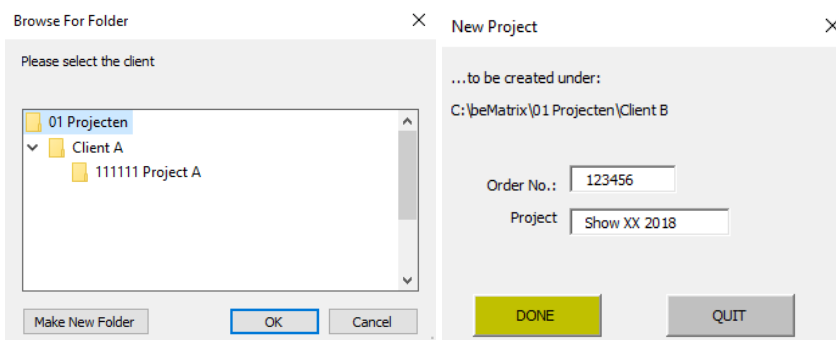
please make sure that the 01 PROJECTS project file is active.

Starting A New Project

Now that everything is set up, it's time to begin a project. We do this by using the New Project button. This will start a macro that will ask and store the basic information for your design.

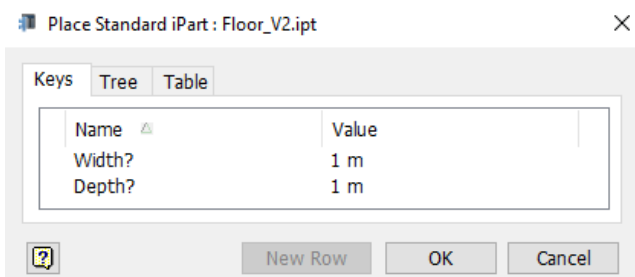


Inventor will ask you to choose the client or create a new one and then give an (optional) order number and project name.



After this a new file will be created under the client's folder in a new subfolder by the [order number] [project name] syntax. This will make sure that each project will get its own unique subfolder to keep things organized.

The new file will open in Inventor where a floor will be present. We can immediately change the floor size to the dimensions that are available for the project.



Once the floor is specified we can begin setting up frames.

The beMatrix Library

Our library is organized in several subfolders some containing even additional subfolders. Take your time to explore this library and get acquainted with it. It might very well be that we have some connectors and parts that you have never been aware of.

All frames and parts are possible to be placed in 3 ways.

- Using the **beCAD library** in Inventor
- **Drag and dropping** the files from an windows explorer window
- Using the "**Place Component**" command in Inventor

The main folders are:

- **10 FRAMES**
- **20 LIGHTBOXES**
- **30 AV SOLUTIONS**
- **40 PARTS**
- **50 INFILLS**
- **60 LIGHTING**
- **70 ACCESSORIES**
- **80 IZI-RANGE**
- **100 XTREME SOLUTIONS**
- **110 POP SOLUTIONS**
- **120 KICK-OFF**

10 FRAMES

This folder contains 2 divisions, our old b55 range and the standard b62 range. In Inventor we will always be using the b62 frames.

The b62 folder is further split up in to all our different types of frames

- **10 – STRAIGHT FRAMES**
- **20 – DOOR FRAMES**
- **30 – CURVED FRAMES**
- **40 – PERFECT CORNERS**
- **50 – FRAME PROFILES**
- **60 – GLASS FRAMES**

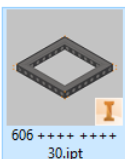
> beMatrix > 02 beCAD 2018 > 10 FRAMES > b62 > 10 - STRAIGHT FRAMES

Autodesk Inventor Assembly (1)



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30.iam

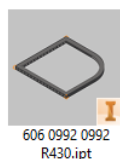
Autodesk Inventor Part (4)



606 ++++++
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Dimensions.ipt

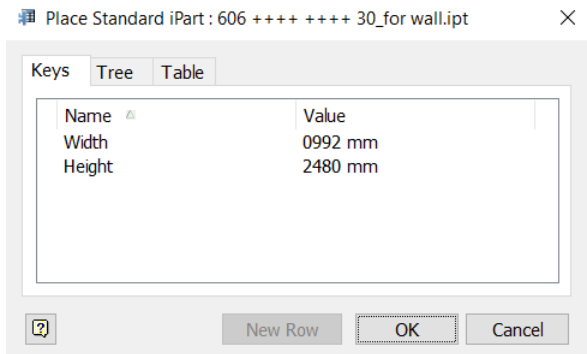


606 0992 0992
R430.ipt



611 911 +++++.ipt

Each of these folders contain several parts. Some of these are single parts, other are iParts containing all standard dimensions within them in a table. When placing these you will get a popup asking the required size of frame.



Once a size is chosen, the iProperties of that part will update to match the frame in your design. This will result in correct BOM (Bill of Materials) that you can utilize to place your order at beMatrix.

As you can see, we do not take in account different color finishes to our parts. All frames and profiles are in an ECO appearance (or their standard RAL color). If required you can always override the appearance of parts in Inventor, but we'll come back to that later in our training.

20 LIGHTBOXES

This folder contains all the different type of lightboxes that beMatrix has.

- **10 – SIDELED LIGHTBOX**
- **20 – BACKLED LIGHTBOX** (no longer sold)
- **30 – LIGHTBOX PROFILES**
- **40 – LIGHTBOX PARTS**
- **50 – MOTIONSKIN** (no longer sold)

We have both iParts that can be used for standard lightbox setups and sizes, but also iLogic parts that can be placed when you need more control over the dimensions, type of profiles, type of holes, ...

30 AV SOLUTIONS / LEDSKIN

These folders contain all audio-visual components such as our LEDskin and the new beTV.

- **10 – LEDSKIN**
- **20 – BETV**

40 PARTS

The parts folder is without doubt the biggest and most used folder with Inventor. The folder contains pretty much all different beMatrix connectors, profiles and tools used to connect and finish your frames.

It's here that the biggest difference lays between the beCAD online tool and the Inventor Plug-in.

While in the online tool you can only calculate the connectors based on rules, but you cannot manipulate or change them. In Inventor you are completely free to add, remove, change or replace connections as you like. Keep in mind that doing so you might compromise the structural stability of your design. So always take care to closely review all connections before releasing your design.

Mainly used subfolders are:

- **10 - COVER PROFILES**
- **20 – CORNER PROFILES**



- **60 – D30 STEEL PINS**
- **70 – D30 CONNECTORS**
- **100 – BASE PLATES**
- **160 – END PIECES**

50 INFILLS

The infills folder contains the standard infill panels for our b62 system. There are panels available for straight and curved frames, doors, iZi Range, Double Deck, POP-Out, ...

Textiles are not (yet) available.

60 LIGHTING

Our SAM, big SAM and CEE lights are all available in the lighting folder as well as their accessories to place them on frames or profiles.

Our Track light system is also included in this subfolder.

70 ACCESSORIES

Accessories contains some auxiliary items such as trolleys or HPL panels.

80 IZI-RANGE

Our iZi-Range eco system containing all the different items with or without the use of panels.

100 XTREME SOLUTIONS

Xtreme Solutions consists of 2 categories. The beTruss system and the Double Deck.

110 POP SOLUTIONS

Both our indoor POP-In as the outdoor POP-Out are located in these folders

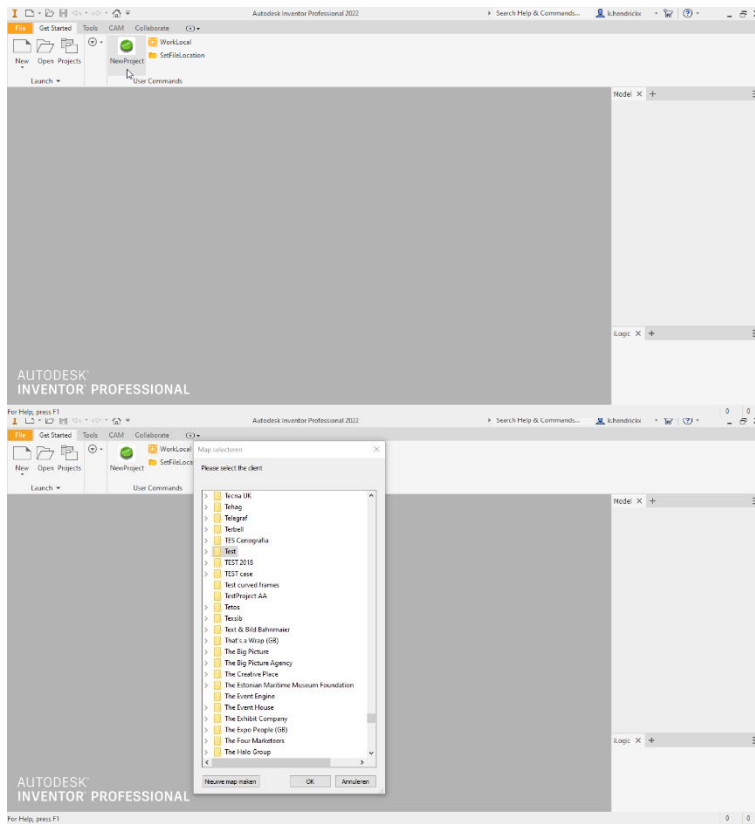
The POP-In subfolder also contains the Glass frame SET's which are used to create large glass walls.

120 KICK-OFF

The outdoor Kick-off folder contains our UnderCovr.

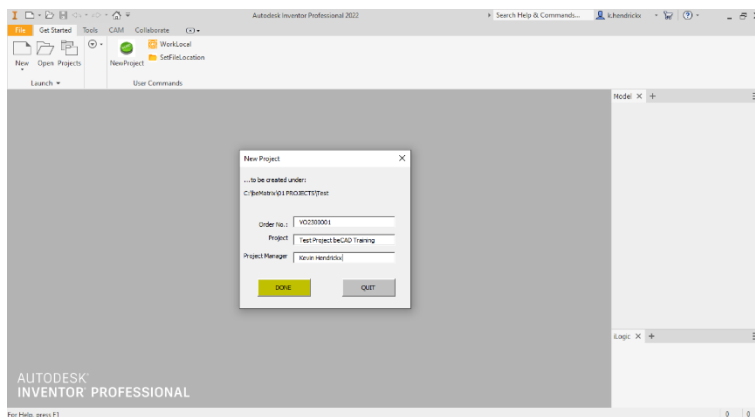
Building your project

When building your design there are some important rules to take into account. Always make sure that the **01 PROJECT** is active, this will ensure that the correct libraries and workspaces are being used. If a different project is active, it will not be possible to create a new beMatrix design.



Once we made sure that the correct project is active, we use the **New Project** button to get started.

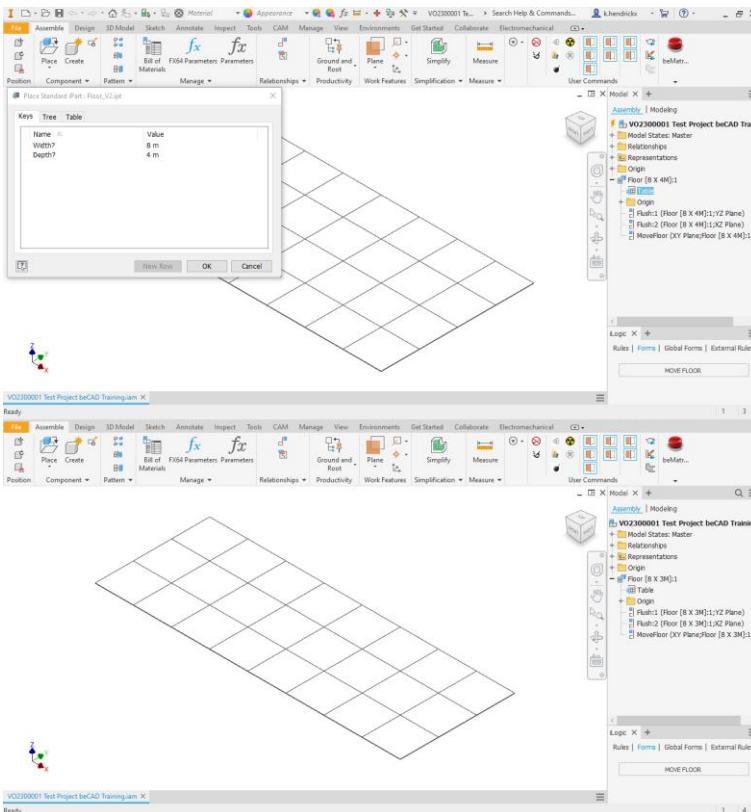
A pop-up will appear asking you to select the client's folder. If you are creating a project for an existing client, choose the name of the client. If it's a new client, create a new folder under the 01 PROJECTS folder with the client's name.



The next pop-up will ask us some details of the project, fill in the Order No., Project name and optionally a Project Manager. Once you've filled in the details click on the **Done** button.

The beCAD plugin will create a new project assembly based on the details we've entered with the following syntax: **[ORDER NO] [PROJECT NAME].IAM**

It will also create a new subfolder under the client's folder to keep your files organized and structured.



Once the assembly has been created, we will be given a floor surface and another pop-up asking us how large the booth surface is. This floor will serve as a guideline to remain within your allocated space.

You can choose any size from the dropdown boxes next to Width and Depth, once you click ok the floor will update to the given dimensions.

We can now start by placing frames on the surface of the floor. Important is to keep in mind that the XY plane of the main assembly will be used as the reference for the engine to calculate the connections. So please pay attention to the positioning of the frames. The floor itself is constraint in place to prevent it from accidentally being moved from the XY plane.

To place frames there are 3 ways:

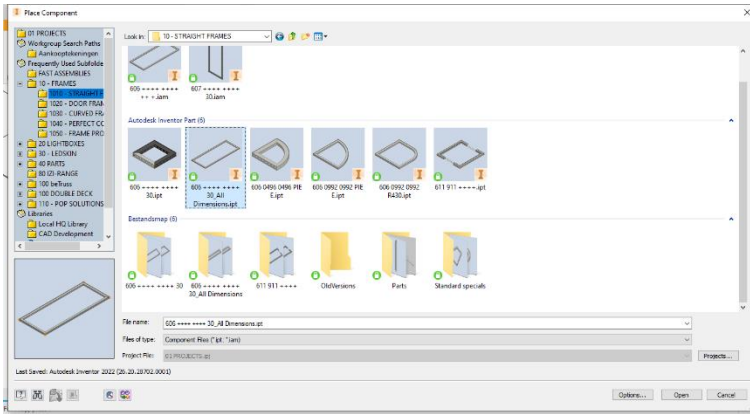
1. Using the **beCAD library** accessible from within Inventor
2. **Drag and drop** from a windows explorer window
3. Using the **Place Component** command from Inventor

When unfamiliar with the beCAD library we recommend using the first option. This will give you an organized library based on groups and families. All parts have a convenient thumbnail to make it easier to identify the different parts. Double click a part to place it in your assembly.

When you double click apart from the beCAD library, it will automatically download the most recent model from the cloud and will store it in your local library. From then on you can always use the local version or keep using the library to constantly update your parts.

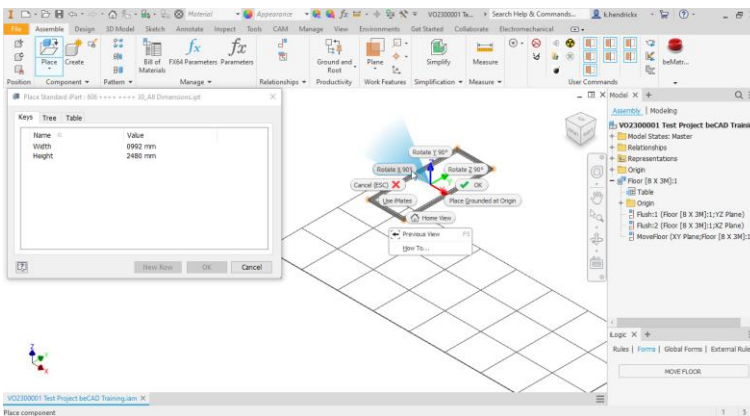
Some parts, such as frames, have multiple variants to choose from when placing. You can for example choose the dimension of the frame before placement.

Let's place our first frame.



For this training I'll be using the **Place Component** command from Inventor. Initially you will need to select the frames and parts from the beCAD library to have them available on your machine.

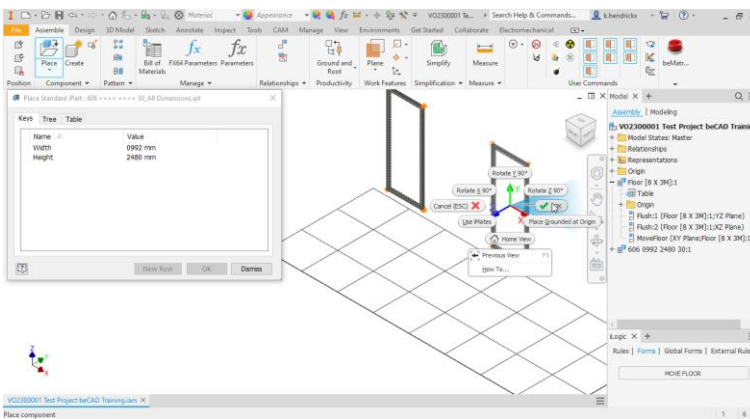
Choose the *606 +++++ 30_All Dimensions.ipt* file.



We will receive another pop-up asking us which dimension we would like to place. Choose the **992 x 2480mm** frame.

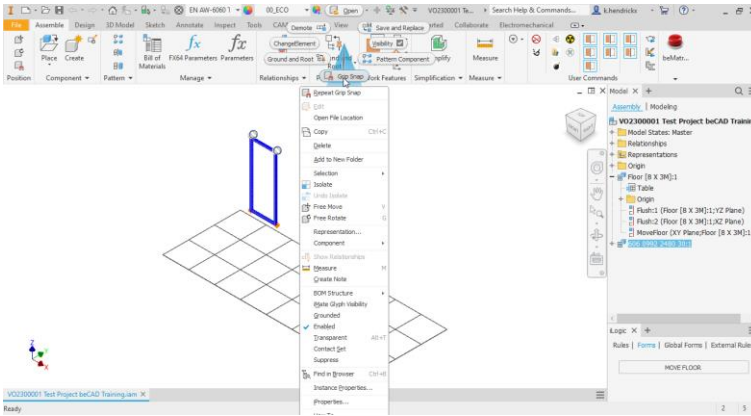
The frame in the window will change to the specified dimension. Right click and choose in the marking menu to rotate the frame 90° along the X-, Y- or Z-axis. You can use the triad on the centre of the frame as a reference.

Rotate the frame 90° along the X-axis to position in vertical.

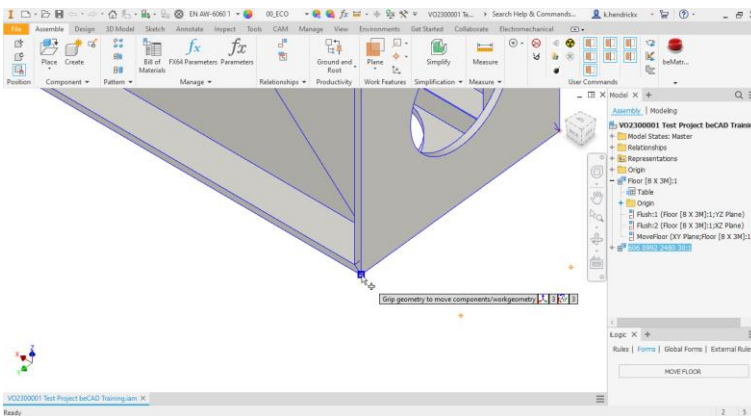


Place the frame roughly where you want to place it and finish by **right clicking** and **OK**.

We now placed our first frame, but still have to position it more accurately on to the floor surface. We will use Grid Snap to precisely move our frames and parts into the correct position.



Right click the frame and choose **Grid Snap** (hotkey = Q)

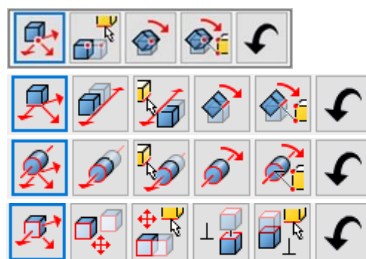


With the Grid Snap command active, zoom in on the frame and notice that there are several options on where to select your frame.

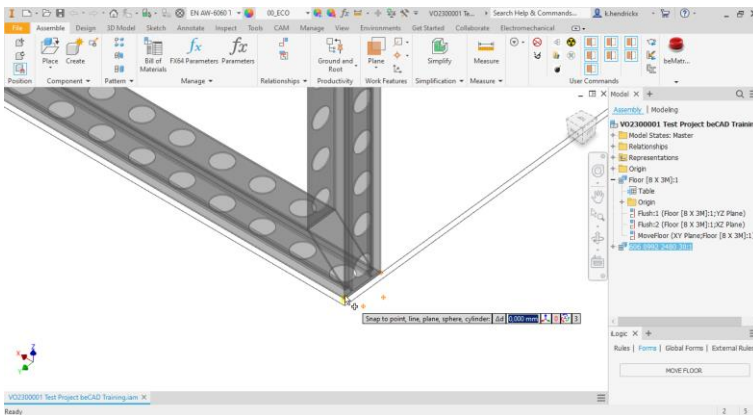
You can use end point, midpoints, edges, edge axis vectors, work points,... each type of reference will give you several options on how to continue. For now, we will select the corner endpoint to position the corner of the frame on to the corner of the floor surface.

With Grid Snap you are able to quickly position one or more component(s). When starting the command, you'll be given the option to select points, lines, axes or surfaces. Depending on your selection you'll be given one of the following options:

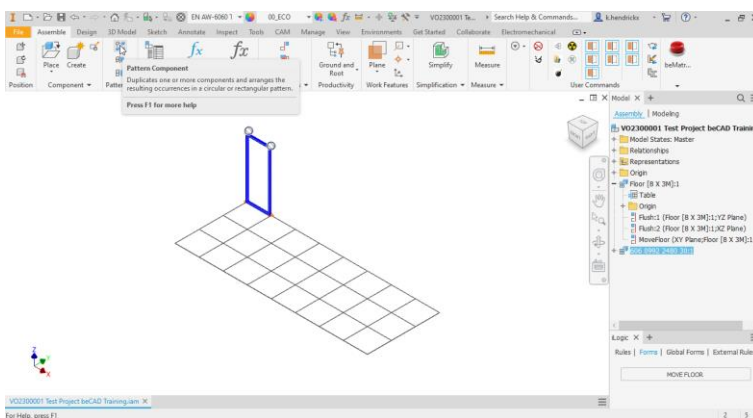
- Point
- Line
- Axis
- Surface



Once you've made a movement, confirm it by **right clicking** and choosing **OK** or **Continue**

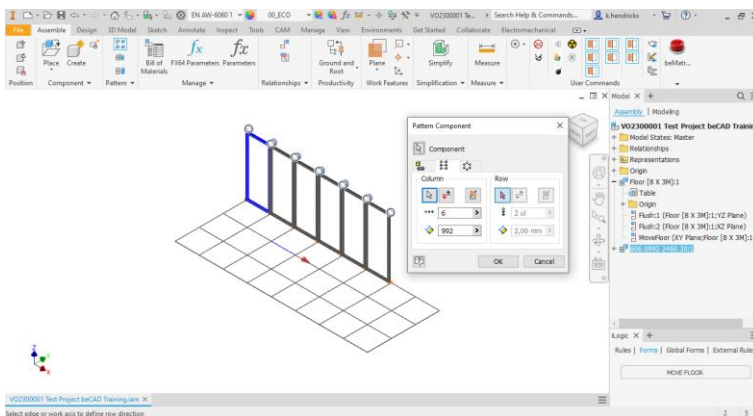


Select the corner of the floor and confirm the move by **right clicking** and choosing **Continue** and again choosing **OK**



With our first frame set up in the correct space, we'll want to quickly create a full wall of frames. We do this by using the Pattern feature.

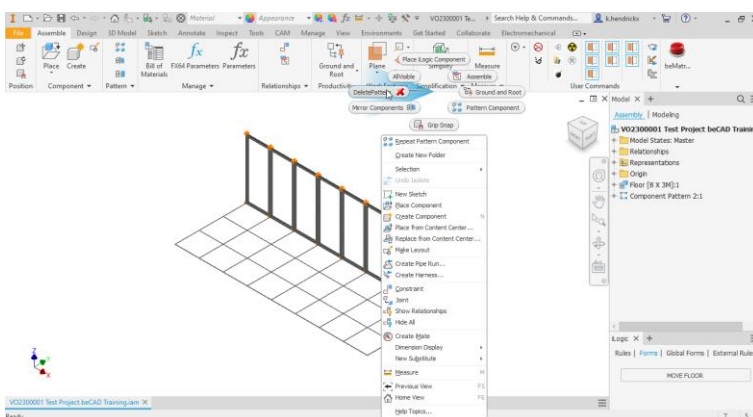
Select the frame and choose **Pattern Components**.



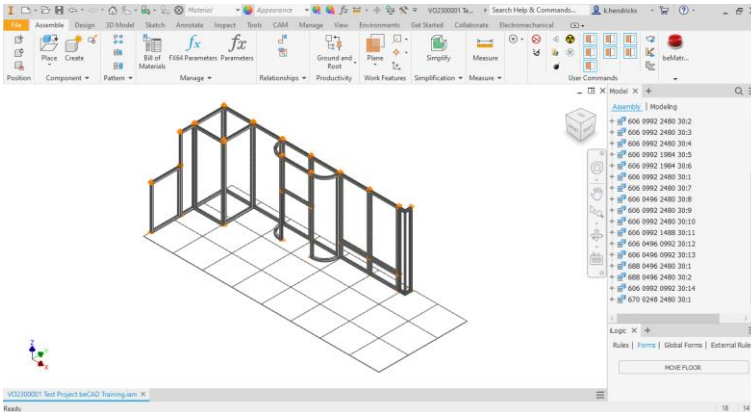
In the dialogue box, first choose the **direction** of the pattern (you can use the edges on the floor as a quick way to select the X and Y directions), the **number of occurrences** and the **distance** between each occurrence.

It's also possible to enter formula's in these parameters.

Click **OK** to confirm the feature.



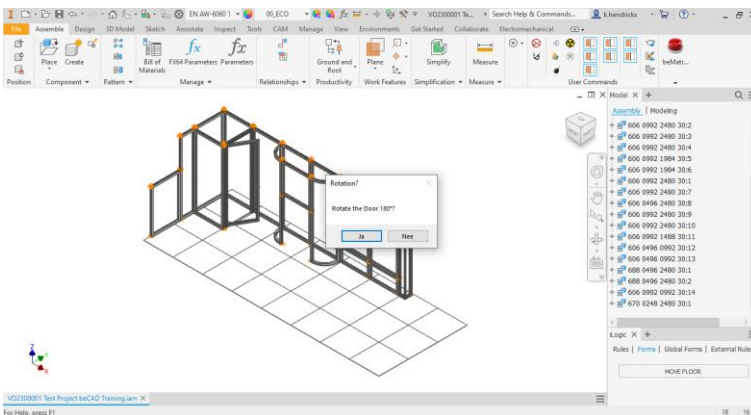
In order to explode the pattern so each frame becomes an individual component again, **right click** on an empty space and choose **Delete Pattern**.



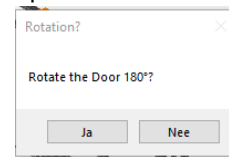
We can now start copy and pasting the frames. Use Grid Snap to place them in the correct spot. While using Grid Snap you can use the workpoints to snap to holes of adjacent frames.

Remember to use the Flip X,Y,Z hotkeys to quickly change the orientation of your frames.

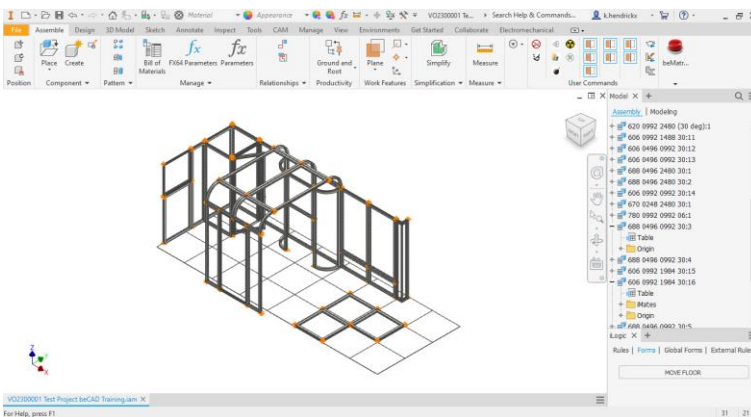
Use Change Element (hotkey **W**) to change the sizes of your frames.



Instead of manually placing a doorframe, it's also possible to replace a standard 992mm wide frame by a doorframe using 1 of the automated tools. Select the right frame of the little storage and use the PlaceDoor tool. The frame will now be replaced by a doorframe of the same height. You will also be given the option to rotate the door 180° if required.

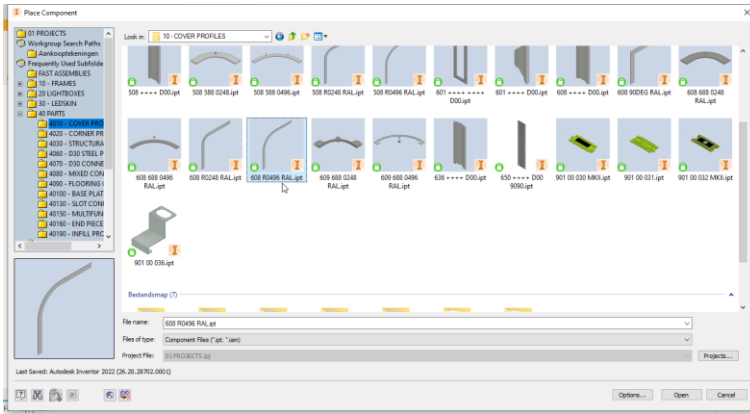


Let's continue placing additional frames.

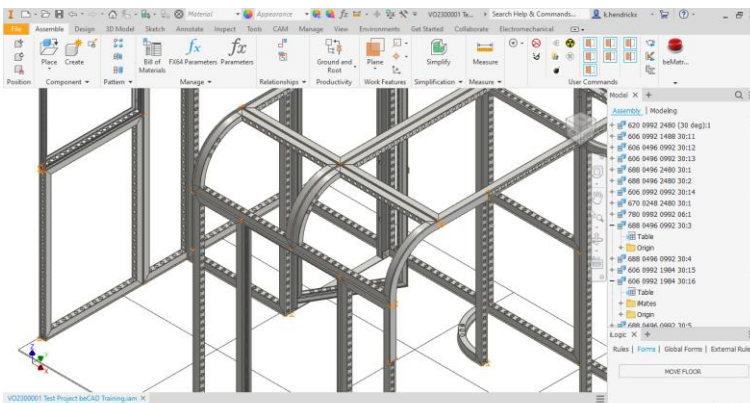


Once all our frames are in place it's time to start with the finishing of our stand. We will first start by placing steel covers, then regular covers, baseplates, shelves, flatscreen support and universal brackets. Finally we will add a ramp to the floor element.

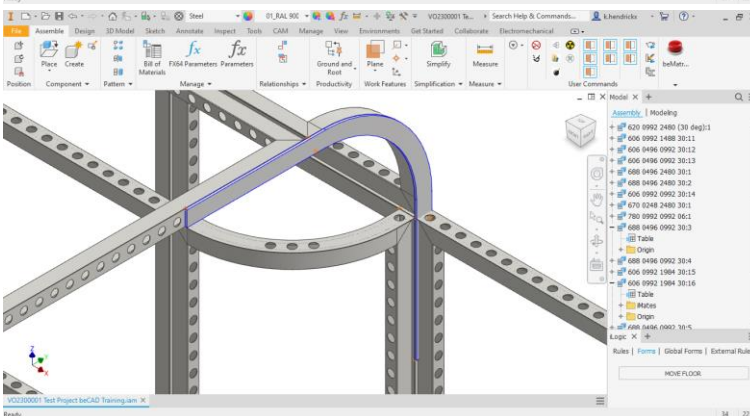
This is also a good time to save our work. Remember Inventor is still a program and can crash, make sure to regularly save your progress since Inventor has NO autosave or backup functionality.



Use the Place Component action and navigate to the **40 PARTS\10 – COVER PROFILES** folder and select the *608 R0496 RAL.ipt* file. Again use Grid Snap to place it in the correct position.



When we look at the other end of this arch we run in to a problem. We should place a steel cover here as well to make the arch more stable, but we do not have a standard part for this.



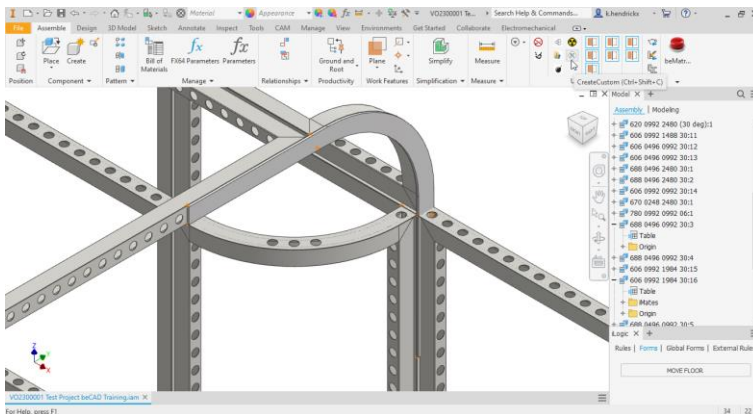
When you need a custom made part which is closely based on a standard part it's easiest to use the Create Custom tool.

This tool will take the library file and make a local copy of it in the folder of your project. Since this is no longer a library file, it becomes fully editable.

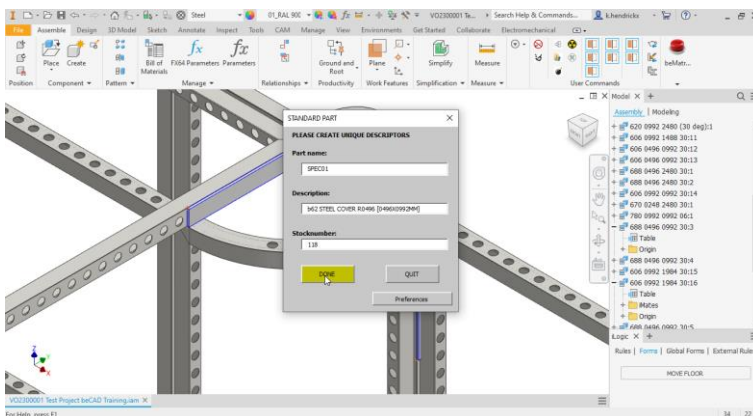
Keep in mind that when creating a custom item in this way, you are limited to what is possible to change. If your custom item is too different it will be better to create it from scratch.

Custom made parts based on library parts

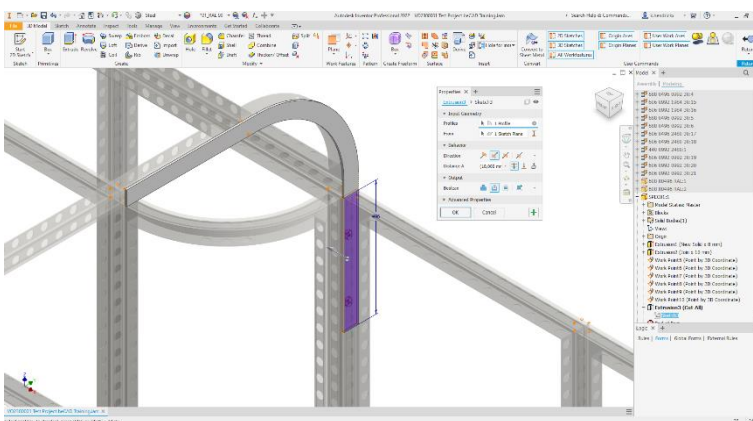
When creating a custom part based on a library part, we will first place the library part and then create a local custom or variant of it.



To solve our problem with the steel cover, first place the standard steel cover, select it and use the **Create Custom** tool.



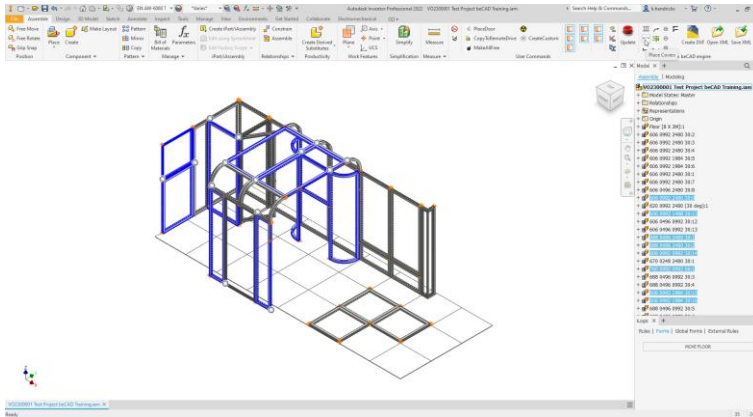
In the dialogue window, enter a new part name for our local copy and change the description to specify the new part.



After clicking **OK**, a local copy will be made and Inventor will immediately open the component to edit it. In this case we want to shorten one of the legs by 496mm. **Create a new sketch** and draw a rectangle over the section that we want to remove. Use the **Cut** feature to remove the material.

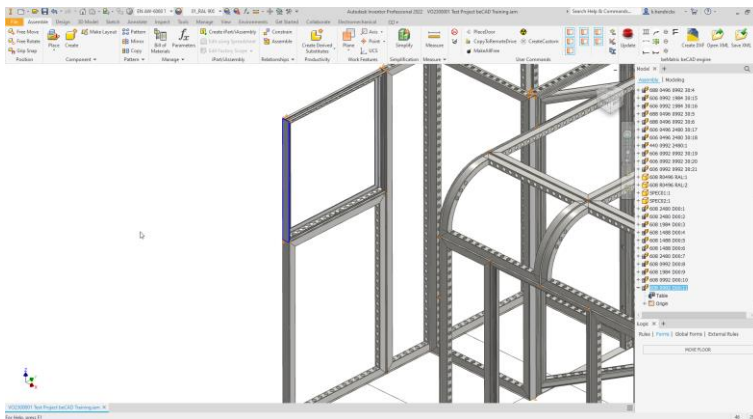
Do the same for the other side.

Now that we have placed our custom part we can start placing covers. You can quickly place covers by selecting frames and using the Place Covers tool. The tool will take in account already present covers as well as the location of the selected frame. A frame placed on the ground will receive different covers then a hanging frame. This is also why we first placed the steel covers, because these will be taken in to account when using the tool.



Select the frames where we want to add covers and use the **Place Cover** tool.

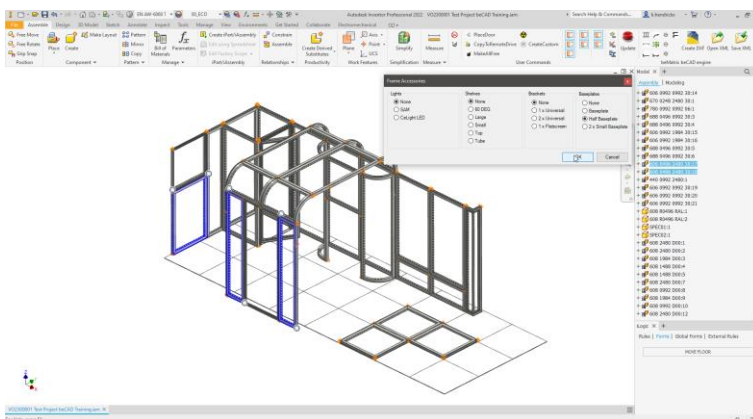
Note that you can select multiple frames even if they are not adjacent to eachother.



There are still some places where the tool doesn't (yet) place covers, here we will have to add the manually.

Join the 2 covers by selecting them and using the **Join Covers** tool. This will join the covers and replace them by one part.

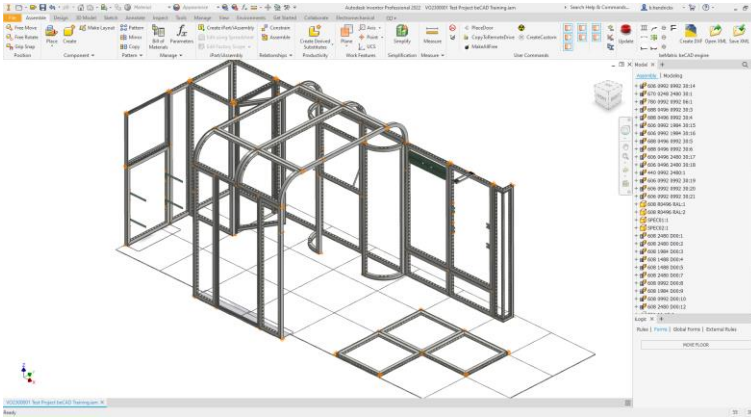
Now we will add some baseplates to ensure that all the walls will be stable. We will add some baseplates under the left side wall and underneath the two small frames at the arch.



We can do this by either manually placing the baseplates or by using the **Place Accessories** tool.

Select **Half Baseplate**, click **OK** andchange the **orientation** if required.

We can use the same tool to place SAM light, all kind of shelves and brackets.



Use the tool again to place some around our booth.
Again notice that it's possible to select multiple frame, but also multiple accessories.

Once we placed our accessories it's time to move further.

Custom made parts from scratch

Since it's possible to use Inventor as a hybrid system you are no longer limited to our parts alone. It's possible to create anything from scratch using Inventor. Keep in mind that some more advanced knowledge is required to create these items.

There are numerous tutorials online (YouTube, Autodesk,...) that will help you in this process.

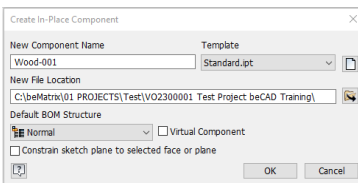
For this training I will show you how to create two parts that might come in handy when designing your booth.

- a wooden board to be used with the universal brackets.
- a textile with a print applied to cover multiple frames.

A wooden board

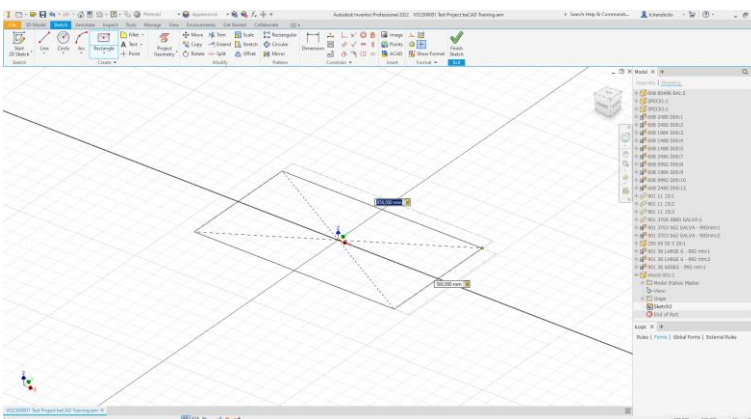


We begin by creating a new part.

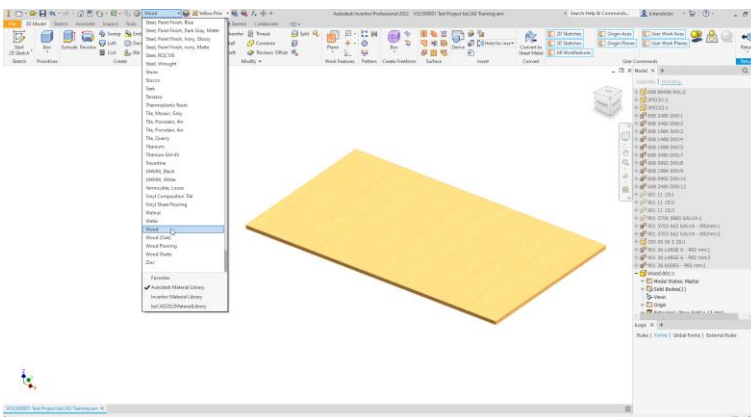


Use an easy to recognize filename and make sure the new file location is within our project.

Click OK and click on an empty space to create our new part.

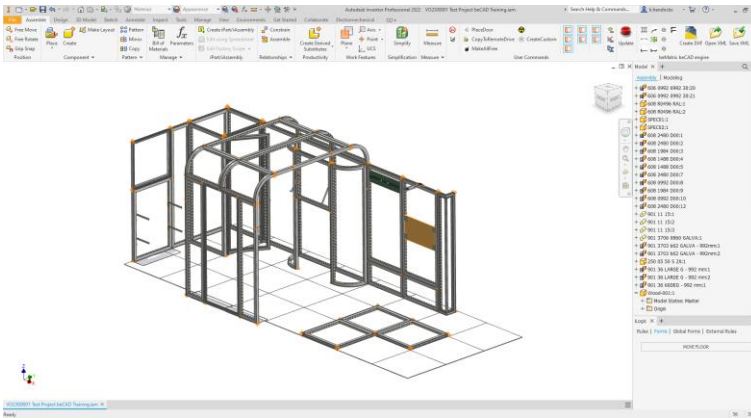


Create a new sketch with a rectangle with a 856 x 500mm size and extrude it with a thickness of 10mm.



We can change the material to wood using the material editor and choosing Wood. The appearance will be automatically set to the default appearance of this material.

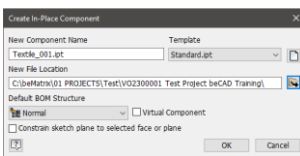
If required, it's always possible to overwrite the appearance by selecting a different one from the appearance editor.



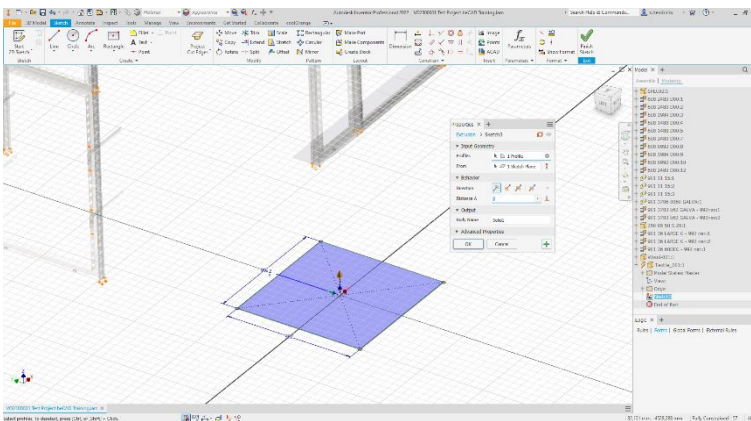
Now our part is finish we can return to our project and use Grid Snap again to position it behind the universal brackets.

A Custom Textile

It's possible to create textiles and add prints to them as well.

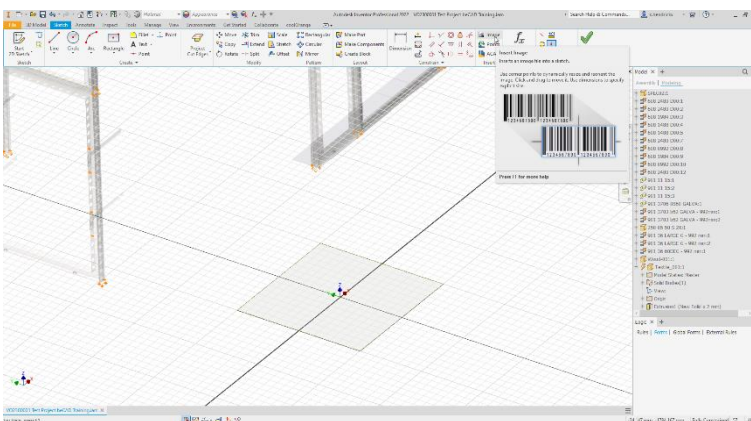


Start by **creating a new part** in our project again.



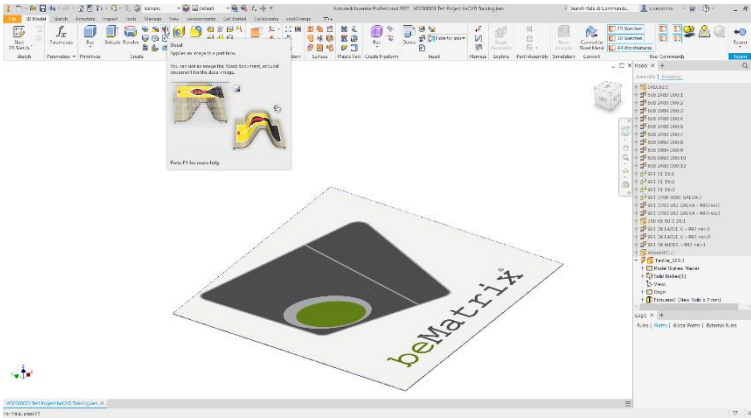
Create a sketch of 992x992mm and extrude it by 2 mm.

It's also possible to sketch out the shape of the textile and extrude it in the height.



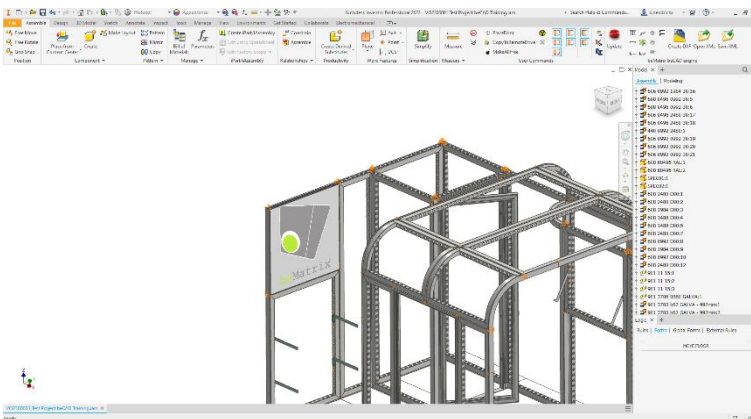
Create a new sketch on the face of the textile and choose **Insert Image**

Place and resize the image so that it fits over the square shape.



Finish the sketch and choose **Decal**

Select to face you want to apply the decal to and Inventor will automatically crop the image and makes it follow the shape of the face.



Position the textile on our small lightbox to finish it off.

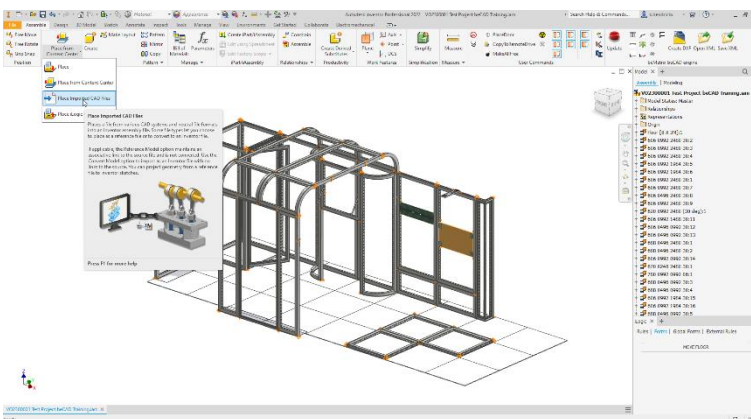
Import CAD files

Next to Inventor files which you can place from the library or create yourself, it's also possible to import different CAD files, such as STP, DWG, STL,... files.

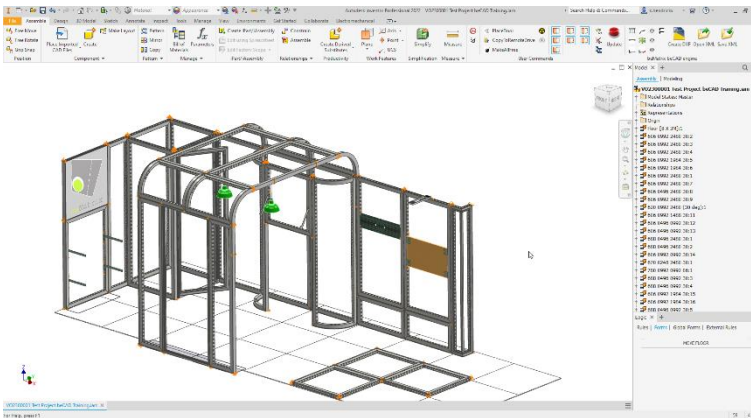
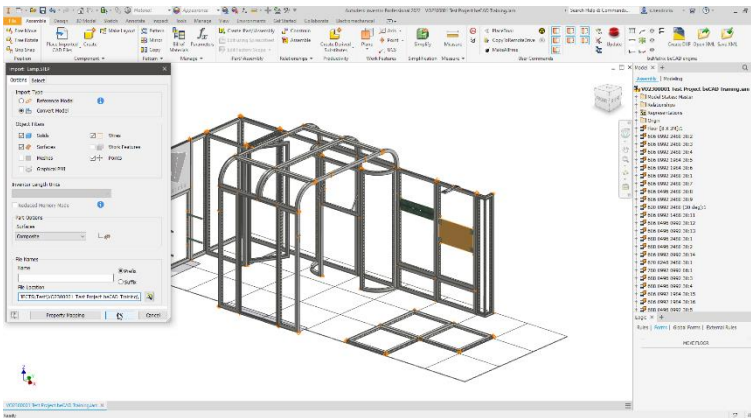
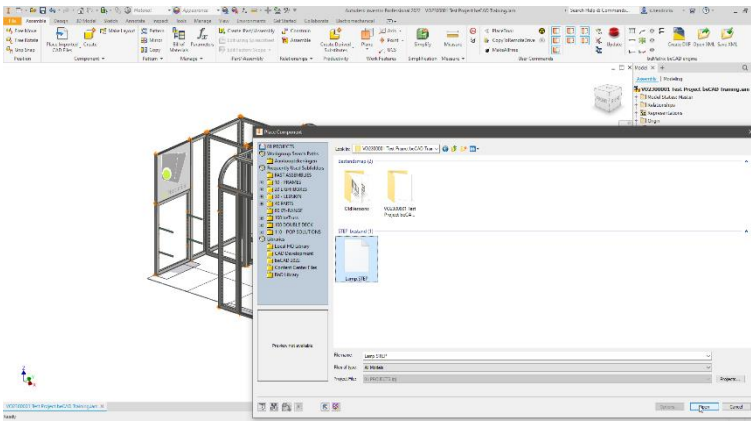
There are numerous places where you can find and download 3D files.

- <https://grabcad.com/library>
- <https://fabory.partcommunity.com/3d-cad-models/sso/>
- <https://archive3d.net/>
- ...

I suggest trying to use STP files as these will carry the most information.



Simple choose **Import CAD Files** under **Place** and select your CAD file.



Calculate Connectors

Now that we have finished placing our frames, accessories and custom parts, it's time to use the update button to have Inventor calculate the connectors for our stand.

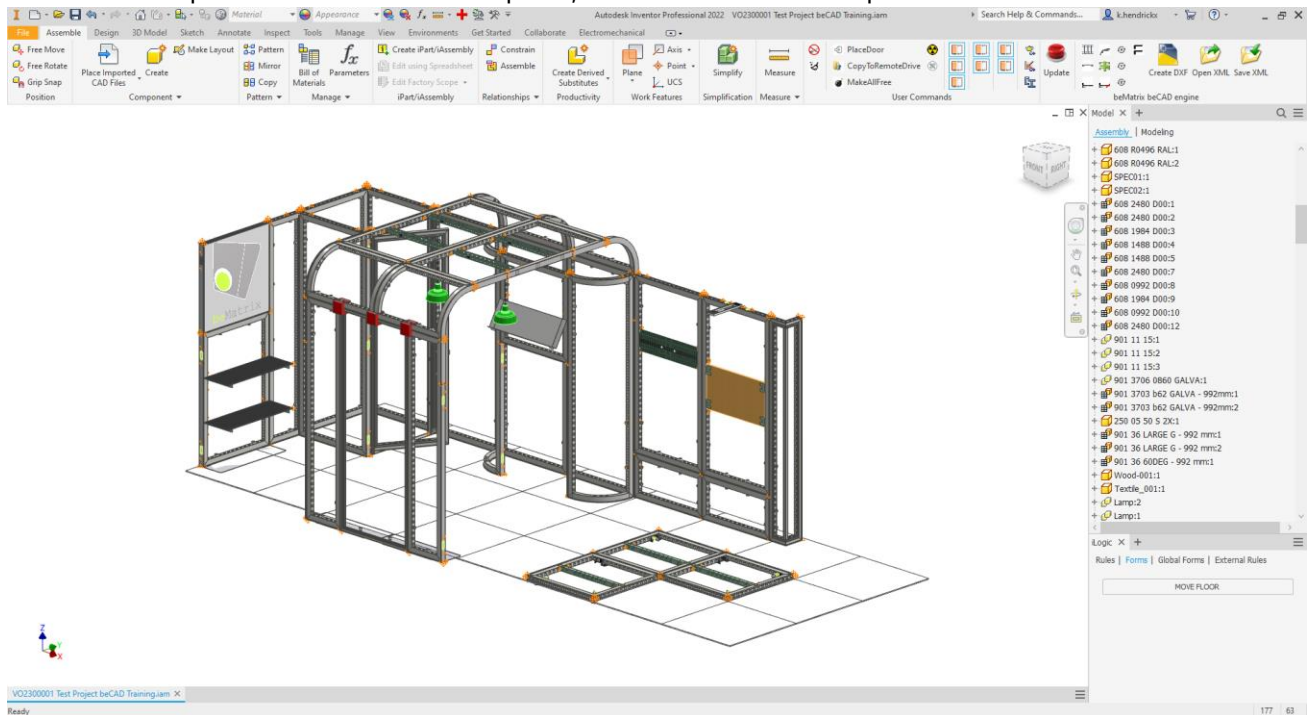
This process is completely automated and will take in to account the basic rules to correctly connect frames depending on which type of connection is required.

The Update tool has the exact same backbone and rules as the one available on the beCAD online platform.

As with every automated process it's very important to always review the connections and make changes where it's needed. In some cases new parts and frames are not yet included in the calculation and therefore will also not correctly be included in the process, but rest a sure, the tool is constantly being updated to include these connection as soon as possible.

Please note that custom parts that are made from scratch will never be able to be included since the possibilities here are endless and these take thorough review to include all the required connections.

Custom made parts based on standard parts, will be included in the process based on their standard



counterparts.

The main rules are the following:

- At each straight connection 2 double clamps and a 180° PIN without bolts on the bottom
- At each L-connection 2x 90° PINS at the top and bottom with M8 bolts on all sides
- At each T-connection 2x T6 PINS at the top and bottom with M8 bolts on all sides
- All horizontal covers are to be fixed with the M8 + COVER SLIDER. Vertical covers are placed with the green cover clips
- Each 3m there has to be either a return frame or baseplate depending on the height of the wall. Additional ballast can be required.



Please check all our manuals on the website for more information regarding the rules to build a safe, stable and strong stand.

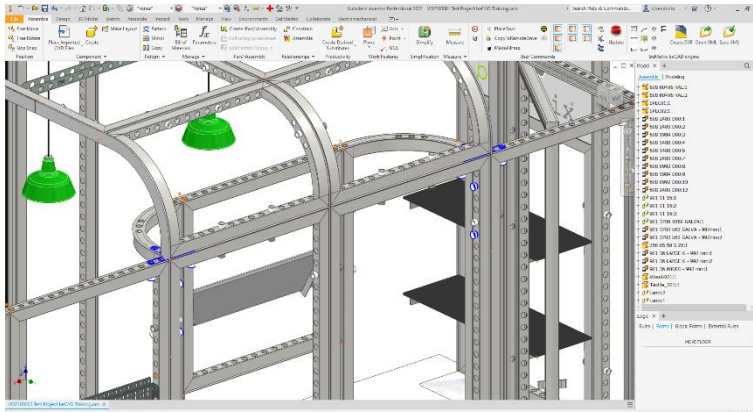
<https://bematrix.com/eu/en/library?f%5B0%5D=downloadtype%3A372>

Review and change automated connections

Once the connectors are placed, we will review the connections and take a closer look at some of the locations where the update tool failed to place a valid connection.

In our design there are 2 locations where the update tool failed to place the correct connections.

The first one is at the back of the arch.

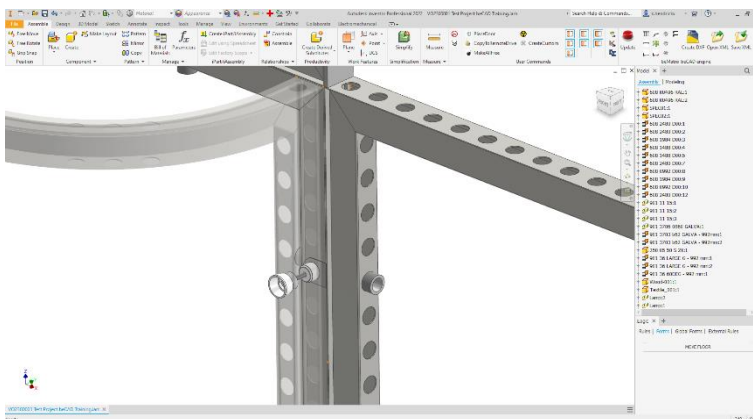


connectors.

Here the update tool placed 4 M8 pins to much. The reason why is because it placed these based on the standard steelcover. In this case these are not needed and can just be deleted.

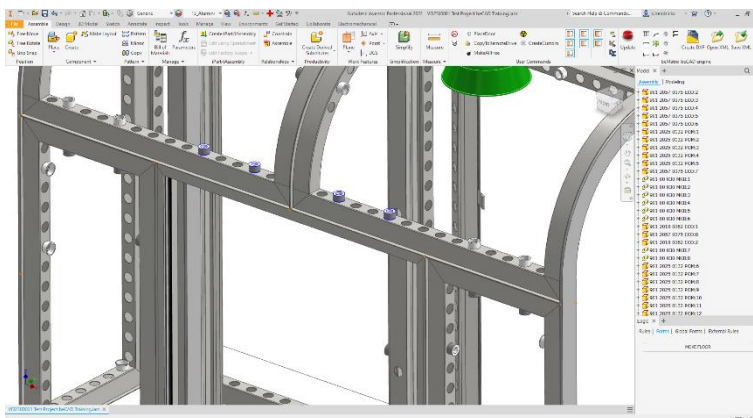
The tool also placed 2x T6 PINS at the top of the wall, we will have to replace this connection by a different one to connect the curved frame to the backwall.

First begin by deleting the unnecessary



We will replace the T PINS by a T-CONNECTION BASE D30 with a M8 PIN that will be positioned over the already present double clamp in the backwall. Keep in mind that there are multiple ways to solve this problem, but always try to look for a solution that will grant a strong and stable stand.

At the front the Update tool placed 3 red block to indicate that it failed to place a connection in this area. The reason here is because the glass frame is still a very new product and not yet fully incorporated in the Update process at this time.



We will connect the two curved frames to the glass profile by using 4 TOOLLESS D30-SLOT CONNECTORS.

Subassemblies – Creating Structure

At this point we've placed frames, accessories, custom parts and calculated and review all our connections.

All parts however are still placed loose in our project assembly. In order to create more structure we will make selections and place those parts together in 1 subassembly.

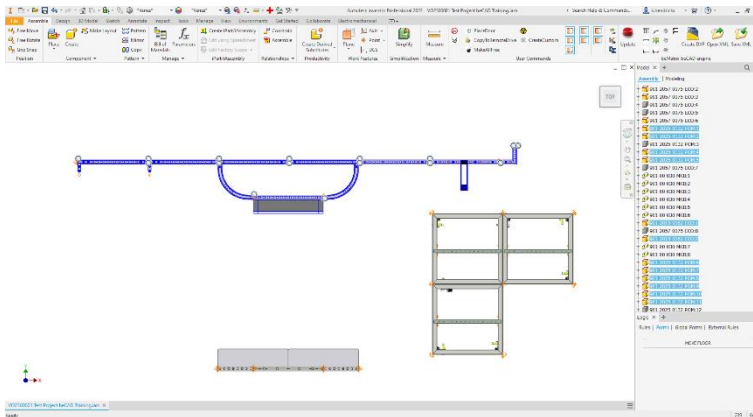
Often used subassemblies are :

- Backwall,
- Frontwall
- Storage
- Arch
- ...

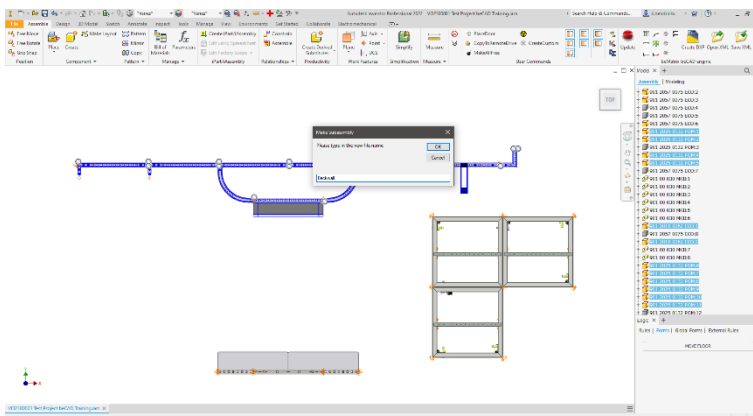
This will divide our design up and makes it easier to create and understand the 2D drawings in a later stage.

Also in big projects this helps with performance as you can work on independent subassemblies without having to open the full design. You can even split the work by working with multiple people with each in its own subassembly.

Since Inventor is a reference based program, all changes made within the subassemblies will also be applied in any assembly containing these subassemblies.



Let's begin by selecting all the part that make up the backwall. In complex projects it can help to hide certain sections that would otherwise be in the way.



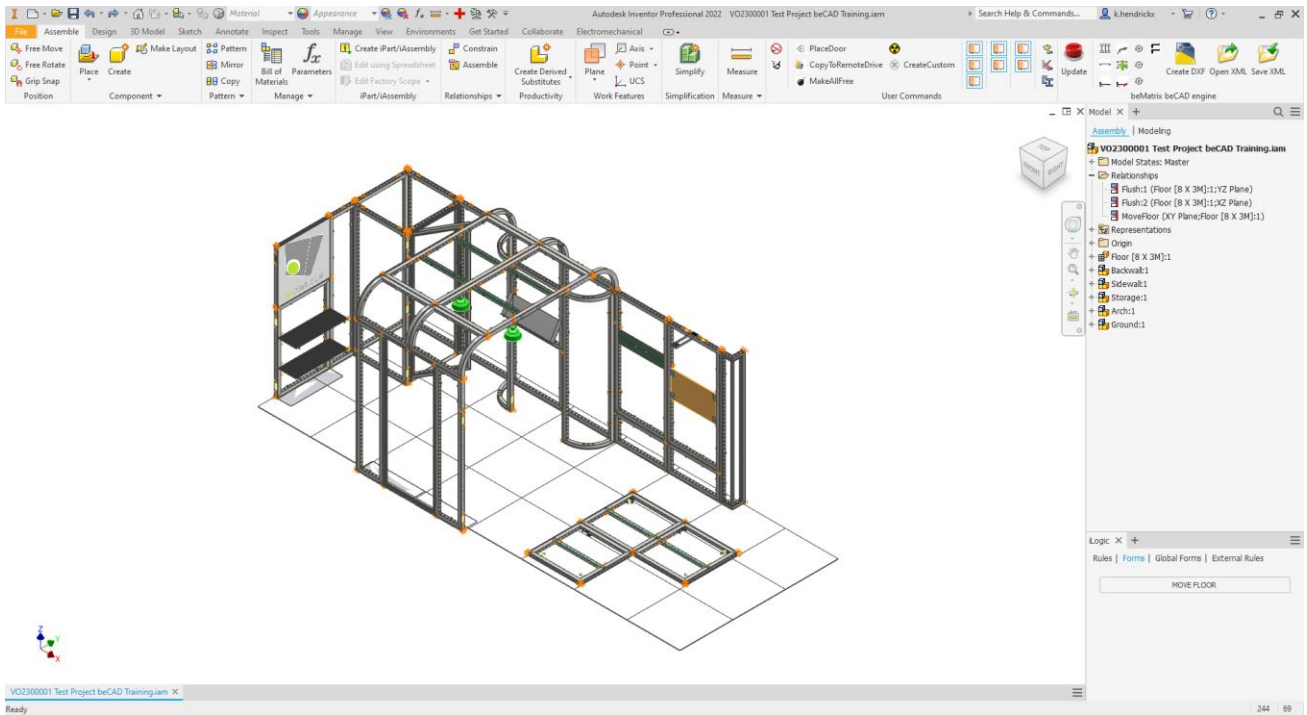
Once we have the parts selected, press the TAB key on your keyboard.

Enter the name of the subassembly.

If you want to place the subassembly in a subfolder of the project folder, start with the folders name followed by a "1" and then the assemblies name.). The selected parts will now be grouped together.

Continue with the other sections. We will create the following subassemblies:

- Backwall
- Sidewall
- Storage
- Arch
- Ground

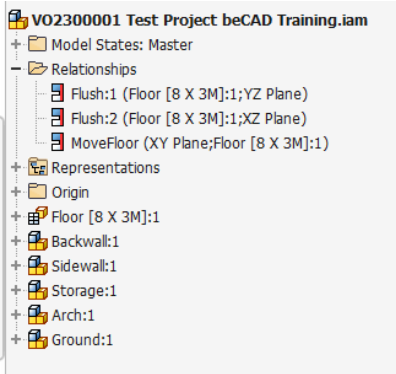


If you want to move parts from one assembly to another you can select the parts and use CTRL + TAB to choose a different subassembly.

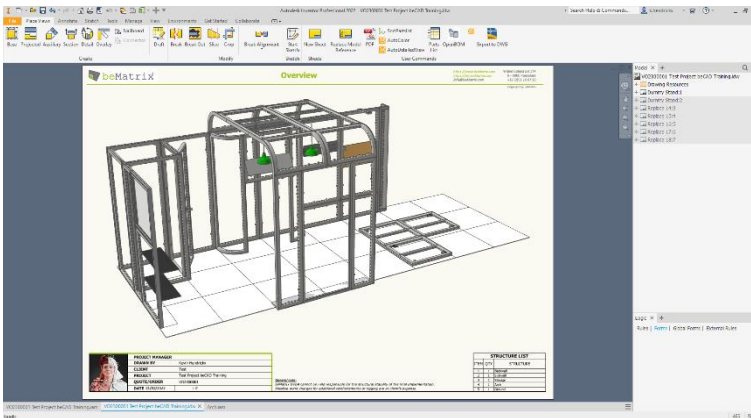
2D Plans

Once the 3D model is complete we are ready to start creating our 2D drawings based on these models. Again the reference based model is used so that any changes we make in our model, will automatically update in our 2D drawings.

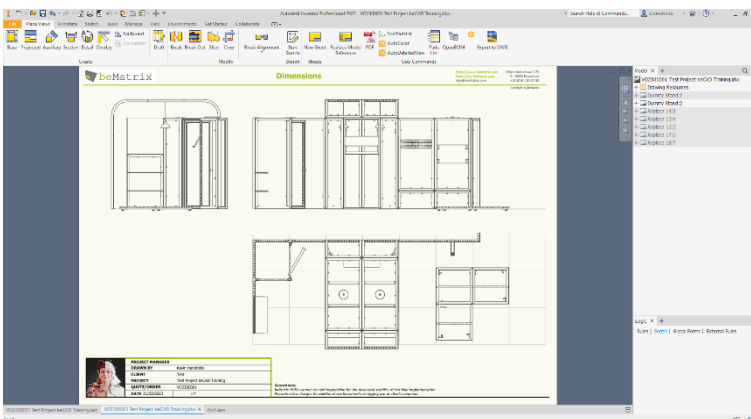
Before we start please double check that aside from the floor object, ALL other parts are inside a subassembly.



Use the **AutoDrawing** tool to start creating the drawings. As you will notice, most of it will run automatically.



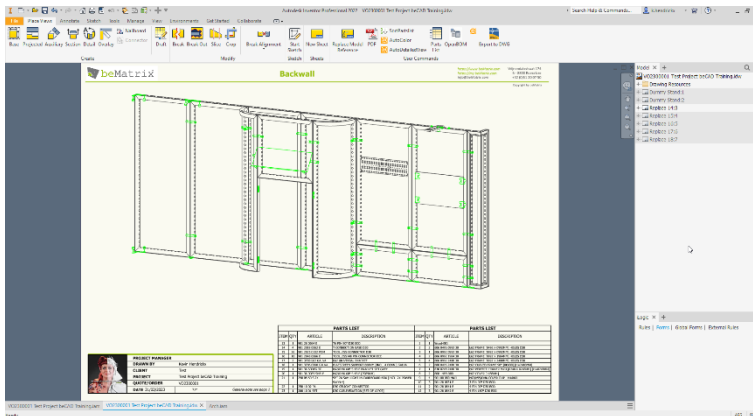
The first page will be an overview of our design with a structure list of all the subassemblies.



The second page is an overview page with a front, side and top view of our design.

You can use this page to place all the dimensions that you need for your panels or textiles.

You can also generate sections or cut-out views on this page or drag it to a new sheet to have it linked to each other.

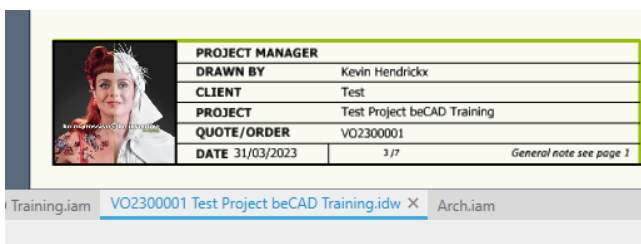


The remaining pages are for each subassembly within the project.

The Autodrawing tool will generate a view based on the subassemblies we created earlier, place and sort a parts list for that section and finally highlight all the connectors and smaller parts in a green overlay making them more visible.

You can use these pages to place balloons identifying the frames and parts with their corresponding item number of the partslist.

You can also place detailed views of parts to have a more detailed view of the connection.



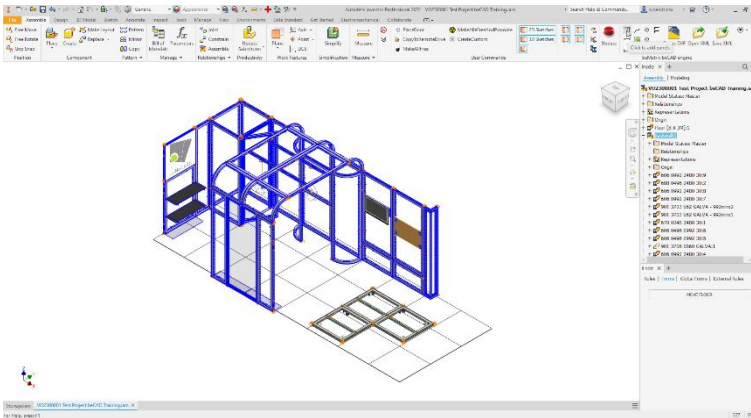
If you study the title section on every page, you'll notice certain fields are already filled in with the information we've provided at the start of our project.

- Your name based on the name provided in the application options
- Your clients name based on the client folder we selected at the start of the project
- The Project name and number by the information given at the start of the project

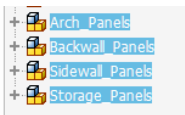
When everything is finished, use the export to PDF function to generate a pdf drawing. The drawing will be placed by default in the same location as that Inventor drawing file.

Adding Panels

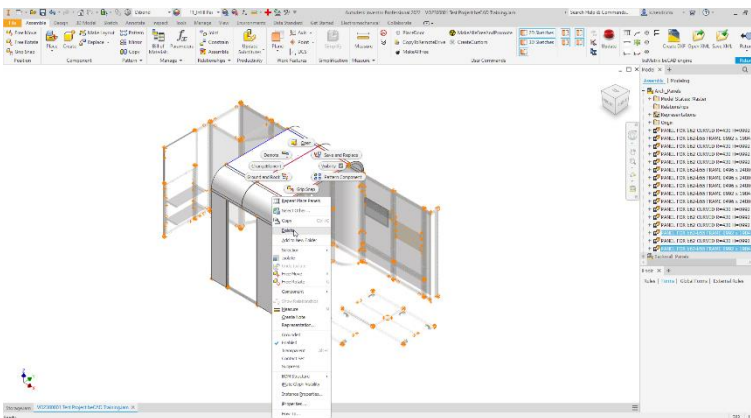
Now that we have the structural side finished, we can start looking at how we are going to finish our stand.



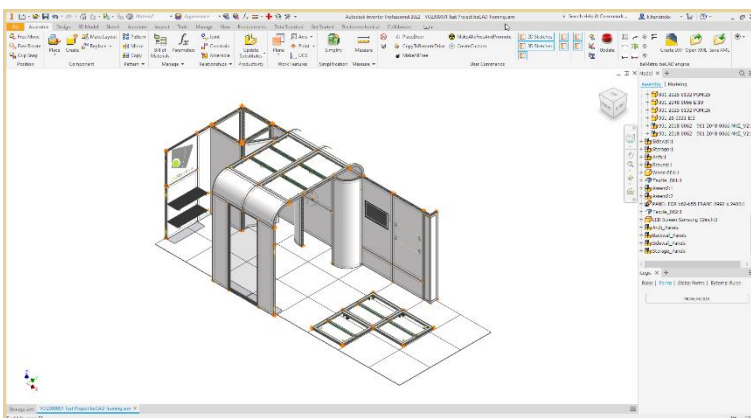
The Inventor beCAD plugin offers you the ability to automatically place infill panels on the frames that you select. You can select either a single or multiple frames, or whole subassemblies.



When using the Place Panel command, Inventor will create a new subassembly and place the panels according to the matching frames.



After using the command, you can delete the panels that you do not need (for example to top side of a roof, or the inside of a storage).



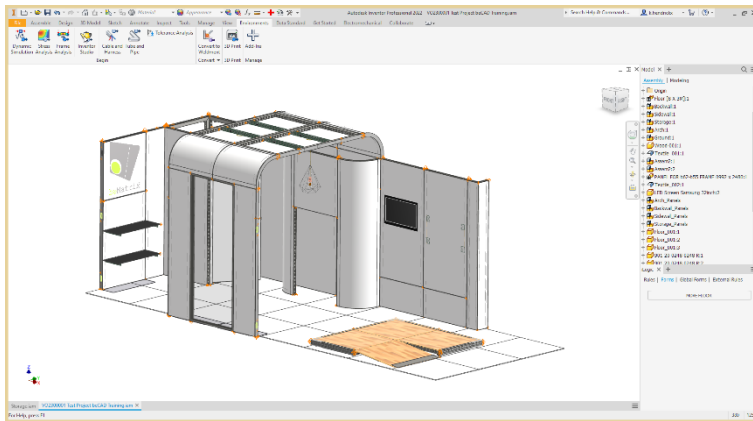
If you want you can use the appearances to override the look of your panels from the standard white to any appearance of your choosing. It's also possible to add decals over one or multiple panels.



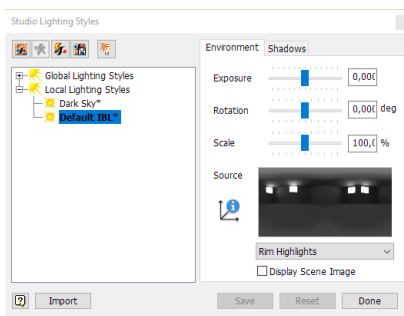
Rendering

Inventor has a basic render engine that enables you to make quick renders from you final design. The quality of these renders depends greatly on:

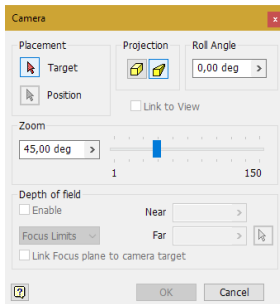
- The materials and appearances that you use
- The amount of time given to the render engine to produce a result
- The level of detail in custom made or imported parts.



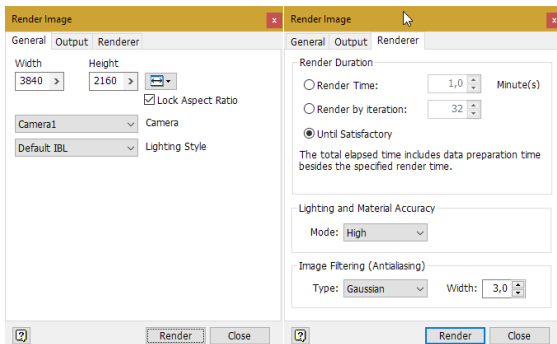
To enable the rendering environment, startup Inventor Studio from the Environments tab.



You can manage the studio lighting styles to suit your lighting and shadow settings



You can create camera's with options for perspective or orthographic view mode, as well as depth of field and position of camera and target.



In the render settings you can choose the resolution, which view to render, and how long you want the render to run. Keep in mind that increasing these settings will result in much longer rendering times.



Please keep in mind that for more advanced renders we highly recommend using more dedicated software such as 3dsMax (included in the Autodesk Production Suite).

Export your design

There are multiple ways to export and share your design.

- Shared Views
- Export CAD
- Pack & Go

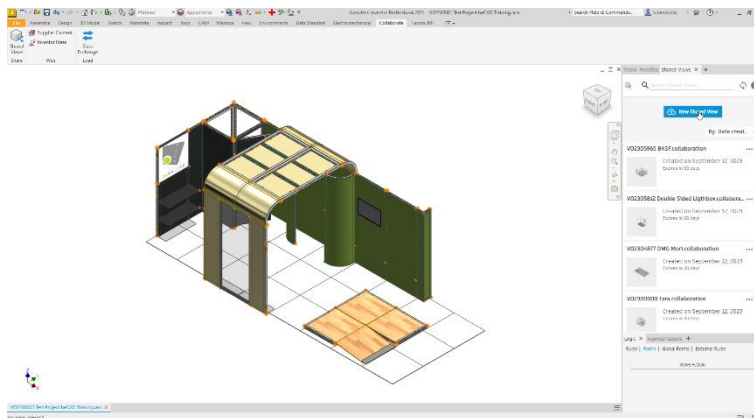
Shared View

Shared view is a cloud based service of Autodesk, where you can export your design directly from within Inventor.

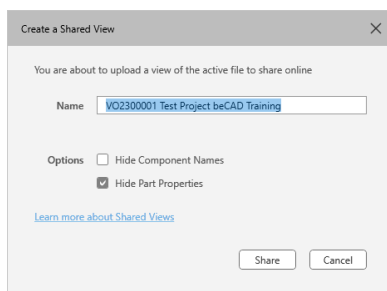
The upload usually takes only a couple of seconds and gives you a weblink which you can share with your client/colleagues.

The link is valid for 30 days after which it gets deleted from the Autodesk server, guaranteeing that your design is safe and protected.

The link can be accessed from any device that has an internet connection and web browser, this means that you can open, view and inspect your design from anywhere and anytime.

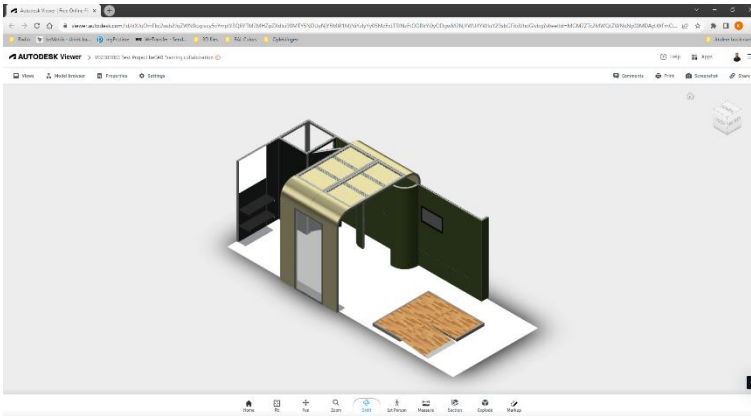


To create a shared view go to the **Collaborate** tab, choose the **Shared Views** tool and create a **New Shared View**.



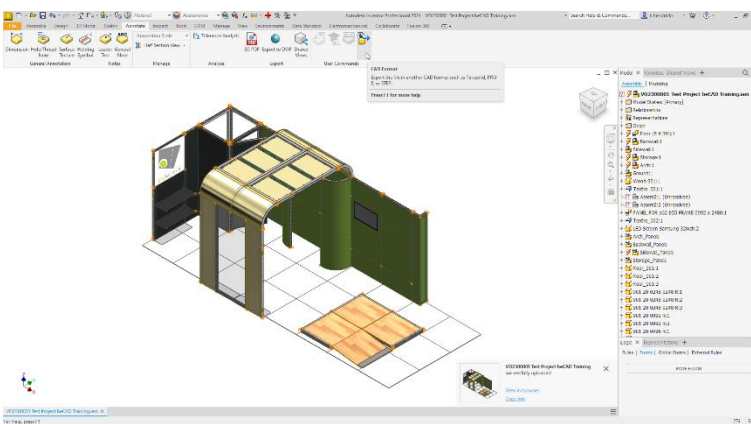
Choose your options and click Share to start uploading your design.

Once finished you can copy the link and share you design with however you want.

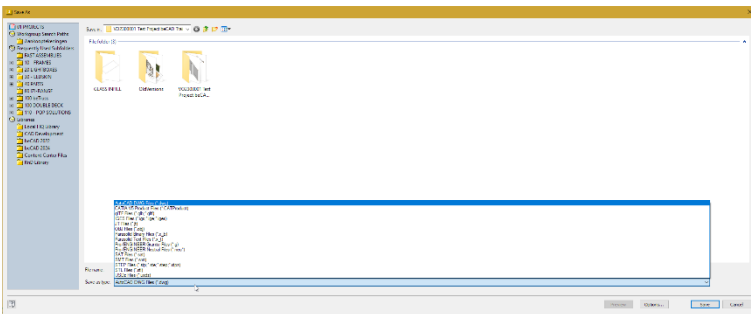


Export CAD

Inventor allows you to export your design to a numerous amount of filetypes.



To export your design choose the **Annotate** tab and then the **CAD Format** tool.



In the dialogue window you can choose to which filetype you want to export. Most common filetypes are DWG and STP files.

Pack&Go

Pack&Go allows you to share your Inventor files with other people who have Inventor but no access to the beCAD library. It packages an Autodesk Inventor file and all of its referenced files under a single location.

Tip & Tricks: Ideal and structured ways of working

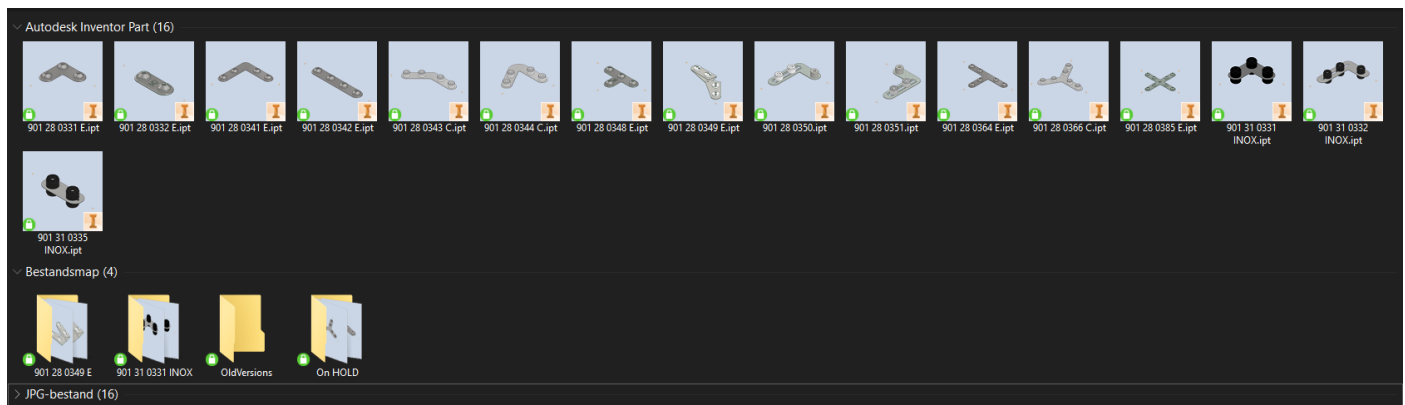
A Clean Working Environment

To be able to work efficiently we like to set up our folders to show large thumbnails of the parts that included in the library. This makes it far easier to identify parts not only by their part number, but also how they look.

To do this we need to navigate to a library folder that contains some parts.

- **Right click** on some empty space and choose **Customize this folder...**
- Go to the **Customize** tab and set the view to **Video** and click **OK**
- **Right click** on some empty space again and choose **View** → **Large Icons**
- Again, **right click** on an empty space and now choose **Group by** → **Type**
- Save this view to the Video preset by going to **File** → **Change folder and search options** → **View** tab → **Apply to folders** → **apply** and **close**
- Go back to the main **beCAD 2024** folder and **right click** on an empty space → **Customize this folder ...** → **Customize** tab → select **Video** and check **Also apply this template to all subfolders**

Your folders should now look like this:



You can now collapse filetypes so you no longer see them.