DIGITAL TWIN FOR EDUCATION

CARL-BENZ-SCHULE GAGGENAU

Mercantec

CREATING A CAD-MODEL FOR A DIGITAL TWIN

MODULE 2

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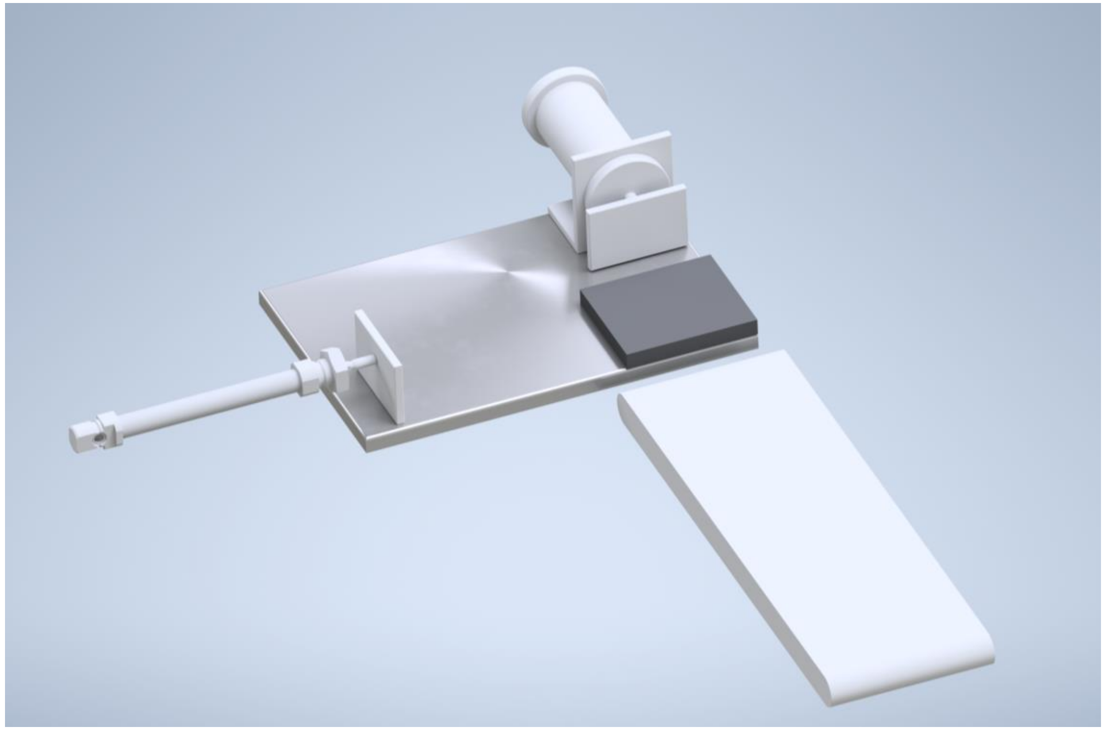

- The Digital Twin consist of a simple 3D model that has been equipped with advanced physical properties.
- The Digital twin also has an interface of mutual communication with the physical Machine.
- In this course we are going to focus on building the 3D model in Inventor.
- When the 3D model is finished, it will be exported to Siemens NX, where the physical properties will be added to the twin.

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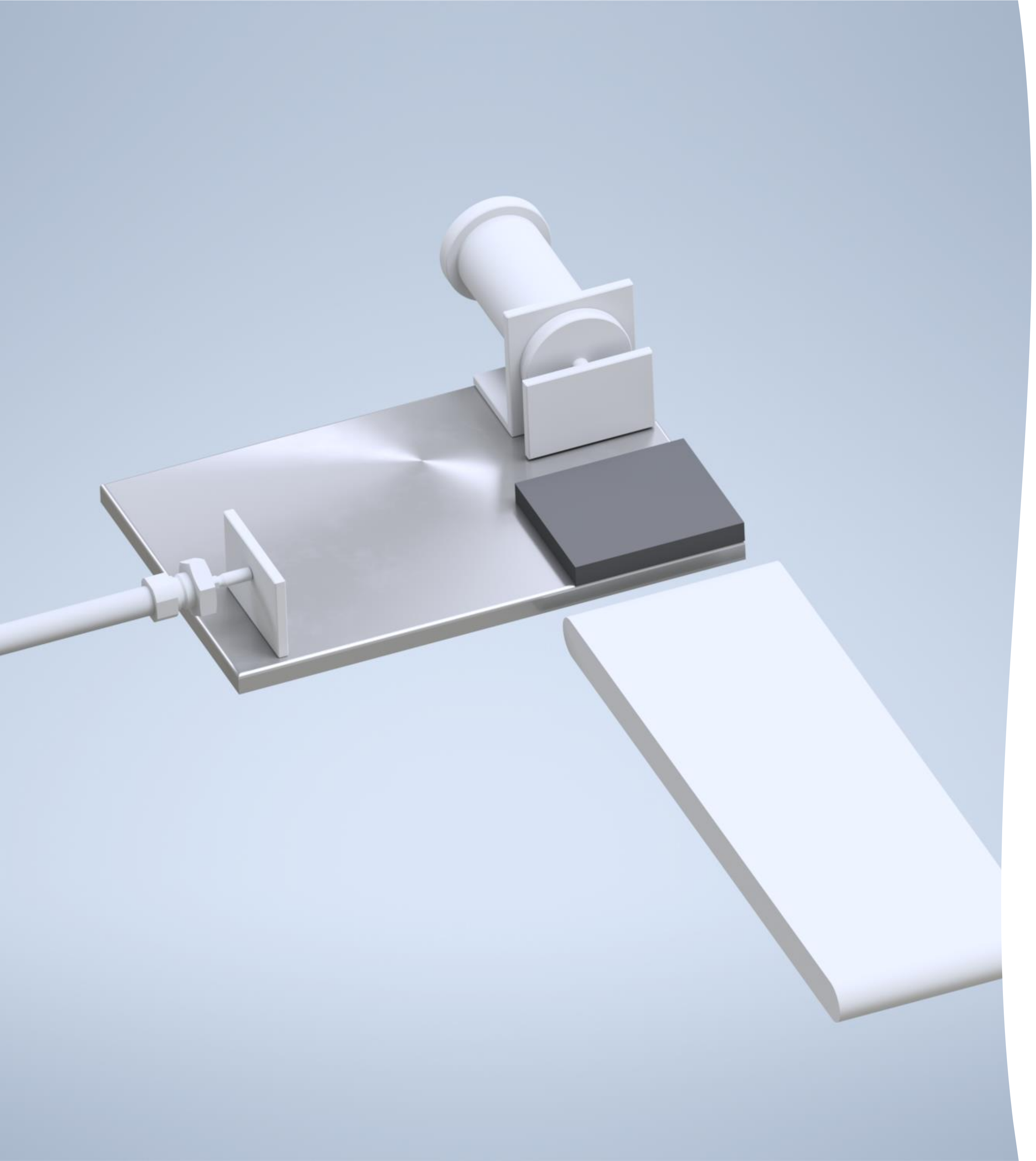

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Content

- Drawing basics
  - Introduction to inventor
  - Create simple box (Machine base)
  - Sketch and Extrude
- Assemblies
  - Create an assembly file
  - Add parts to an assembly
  - Create and modify parts inside an open assembly
- Subassemblies and relationships
  - Creating a Pusher subassembly
  - Connecting parts with relationships
  - Placing the subassembly in the main assembly
- Modelling the conveyor
  - Sketch - Arc
  - Mate relationship with offset
- Import CAD from vendor
- Export to STEP

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Drawing basics (Video 1)

Introduction to Inventor – Project folder, file type, Units and drawing standard

Create a simple 3D box, the machine base plate

Sketch and Extrude

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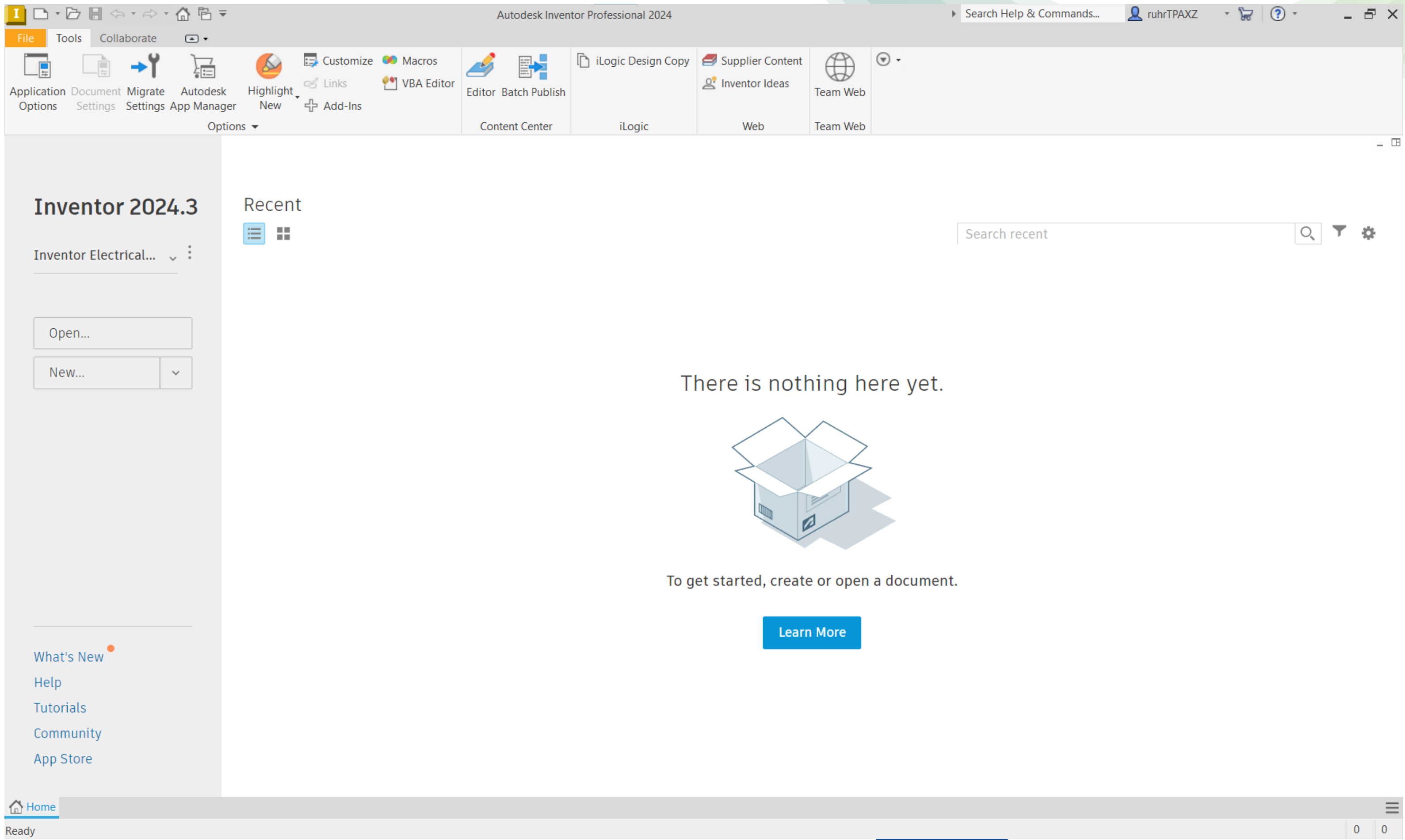
DIGITAL TWIN FOR EDUCATION

Welcome screen

- Project folders
  - Create new project
  - File location
- Application settings
  - Units
  - ISO standard
- Create file
  - Part
  - Assembly
  - Drawing
  - Presentation



DIGITAL TWIN FOR EDUCATION

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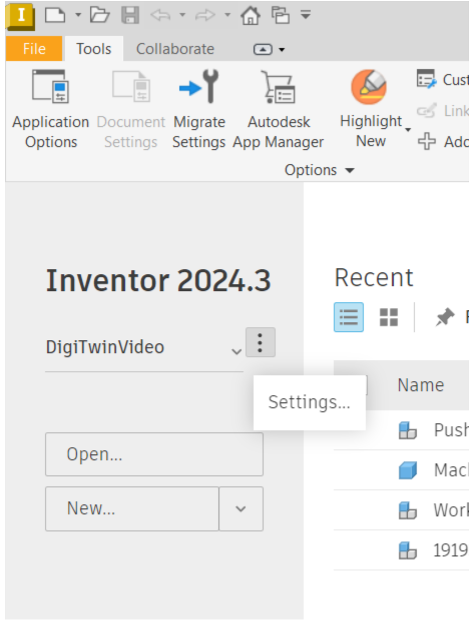
An Inventor project will typically consist of severel folders and files of different types. To avoid making a huge pool of files belonging to all different projects, it is recommended to create a project folder for each project.



Click the project folder menu (3 dots) and open the settings

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Welcome screen

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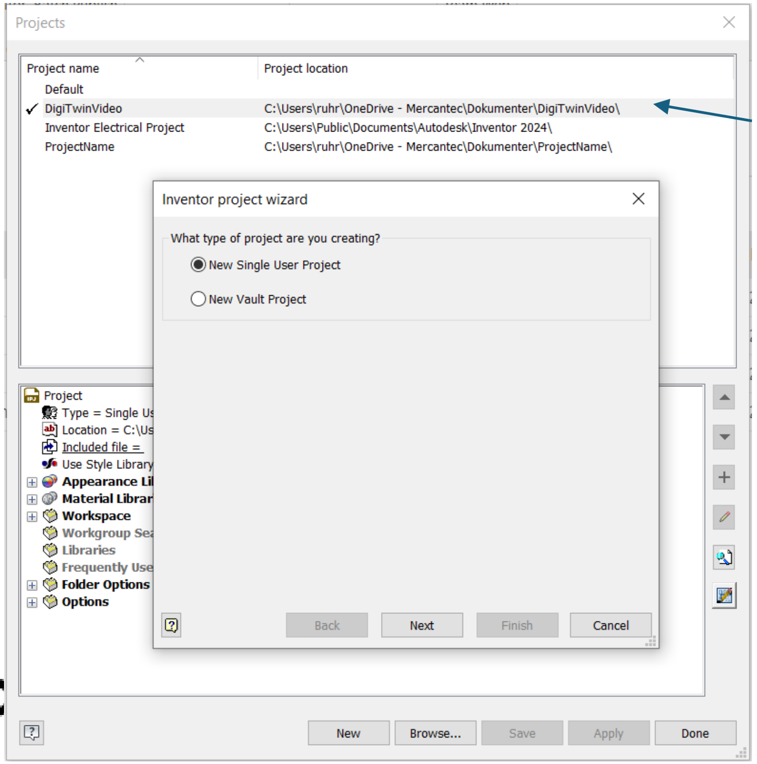

In the top window, the existing project folder and their locations are shown

Click ”New” to create a new project folder

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Welcome screen

- Project folders
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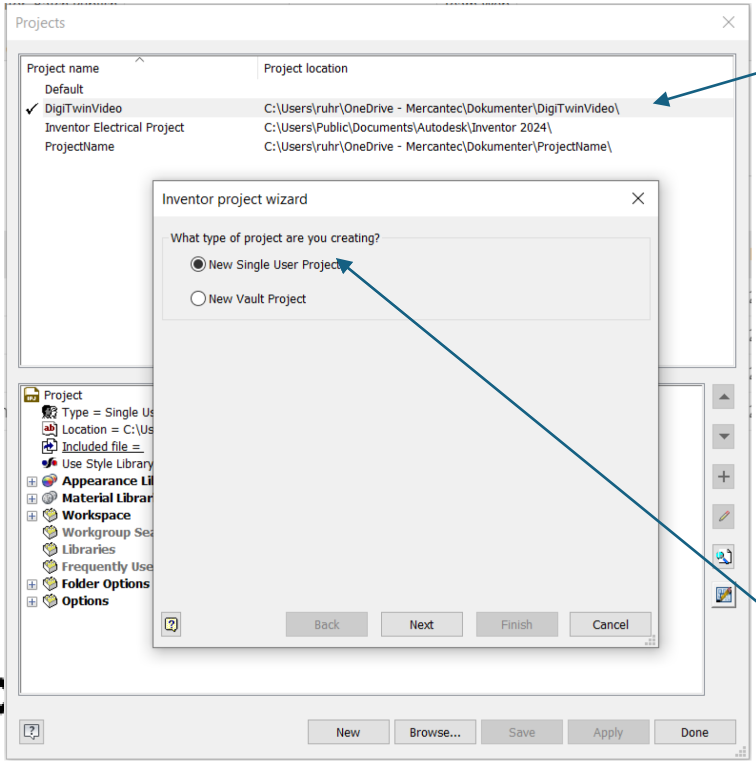

In the top window, the existing project folder and their locations are shown

Click ”New” to create a new project folder

Vault folder is a cloud project for collaboration between multiple user.

Single user is, like the name elaborates very well, for a single user to work on.

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Welcome screen

- Project folders
  - Create new project
  - File location
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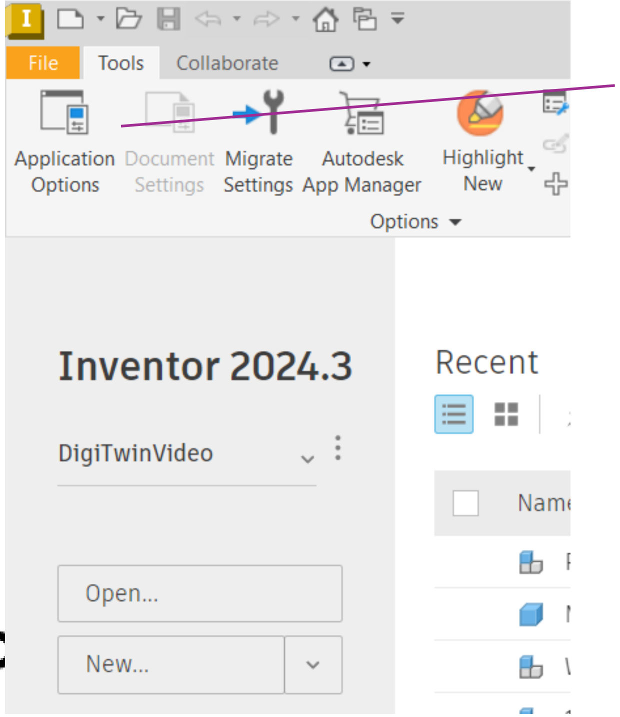
  - Units
  - ISO standard
- Create file
  - Part
  - Assembly
  - Drawing
  - Presentation

Every time we create a file, it is created with some default settings. In this project, we want the documents created in ISO standards and with metric measuring units.
These settings are chosen as default in the following way:



In the top left corner of the welcome screen, press *"Application Options"*

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Welcome screen

- Project folders
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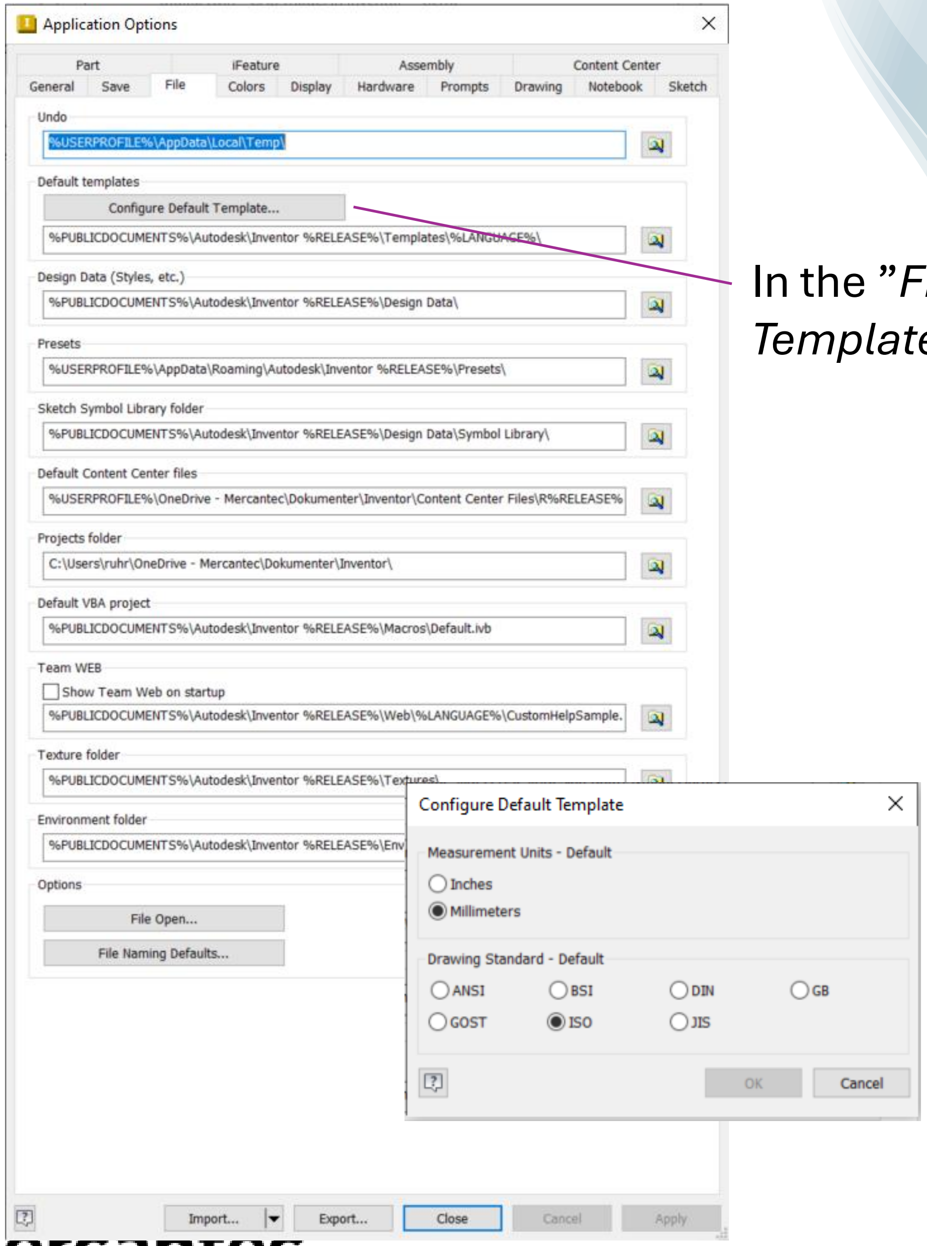

In the *"File"* tab, press *"Configure Default Template"*

I popup ruden kan man vælge måleenhed og default tegningsstandard.
Afslut med *"OK"*

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Welcome screen

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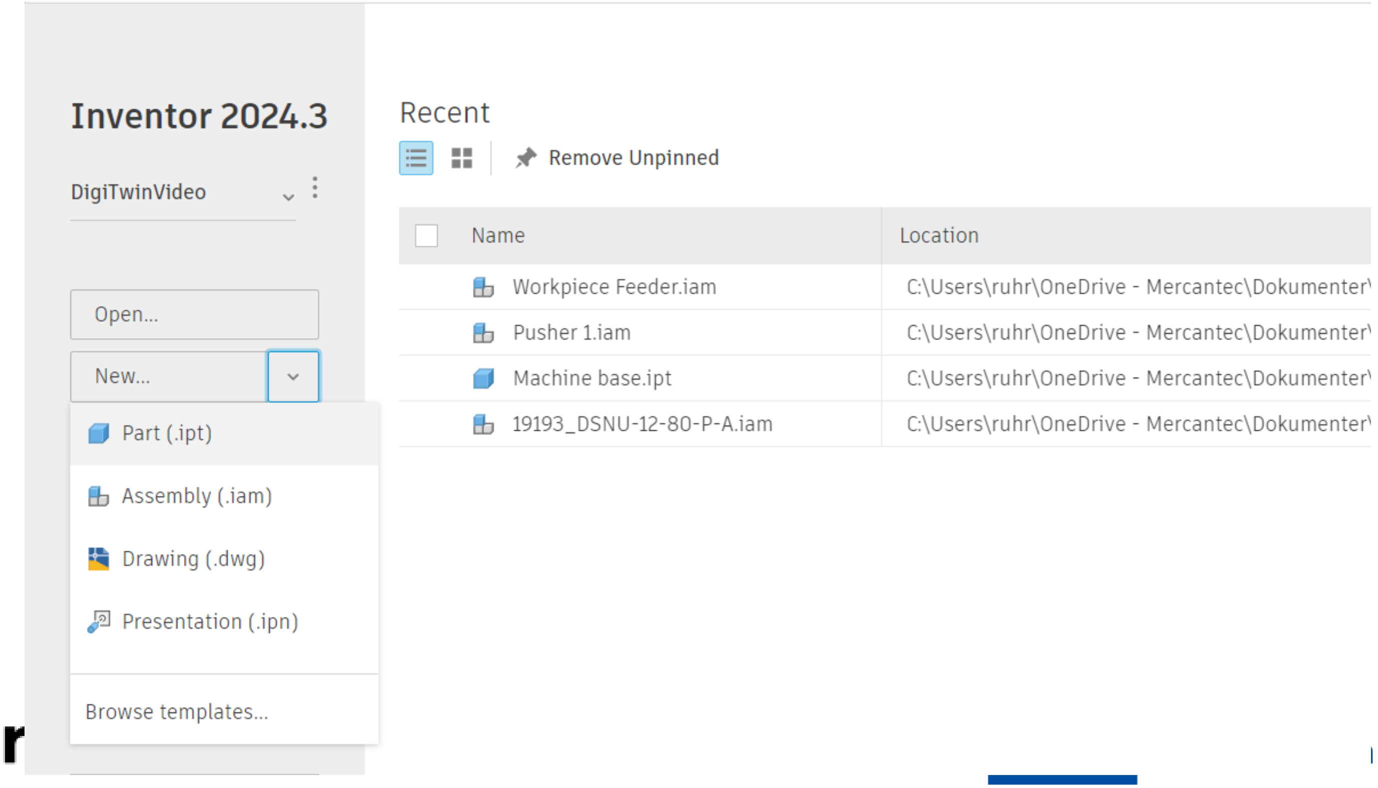
- Project folders
  - Create new project
  - File location
- Application settings
  - Units
  - ISO standard

To create a new Part file from the welcome screen, can be done in two ways. If we open the drop-down menu, we can choose between the four file types. Press *”Part”* and a standard Part file will be created.

- Create file
  - Part
  - Assembly
  - Drawing
  - Presentation



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Welcome screen

- Project folders
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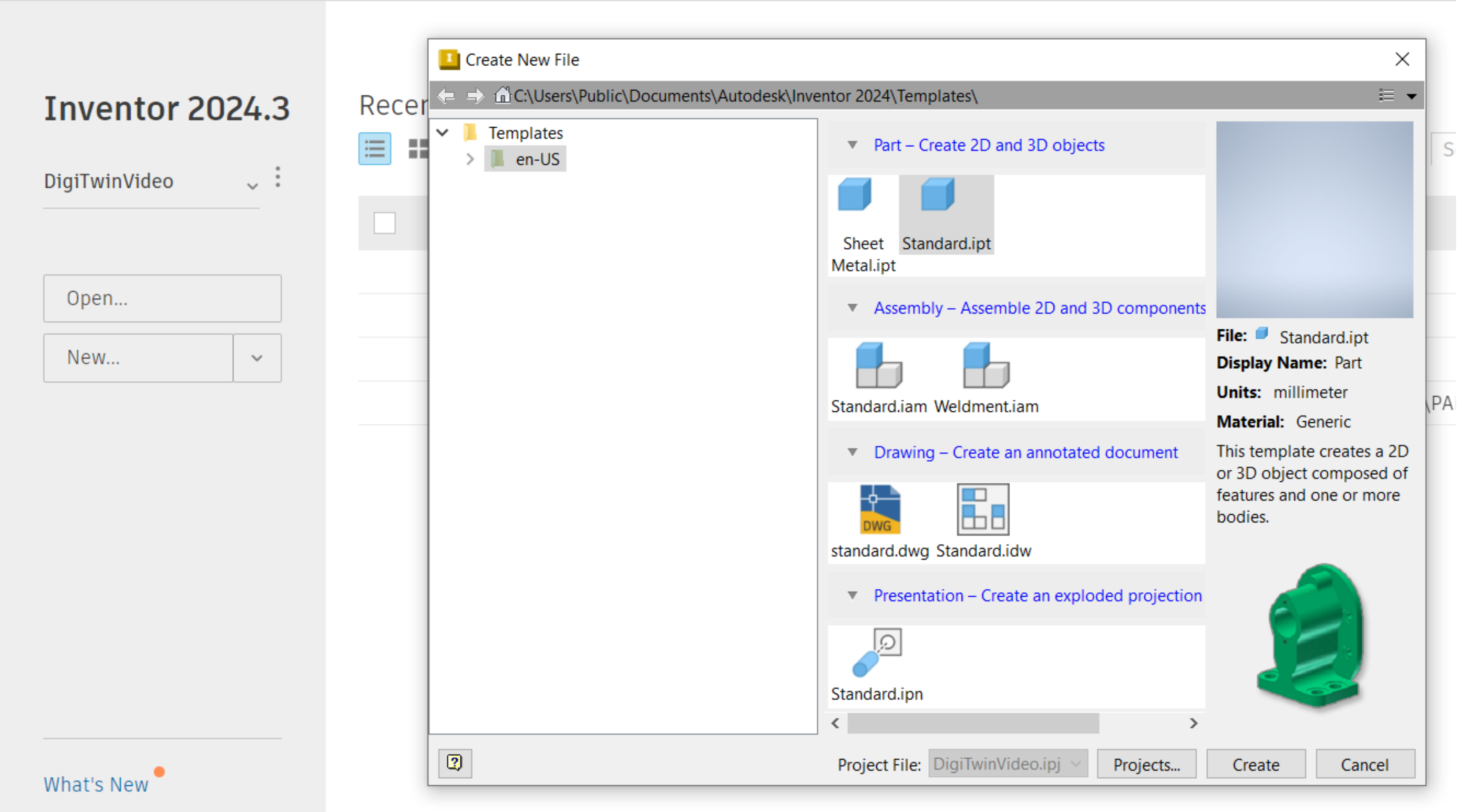
If we press *"New"*, a window will open with additional options like for example *"Sheet metal"*. In this project we will only create Standard parts.



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Welcome screen

- Project folders
  - Create new project
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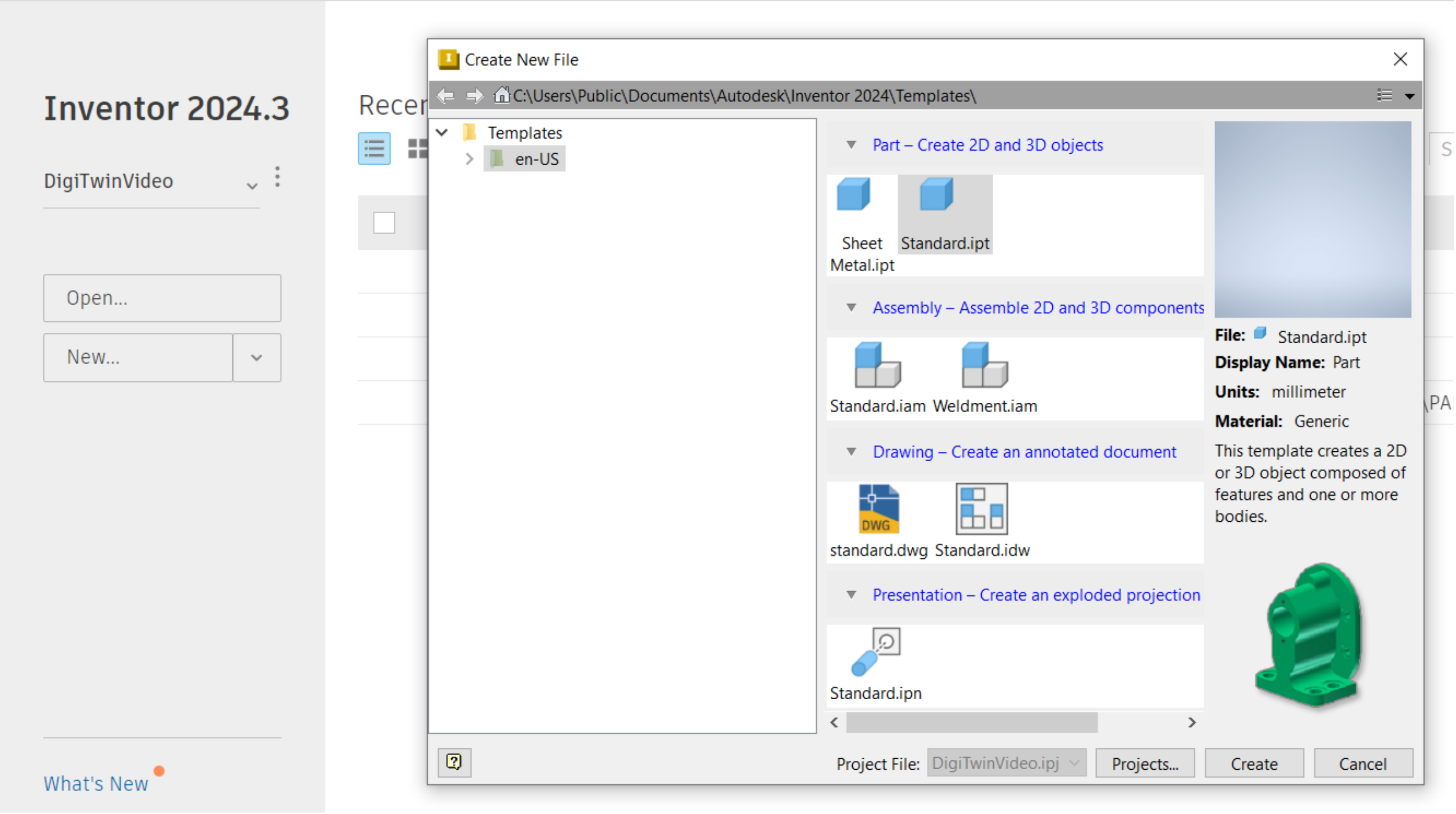
If we press *"New"*, a window will open with additional options like for example *"Sheet metal"*. In this project we will only create Standard parts. When a file is created, it will be opened in the Inventor Workspace.



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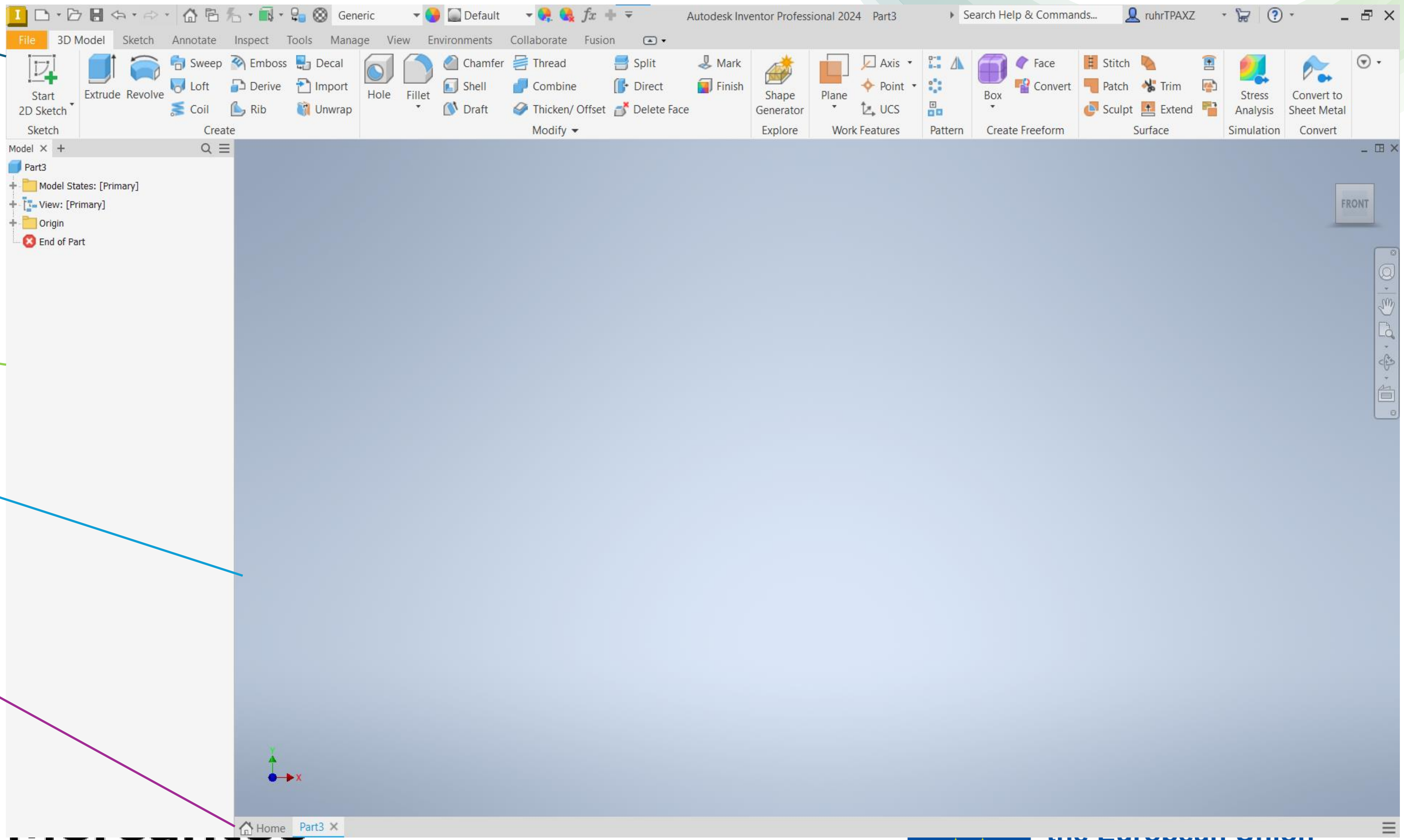
Inventor Workspace

- Menu - Ribbon
  - File
  - Assemble
  - 3D Model
  - Sketch
  - Tools for modelling
- Project tree
  - Folder structure of the open document
- Graphics window
  - Navigation
  - Visualization
- File Tabs
  - Open documents



Creating the first Part ”Machine base”

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- 2D sketch
  - Create sketch
  - Place in Plane
  - Draw a sketch
- 3D model
  - Create body(solid) from sketch
  - Modify/adjust body

When drawing a CAD model(Part) in inventor, the workflow is usually:

1. Create a Part(.ipt)
2. Chose or create a Plane
3. Place and draw a sketch in the plane
4. Create a 3D model from the sketch
5. Modify body
6. When multiple Parts exist, we can assemble them in an assembly(.iam)

If the CAD model needs to be produced in real life, we can create Drawings for production purposes(.DWG files).

If the model is to be introduced to customers or co-workers, it can be presented in a Presentation file(.ipn).

Other workflows exist, but this one is the most common one.

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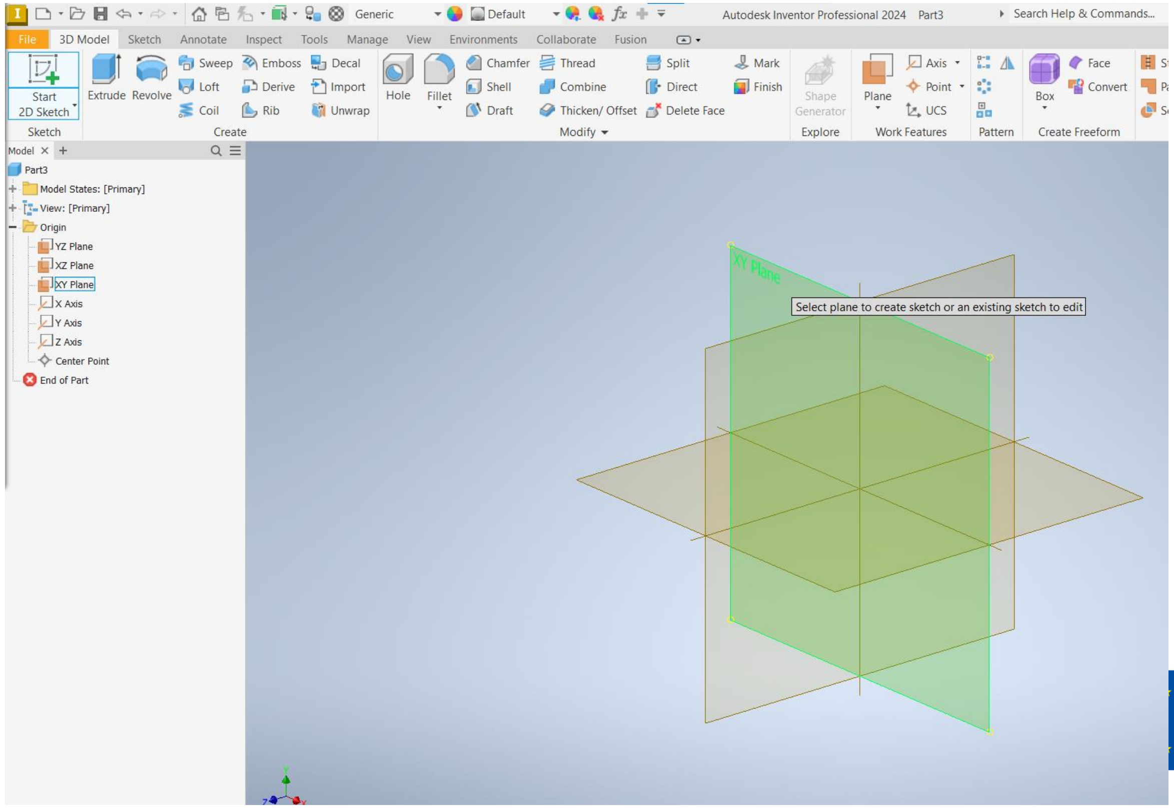
Creating the first Part ”Machine base”

- 2D sketch
  - Create sketch
  - Place in Plane
  - Draw a sketch
- 3D model
  - Create body(solid) from sketch
  - Modify/adjust body

To create a sketch, press the button *“Start 2D sketch”*.
Now the Origin planes will get visible and we have to chose one of them for the sketch.



Hover the mouse over the planes.

When the desired plane is green, left-click with the mouse

Creating the first Part ”Machine base”

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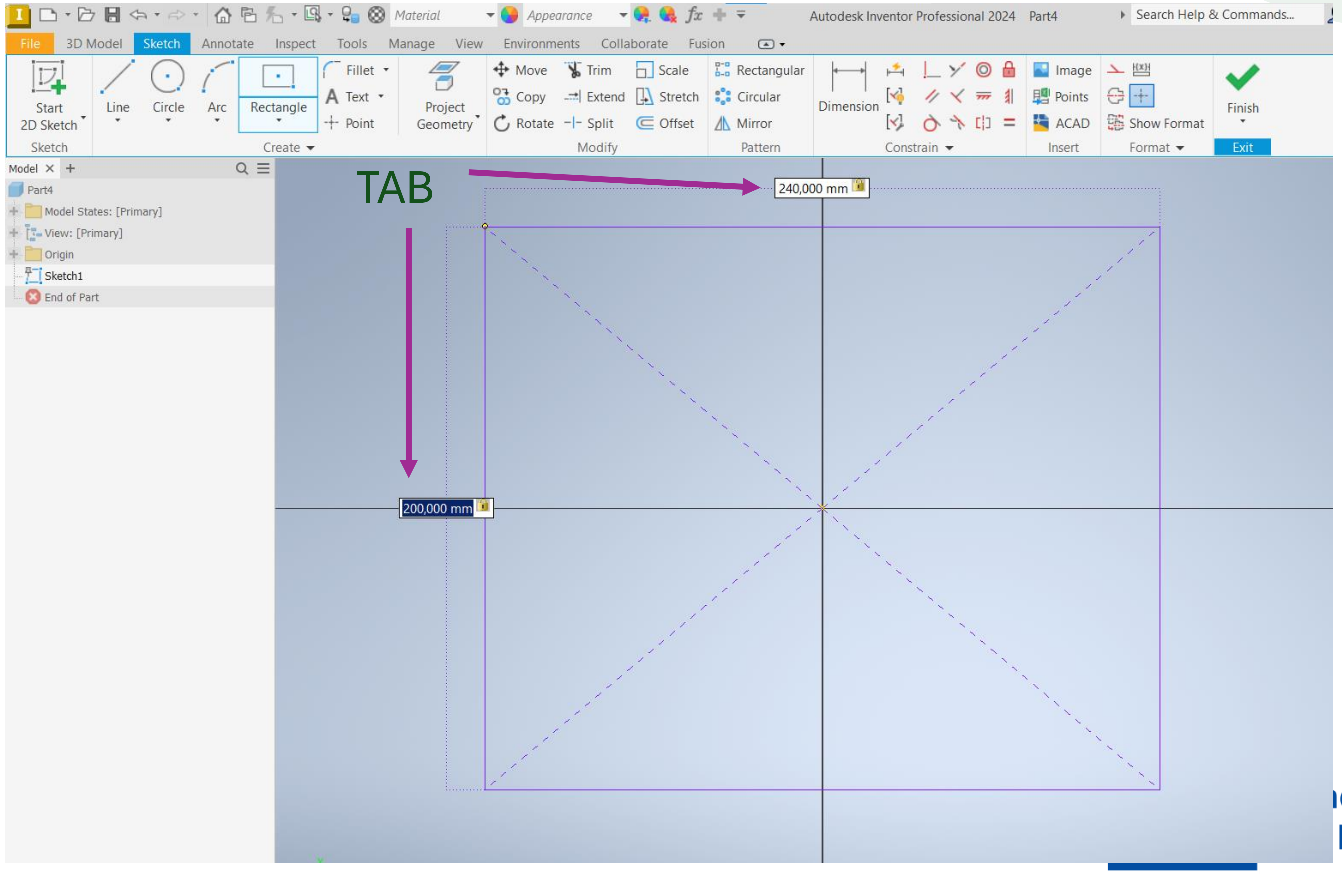
Now the sketch is connected to a plane and ready for drawing.
Draw the desired shape on the sketch and add dimensions to the sides.

- 3D model
  - Create body(solid) from sketch
  - Modify/adjust body



Here the two-point rectangle is chosen and placed at the center of the sketch.

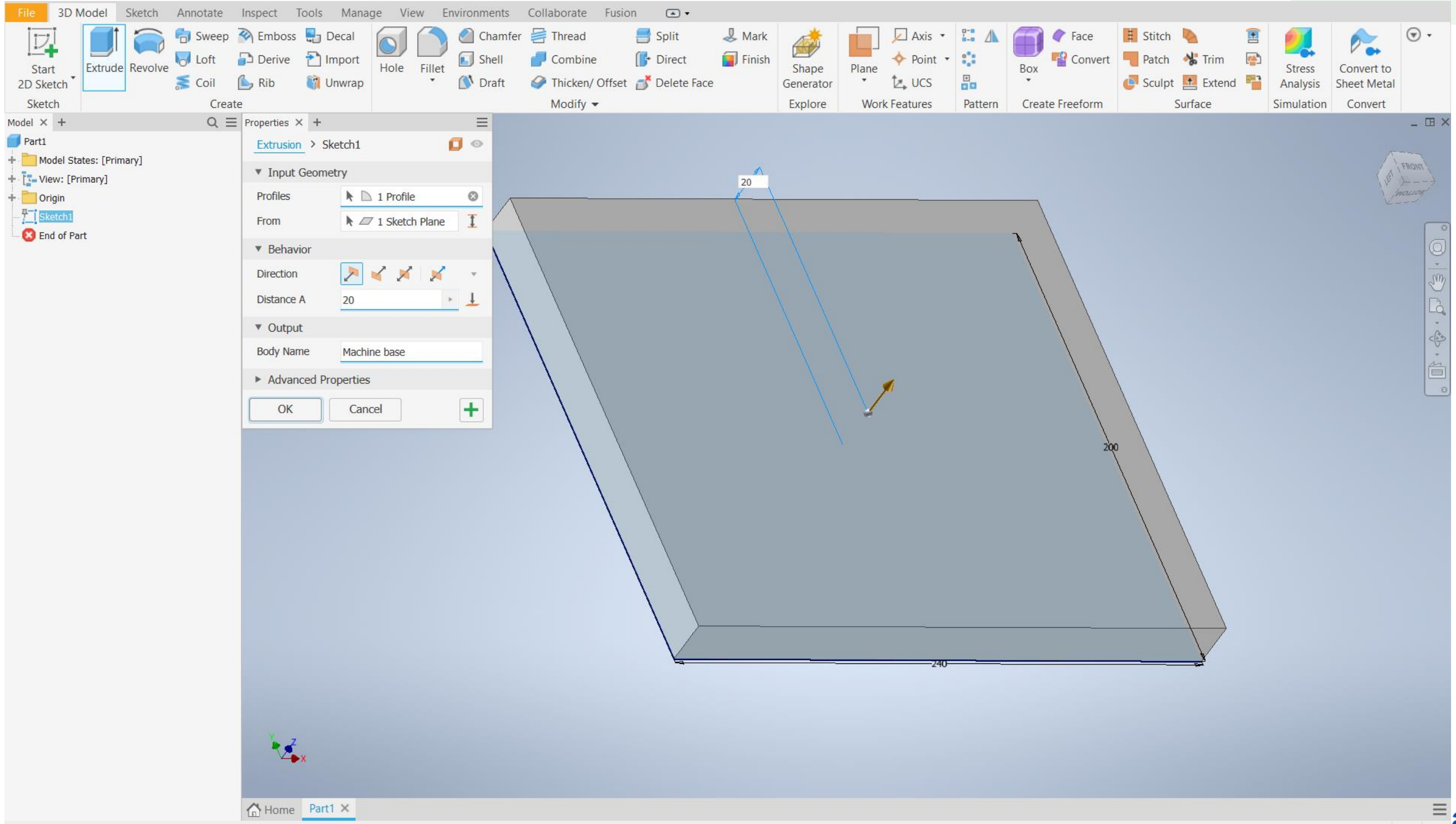
Write the length of the first side.
Press “Tab” to jump to the other side.
Finish the rectangle with “Enter”

When sketch is ready, press “Finish”

Creating the first Part ”Machine base”

- 2D sketch
    - Create sketch
    - Place in Plane
    - Draw a sketch
- 3D model
    - Create body(solid) from sketch
    - Modify/adjust body

Now it is time to add the 3rd dimension. In 3D Model ribbon, there is a group of tools named *“Create”* which are all for creating solids.



For this part, we just need a simple box or “plate”. It can be created with *“Extrude”.*

Press Extrude and click the rectangular profile on the sketch.

An extrusion window will open. Here we can choose the direction, the distance and a name for the solid.

Finish with *“OK”*

Assemblies (Video 2)

Create an assembly file

Add parts to an assembly

Create and modify parts inside an open assembly

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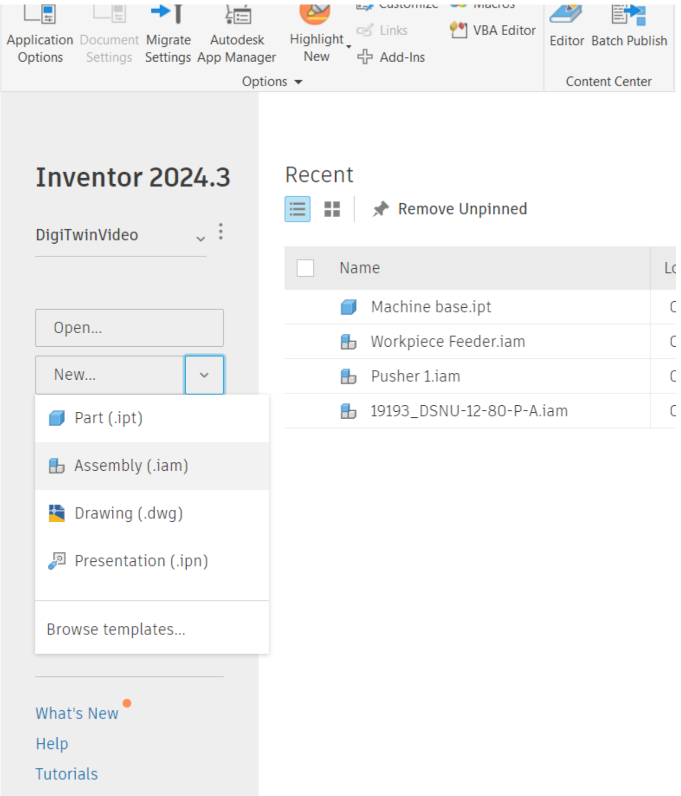
Assemblies

Assemblies can be created from the Welcome screen or from the workspace.

- Create Assembly file
  - Name the assebly
- Place part in assembly
- Create new parts in an open assembly
  - Create a workpiece on the machine base
  - Navigating between parts in assembly



To create an Assembly file, open the drop-down menu in the welcome screen and choose Assembly(.iam)

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Assemblies

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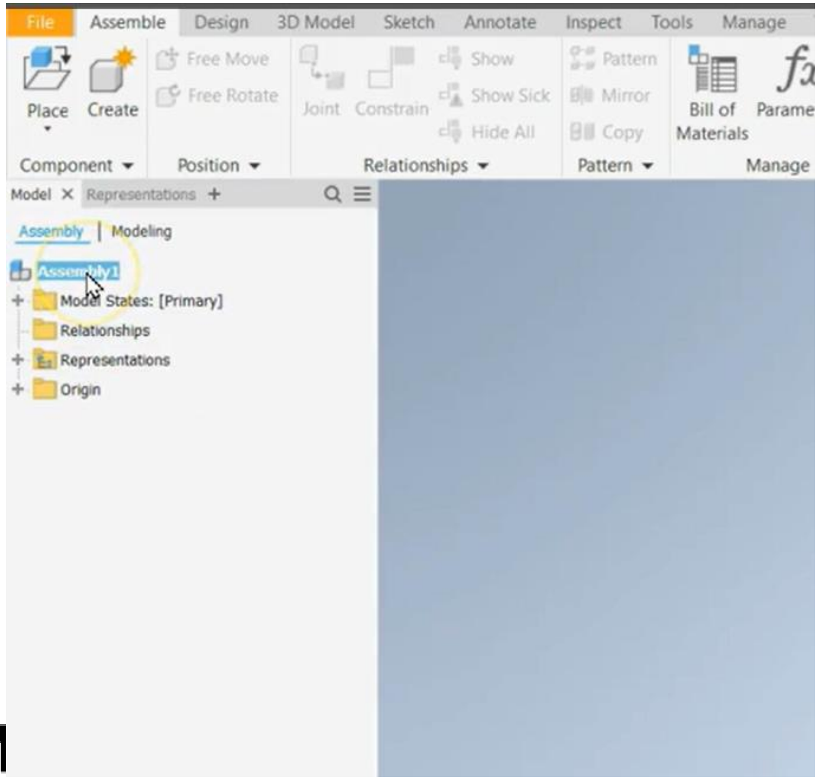

To rename the new assembly, double click the assembly root in the project tree.

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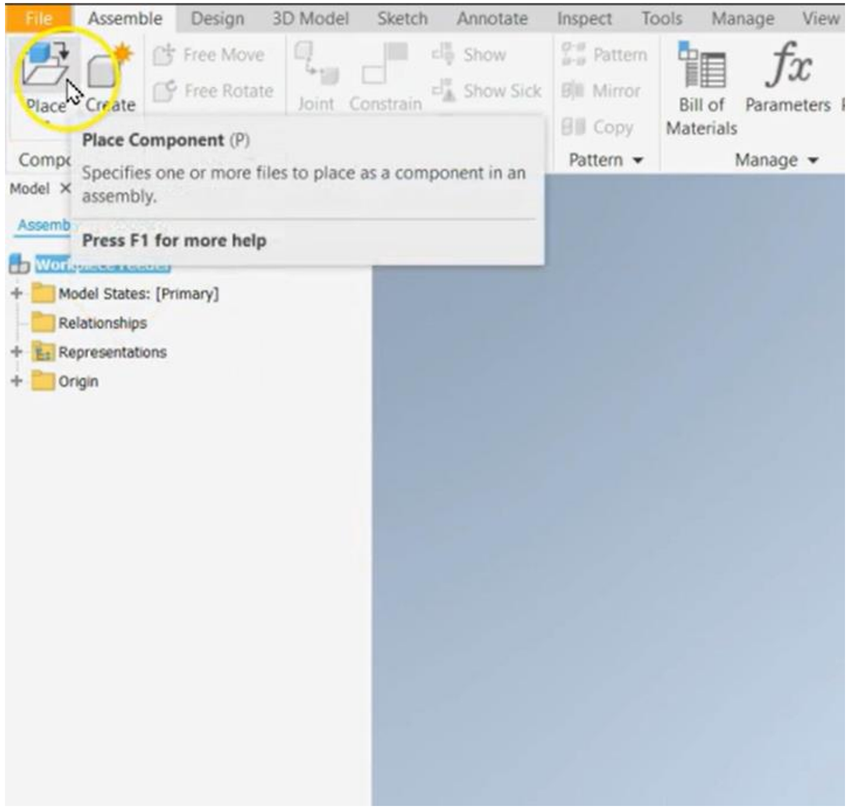
Assemblies

DIGITAL TWIN FOR EDUCATION

- Create Assembly file
  - Name the assebly
- Place part in assembly
- Create new parts in an open assembly
  - Create a workpiece on the machine base
  - Navigating between parts in assembly

When an assembly is open, it is possible to place existing parts in the assembly. It is also possible to create and modify parts inside the open assembly.



To place an existing part in the assembly, click *"Place"*

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Assemblies

DIGITAL TWIN FOR EDUCATION

- Create Assembly file
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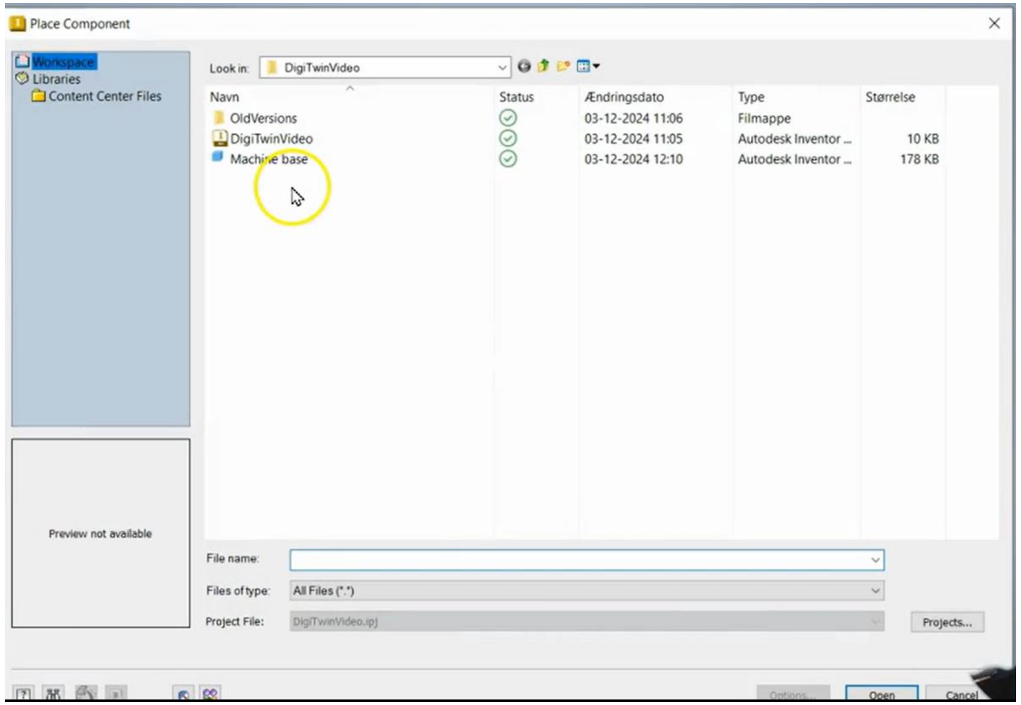

Choose the desired part to add it to the assembly

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Assemblies

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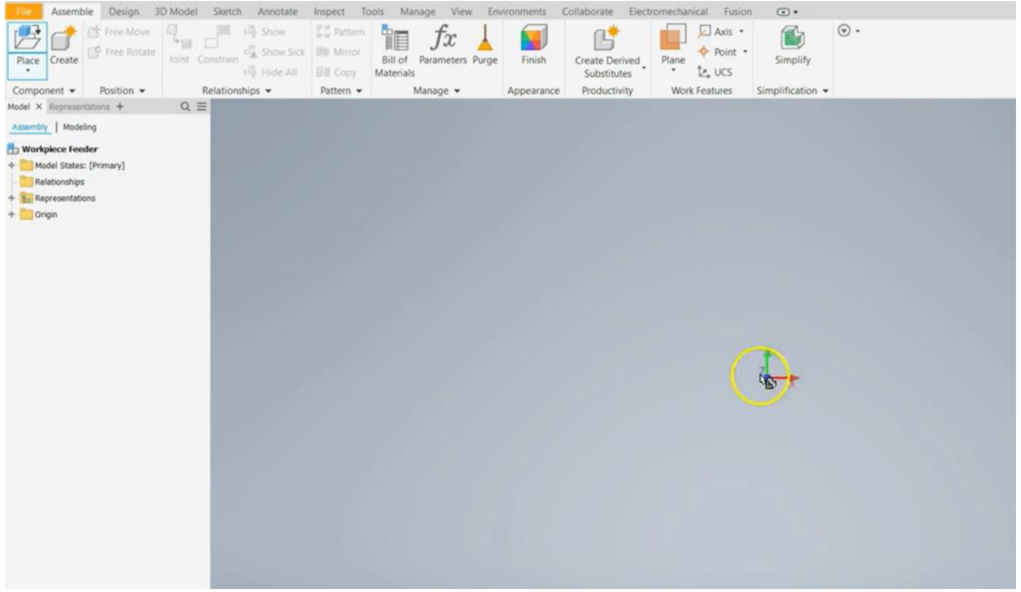

After choosing a part, the part must be placed in the assembly by left-clicking the mouse in the graphics area

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Assemblies

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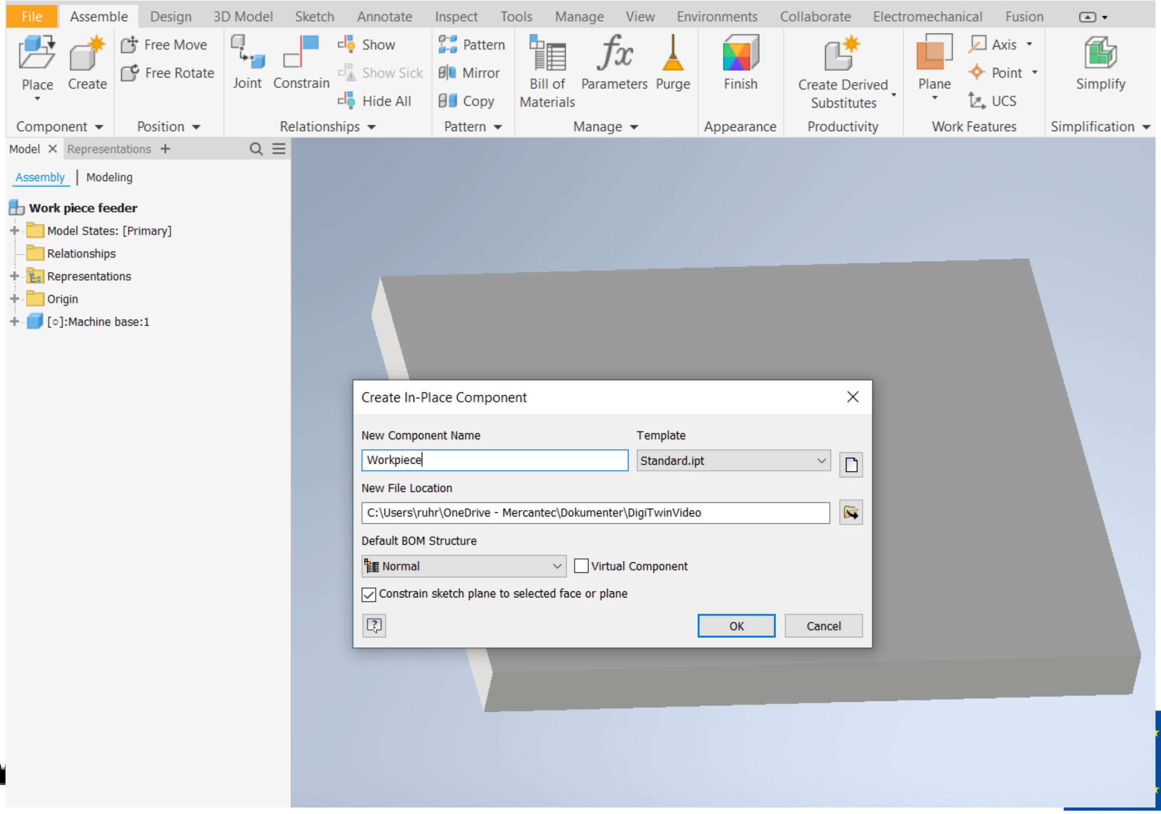

To create a new part inside the open assembly, click *"Create"* in the Components group in the Assembly Ribbon.

Assemblies

DIGITAL TWIN FOR EDUCATION

- Create Assembly file
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- Create new parts in an open assembly
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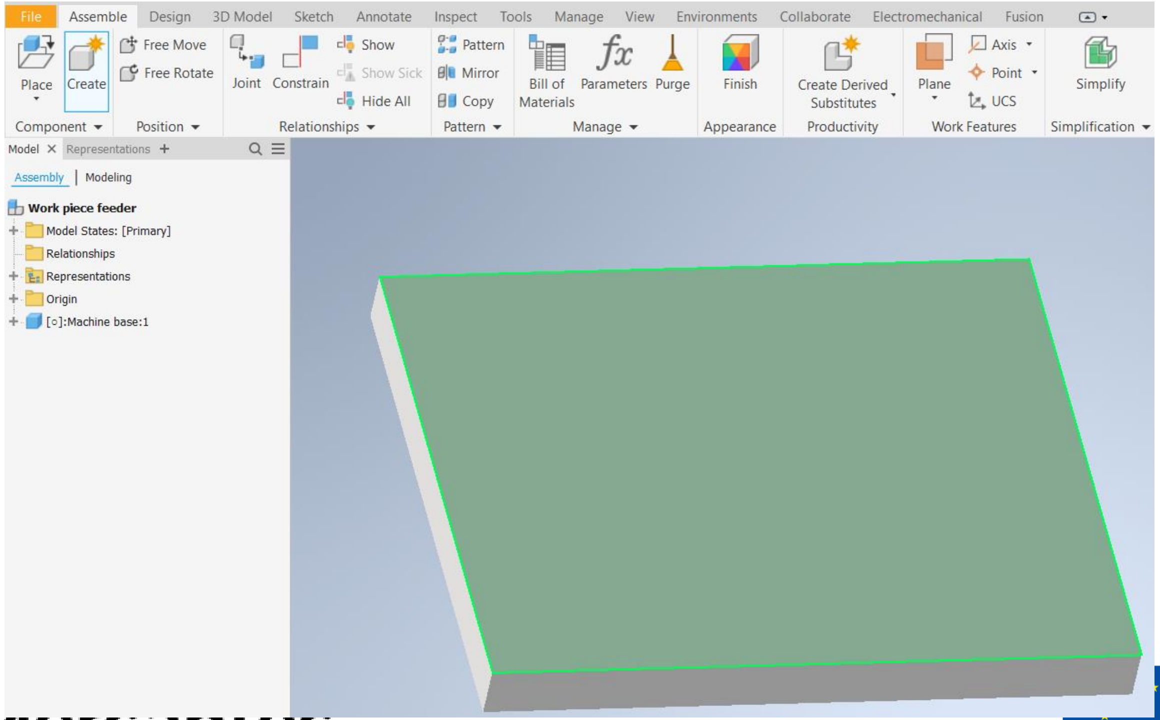

Now the mouse is holding the new part.

If the mouse is hovered over an existing part, a face will be highlighted in green.
If you place the part on the highlighted face, the origin of the new part will be constrained to the face.

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Assemblies

- Create Assembly file
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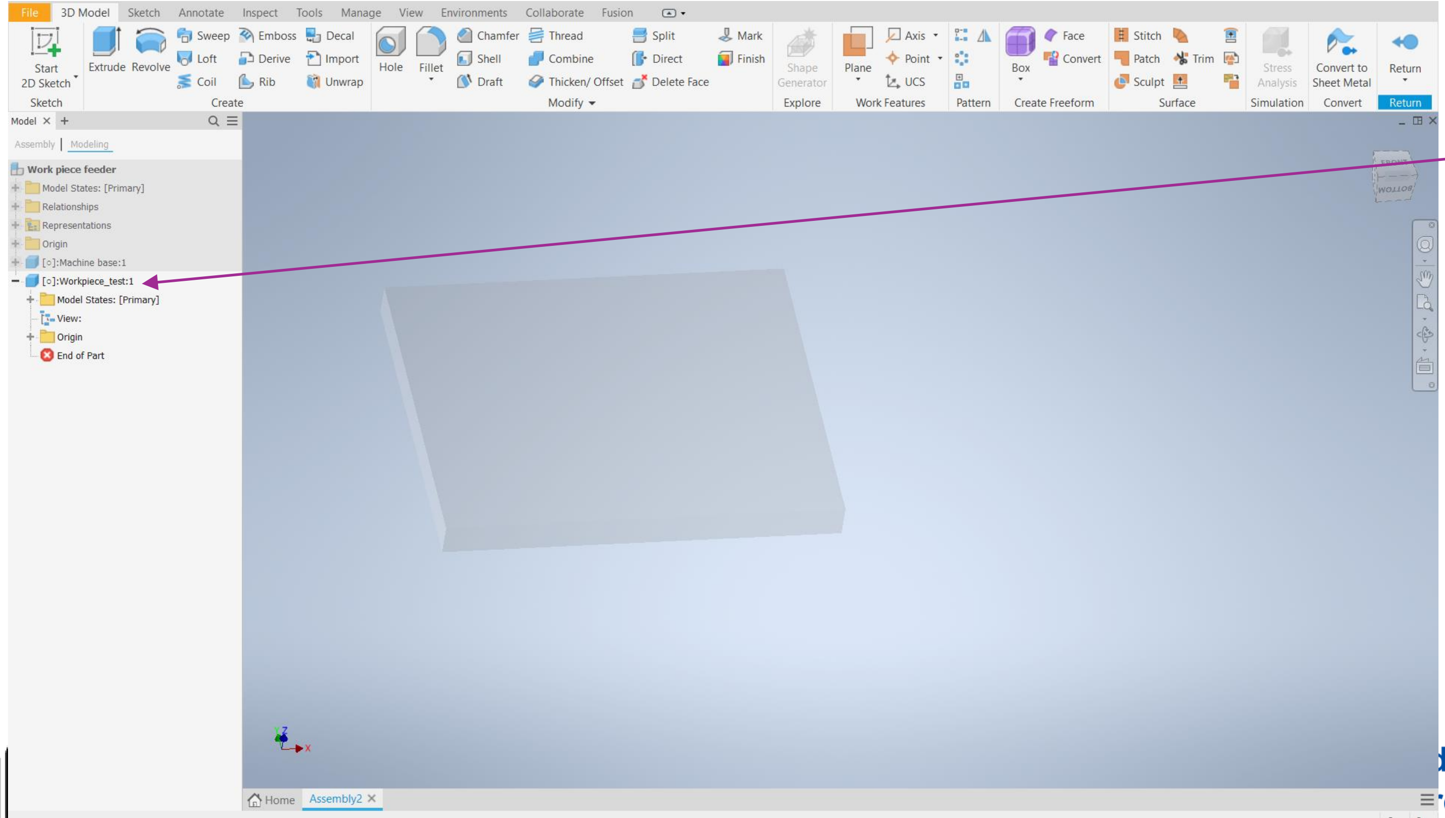
Now the new part is created inside the assembly.



Notice the new branch on the project tree.

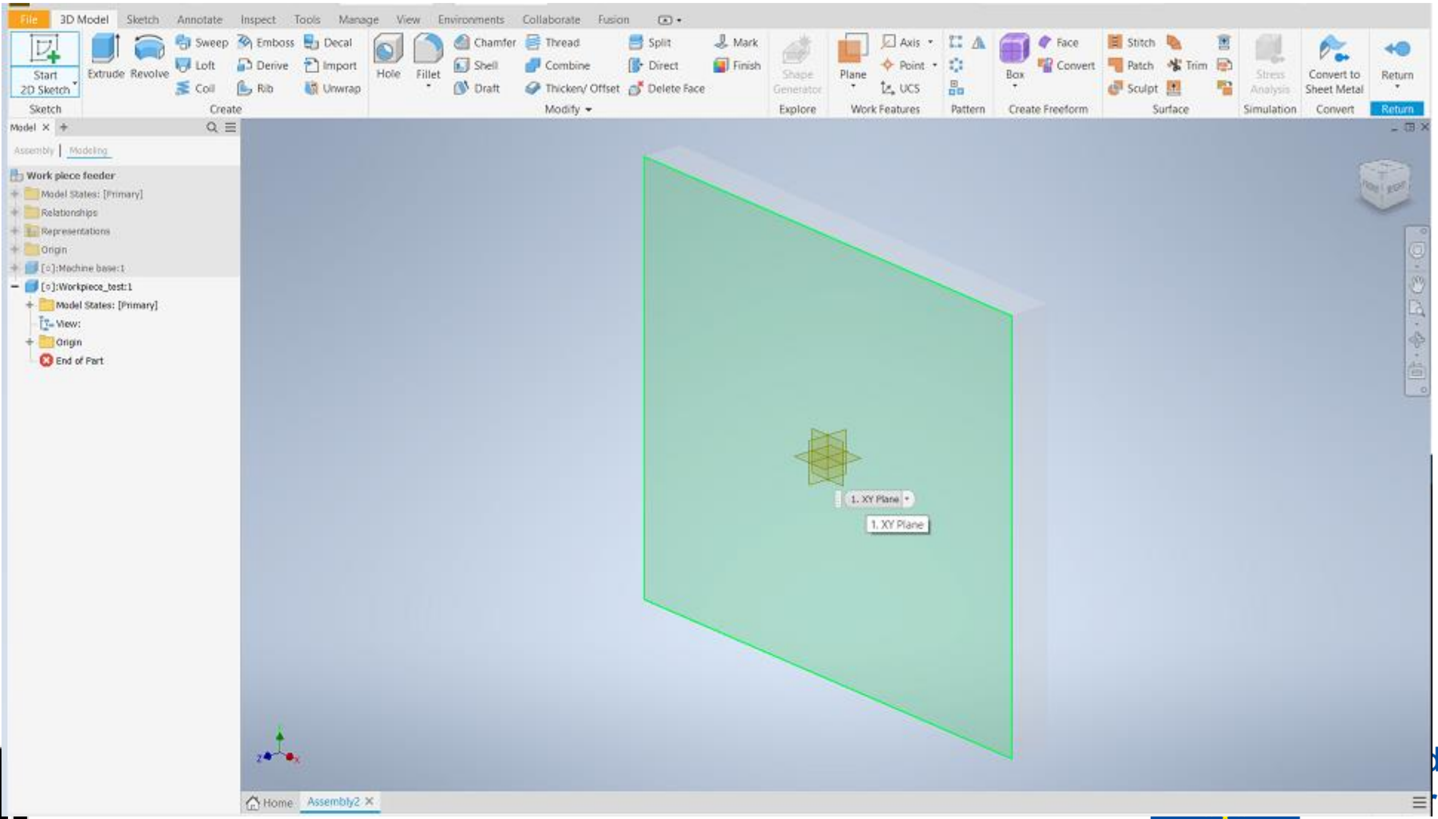
And notice that the new background of the new part is white, while the rest of the project tree is grayed out. This means that the new part *"Workpiece"* is open and the rest of the assembly is deactivated.

Assemblies

DIGITAL TWIN FOR EDUCATION

Now the new part is created inside the assembly.

- Create Assembly file
  - Name the assembly
- Place part in assembly
- Create new parts in an open assembly
  - Create a workpiece on the machine base
  - Navigating between parts in assembly



Now you can start designing the part, by placing a 2D sketch on a plane or on a surface

To close the new part and reactivate the entire assembly, press *“Return”*

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Subassemblies
&
Relationships (Video 3)

Creating a Pusher subassembly

Connecting parts with relationships

Placing the subassembly in the main assembly

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Subassemblies & relationships

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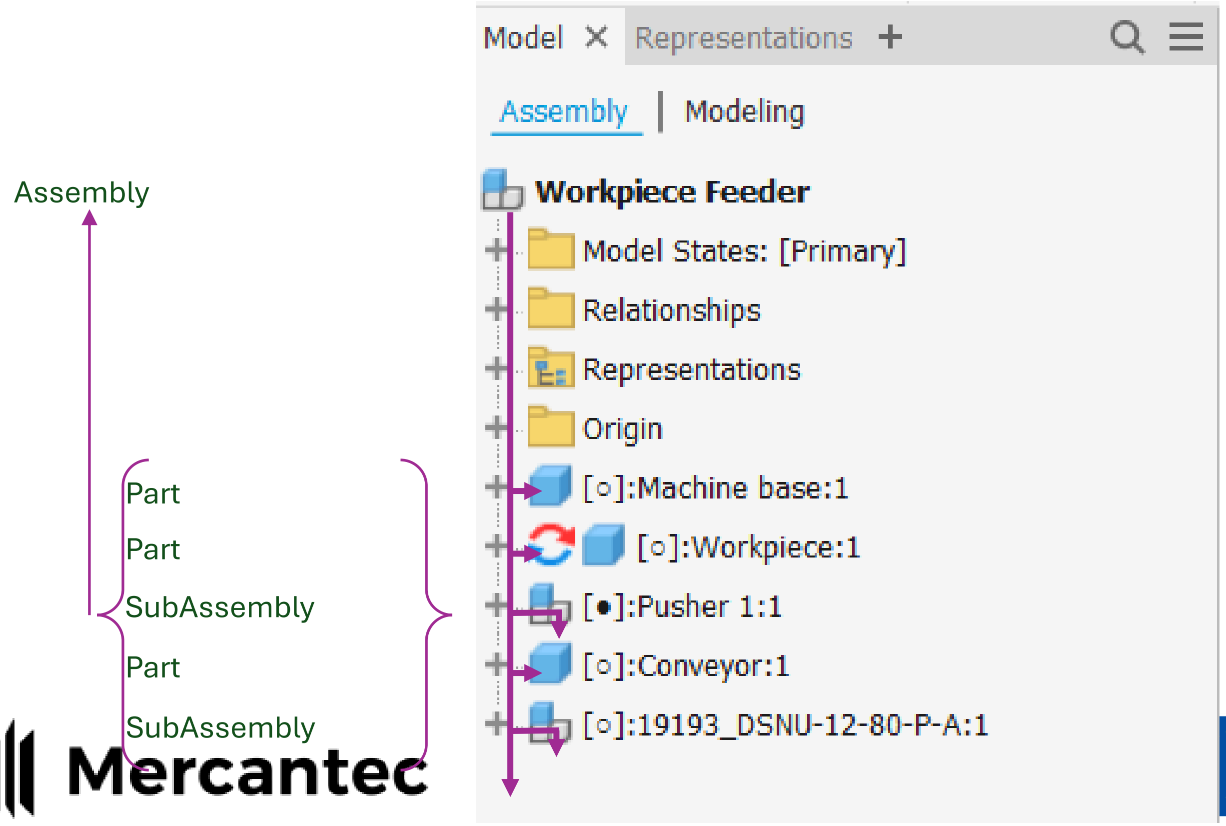
- Create a subassembly
  - Assembly inside assembly
- Relationships
  - Joints
  - Constrains
- Project Geometry

A sub assembly is just a normal assembly who lives inside a parent assembly. Subassemblies can also live inside subassemblies.



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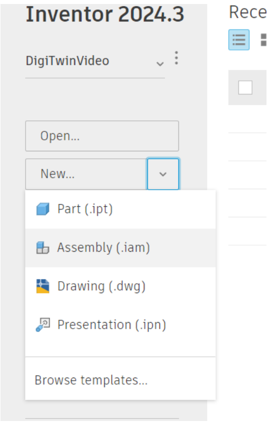
Subassemblies & relationships

- Create a subassembly
  - Assembly inside assembly
- Relationships
  - Joints
  - Constrains
- Project Geometry

To create a structure of Assembly/Subassemblies, simply create an Assembly file and Place it inside another Assembly.



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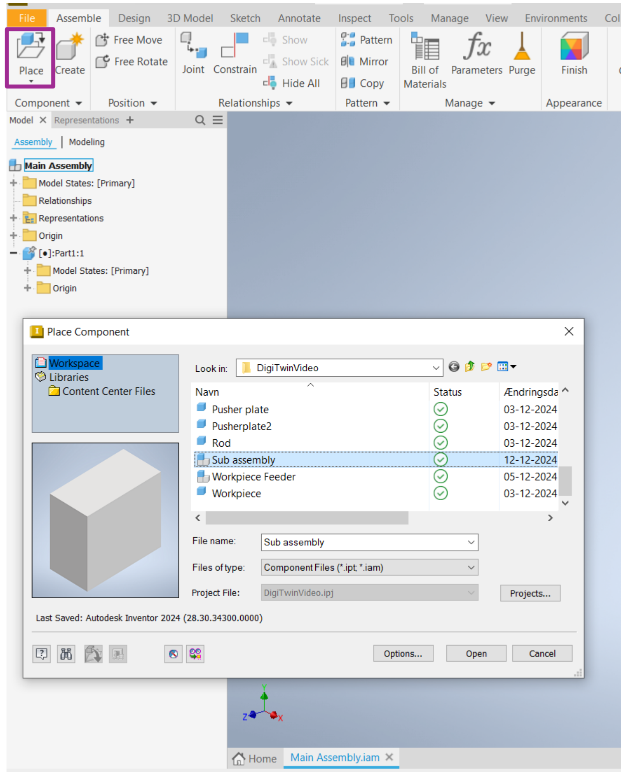

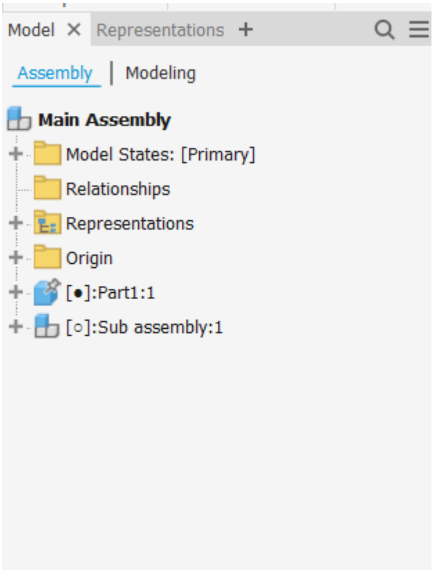

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Subassemblies & relationships

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- Create a subassembly
  - Assembly inside assembly
- Relationships
  - Constrains
  - Joints
- Project Geometry

Relationships are used to connect parts and assemblies to each other with a defined set of rules.
Both groups are used in video 3 as an example.

Constrains

Constrains are very specific limitations made for a geometry in a sketch or a component. Examples could be:

- Line1=12mm.
- Angle between Line1. and Line2 = 60°.
- Face1 and face2 = Tangent.
- Face1 and face2 = Mated.

Joint

Joint relationships are preconfigured tools consisting of several Constrains:

- Rigid is a relationship without movement allowed.
- Rotational has a defined rotatin allowed
- Slider allows linear movement
- Planar allows 2D movement in a specified plane

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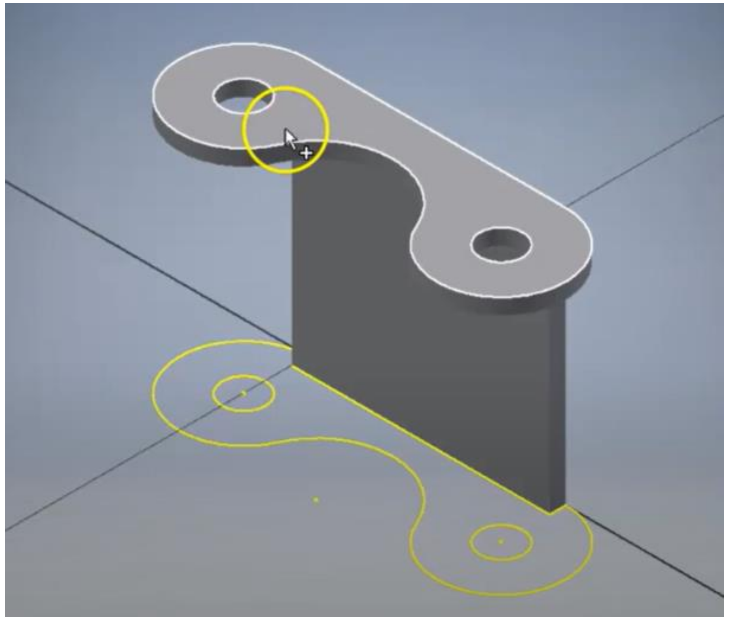
Subassemblies & relationships

- Create a subassembly
  - Assembly inside assembly
- Relationships
  - Constrains
  - Joints
- Project Geometry

Project is a handy tool that “copies” a geometry to a sketch, so you do not need to draw the geometry again.



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Modelling the conveyor (Video 4)

Sketch - Arc

Mate relationship with offset

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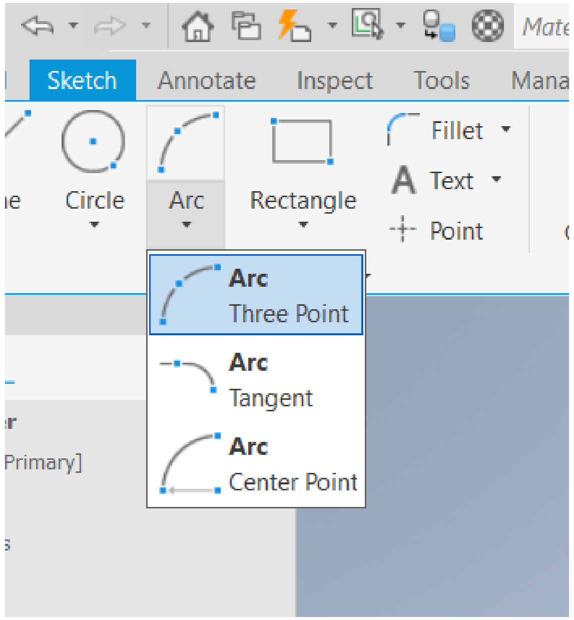
Modelling the conveyor

- Sketch Arcs
    - Tangent arc
- Relationship
    - MATE with offset

Arcs can be crated in three different methos: Three Point, Center and Tangent



Three Point Arc is drawing an arc from point 1, through point 2, to point 3

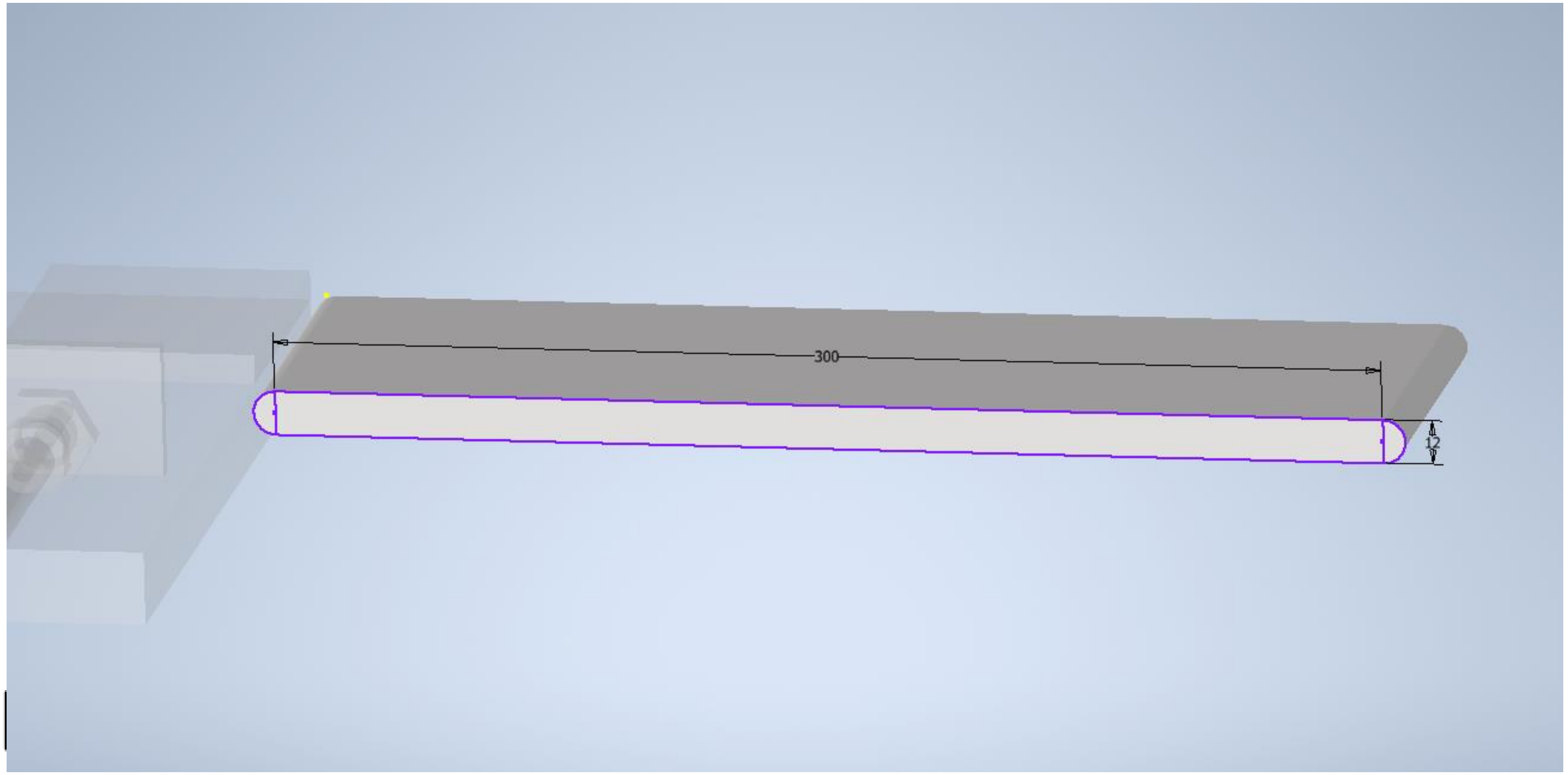
Tangent Arc must be connected with two points to existing geometry and will size the arc to be tangent with the two geometrys, like in video 4.

Center Point Arc is created by placing an Arc center and the two end points.

Modelling the conveyor

- Sketch Arcs
  - Tangent arc
- Relationship
  - MATE with offset

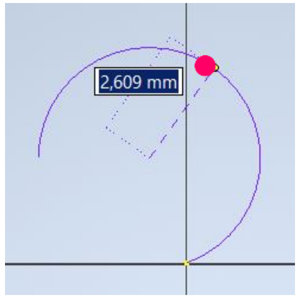
The conveyor is very simple. It consist of a Sketch with a box. On each end of the box, half a circle is connected.
3rd dimension is a simple Extrusion of the box and arcs.

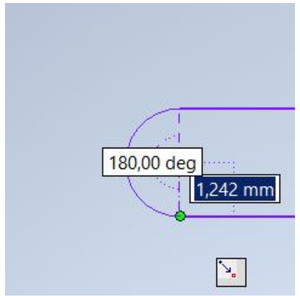


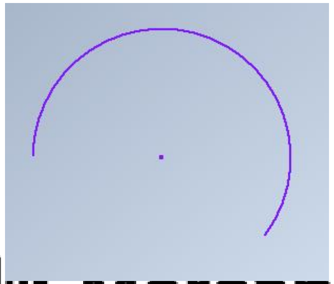
Modelling the conveyor

- Sketch Arcs
  - Tangent arc
- Relationship
  - MATE with offset

Arcs can be crated in three different methods: Three Point, Center and Tangent







Three Point Arc is drawing an arc from point 1, through point 2, to point 3

Tangent Arc must be connected with two points to existing geometry and will size the arc to be tangent with the two geometrys, like in video 4.

Center Point Arc is created by placing an Arc center and the two end points.

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Import CAD from vendor Pusher 2 (Video 5)

Import STEP file from Festo

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Import model from Vendor

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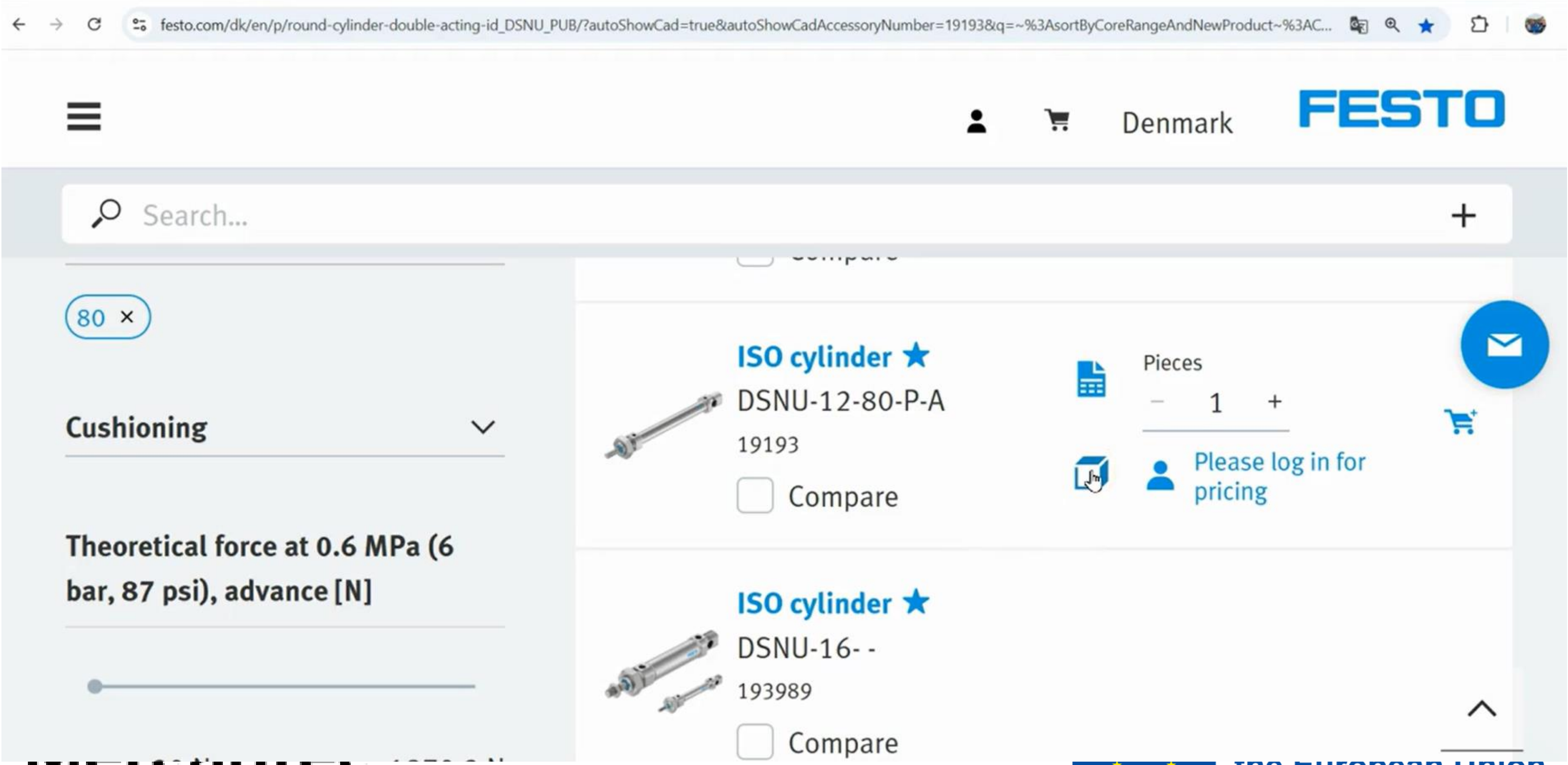
- Find CAD from Festo
  - STEP
  - File to Inventor project folder
  - Import to project
- Relationship
  - MATE with offset
  - Adjust position on the base plate

Inventor can import various CAD file types. The most commonly used file type across different CAD-software is STEP.

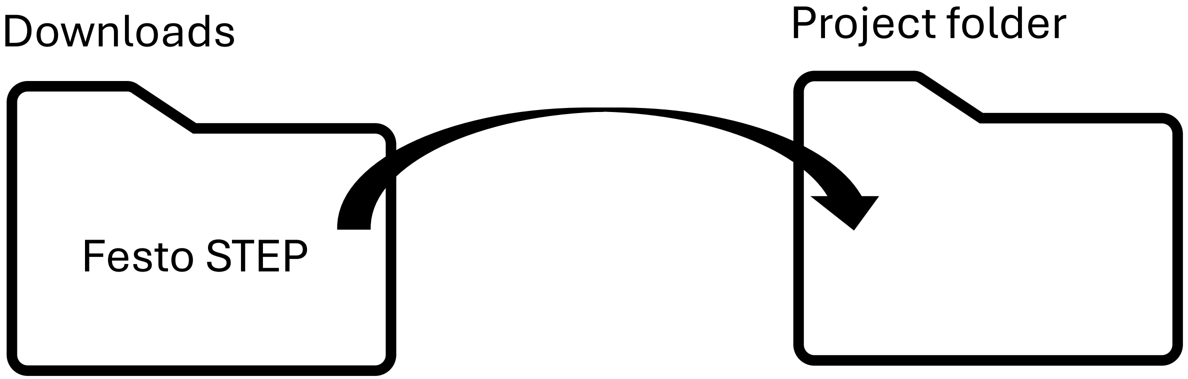


Import model from Vendor

DIGITAL TWIN FOR EDUCATION

- Find CAD from Festo
  - STEP
  - File to Inventor project folder
  - Import to project
- Relationship
  - MATE with offset
  - Adjust position on the base plate

It is good practice to always keep all implemented files inside the Inventor project folder.
Therefor we will move the downloaded file from Download folder to project folder before implementing it in the the assembly.



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Import model from Vendor

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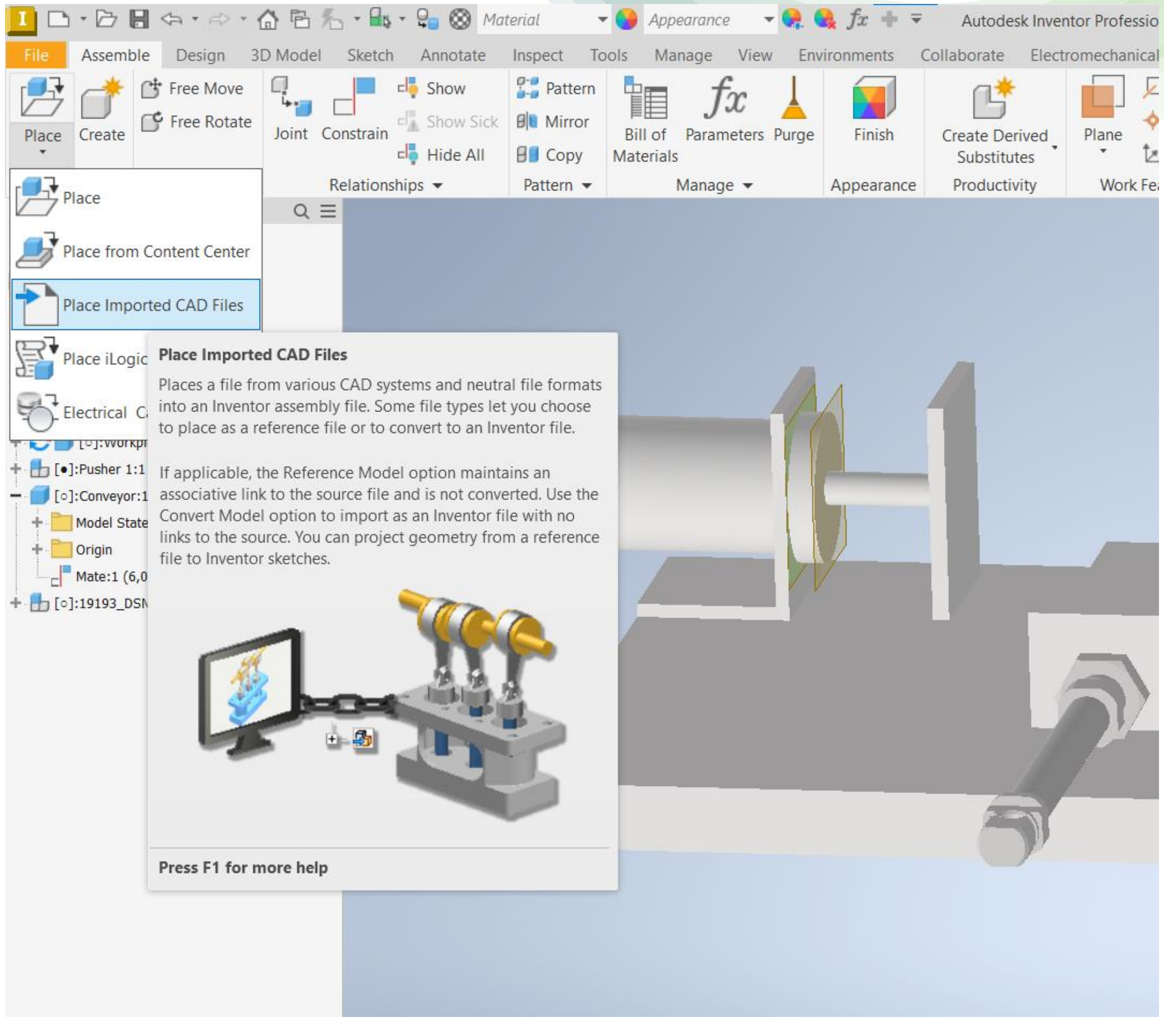
- Find CAD from Festo
  - STEP
  - File to Inventor project folder
  - Import to project
- Relationship
  - MATE with offset
  - Adjust position on the base plate

In the “Assemble” tab, make sure to have the Main Assembly active and presse the “Place Imported CAD Files” button.
Find the file in the browser to add it.



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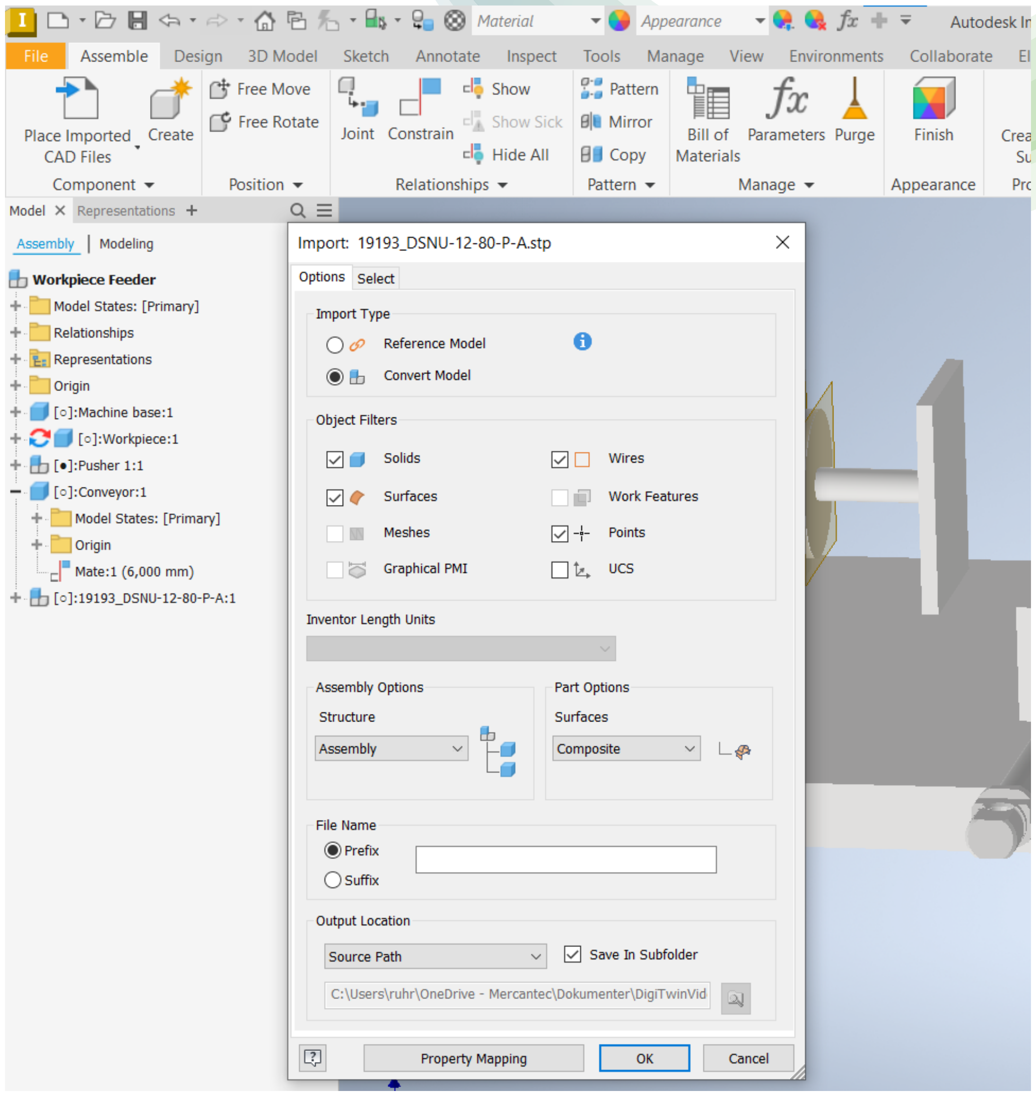
Import model from Vendor

- Find CAD from Festo
  - STEP
  - File to Inventor project folder
  - Import to project
- Relationship
  - MATE with offset
  - Adjust position on the base plate

Choose the desired import settings and press OK.

In this example, the default settings has been used for the import.



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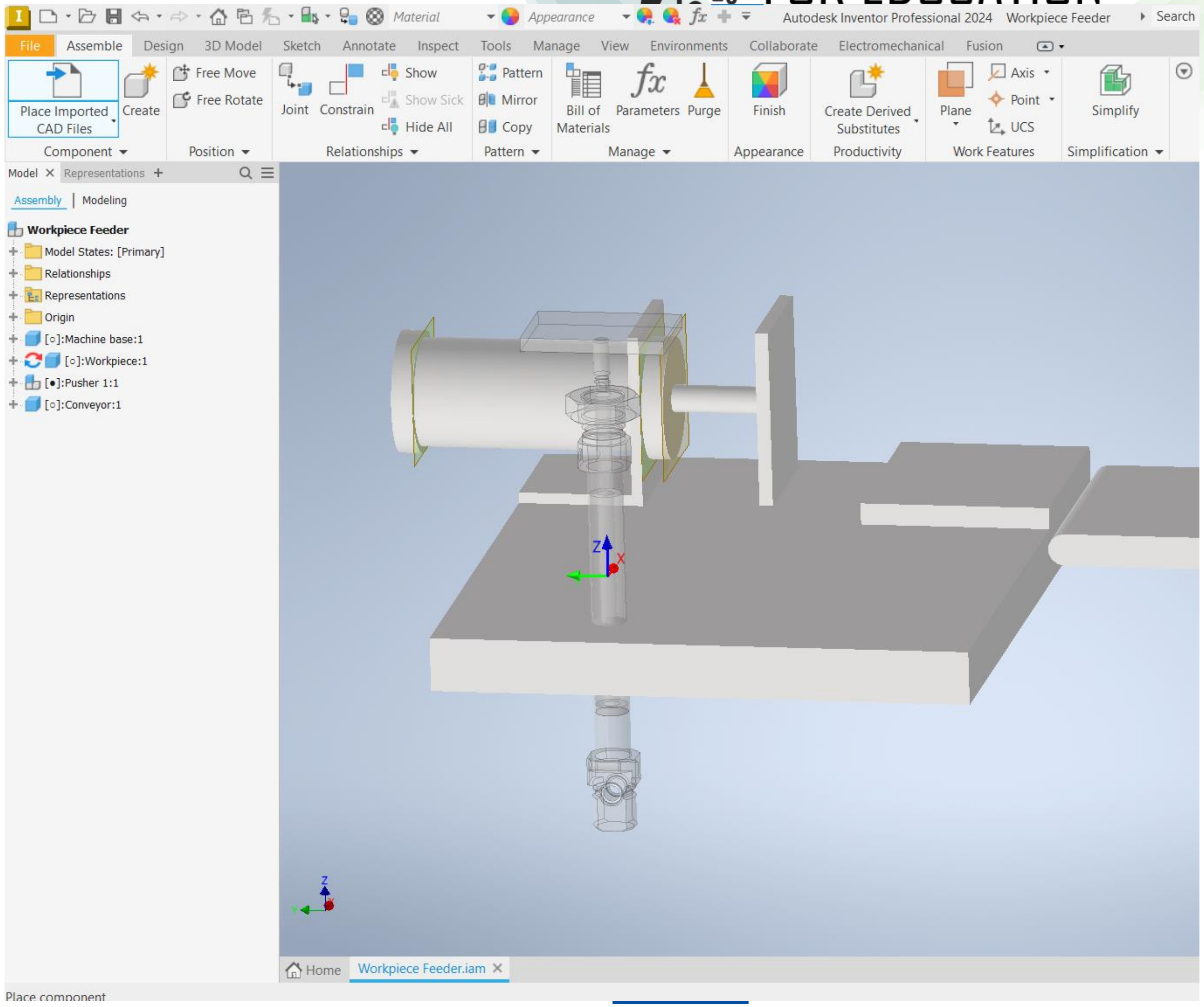
Import model from Vendor

- Find CAD from Festo
  - STEP
  - File to Inventor project folder
  - Import to project
- Relationship
  - MATE with offset
  - Adjust position on the base plate

Place the imported component in some random place in the graphics window.

Use a Relationship tools to fixate the component in the assembly with relevant rules and position.



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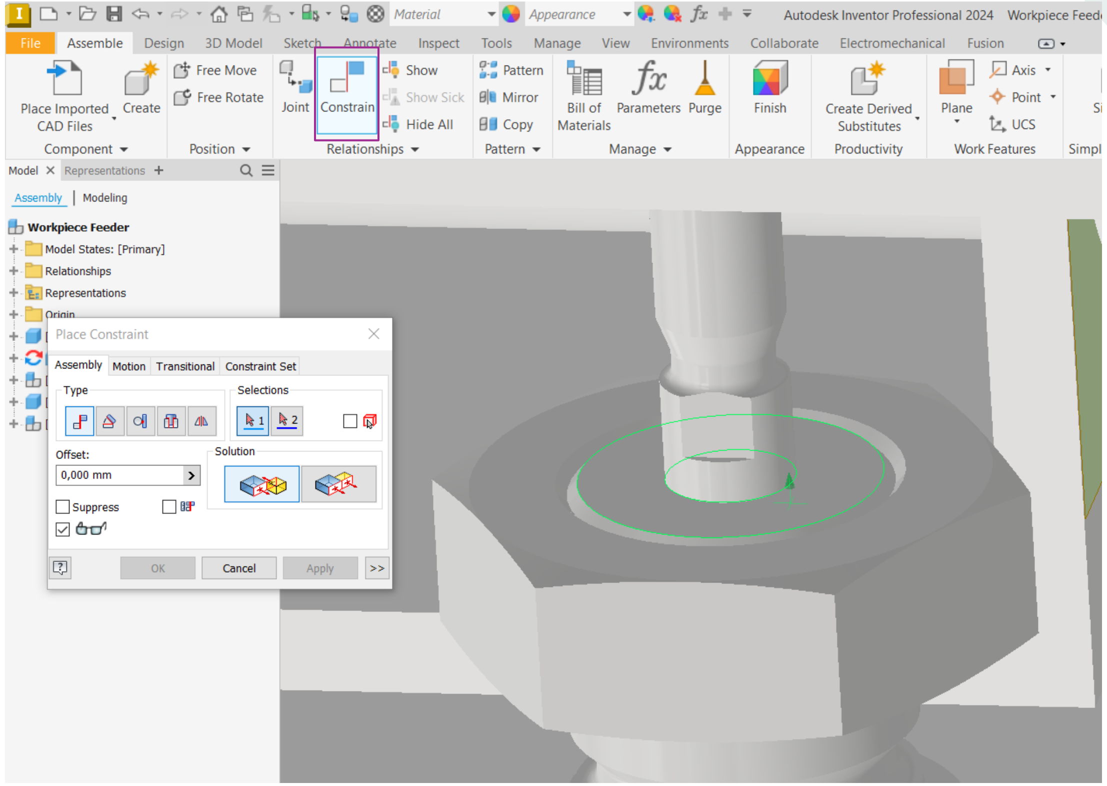
Import model from Vendor

- Find CAD from Festo
  - STEP
  - File to Inventor project folder
  - Import to project
- Relationship
  - MATE with offset
  - Adjust position on the base plate



Press the “Constrain” button

Choose the MATE-option in the Constrain window.

Choose the first face with a left-click.

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- Find CAD from Festo
  - STEP
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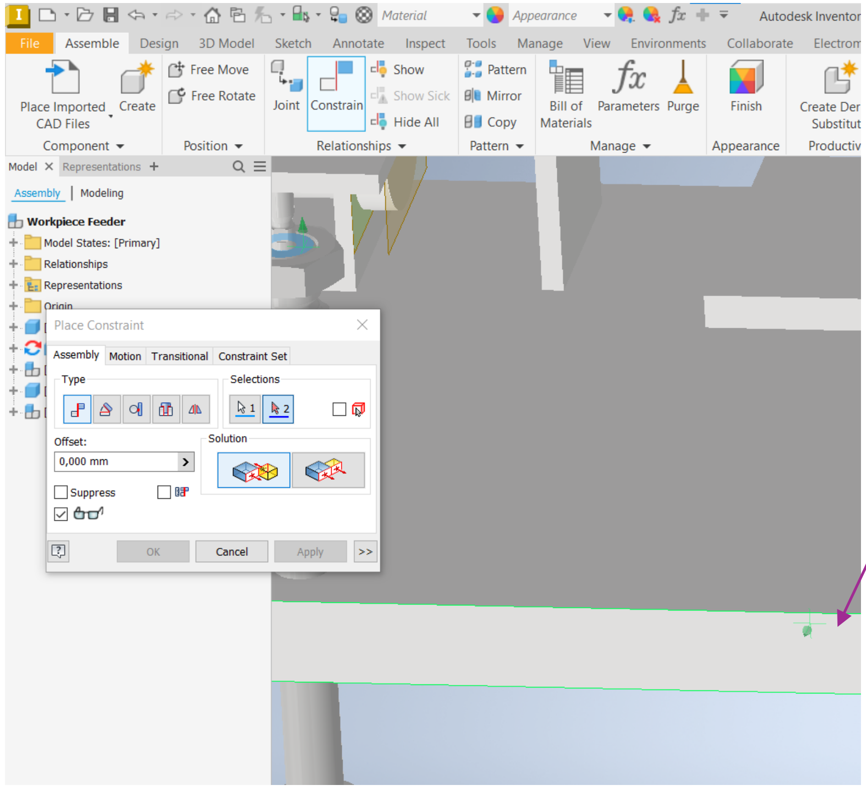
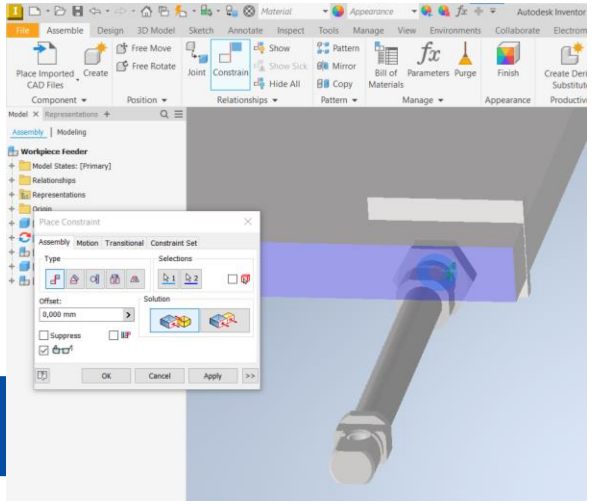
  - Import to project
- Relationship
  - MATE with offset
  - Adjust position on the base plate



Then click the face that the cylinder must MATE with



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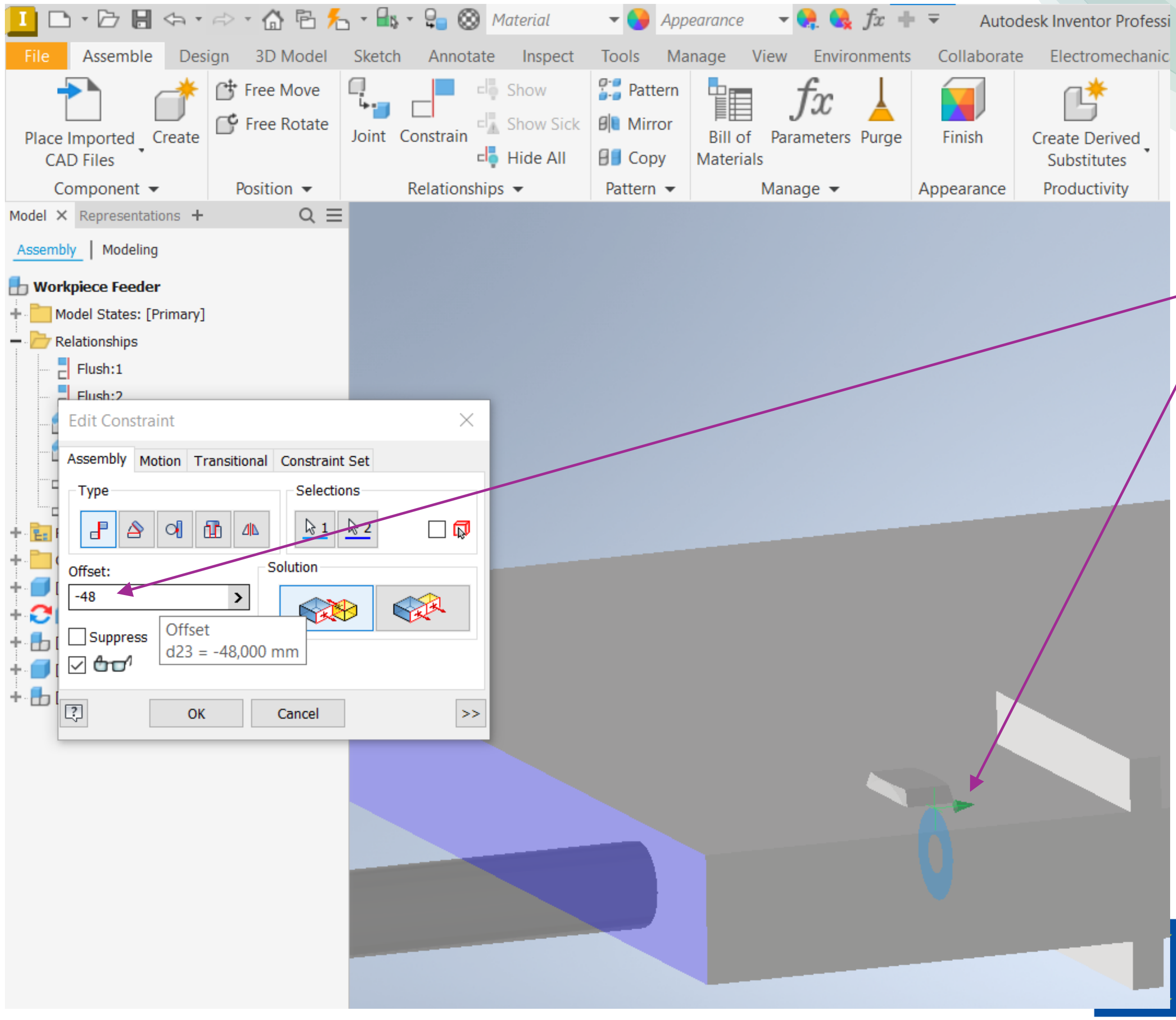
Import model from Vendor

- Find CAD from Festo
  - STEP
  - File to Inventor project folder
  - Import to project
- Relationship
  - MATE with offset
  - Adjust position on the base plate



Now offset the MATE by dragging the arrow between the two faces or write the desired offset into the Constrain window.

Finish with “OK”

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Import model from vendor

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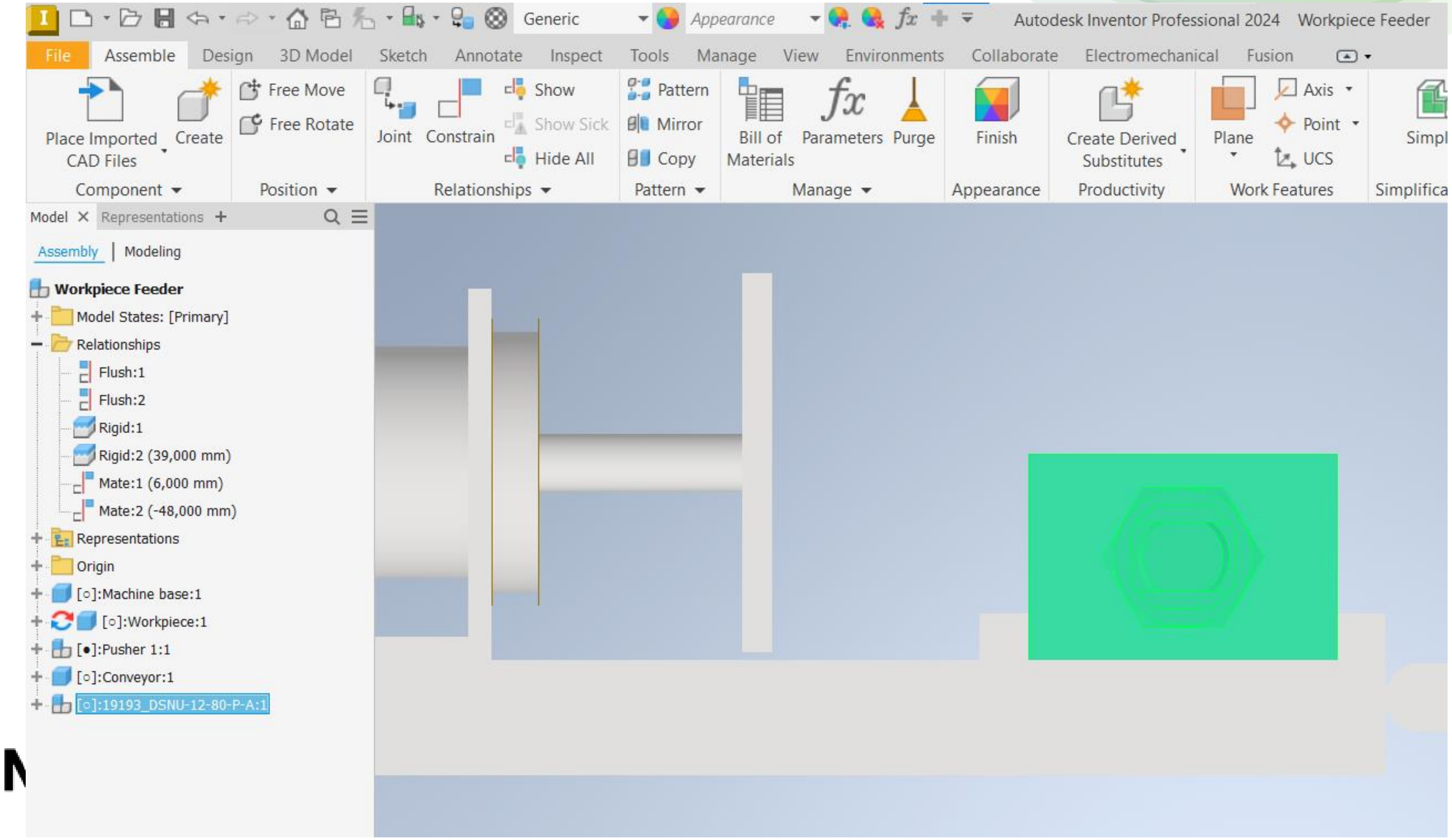
Lastly, drag the Festo cylinder to the optimal position with the mouse.
The component still have two dimensions of freedom to do this.

- Find CAD from Festo
  - STEP
  - File to Inventor project folder
  - Import to project

- Relationship
  - MATE with offset
  - Adjust position on the base plate



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Export project to STEP (Video 6)

Export the entire CAD drawing to STEP file

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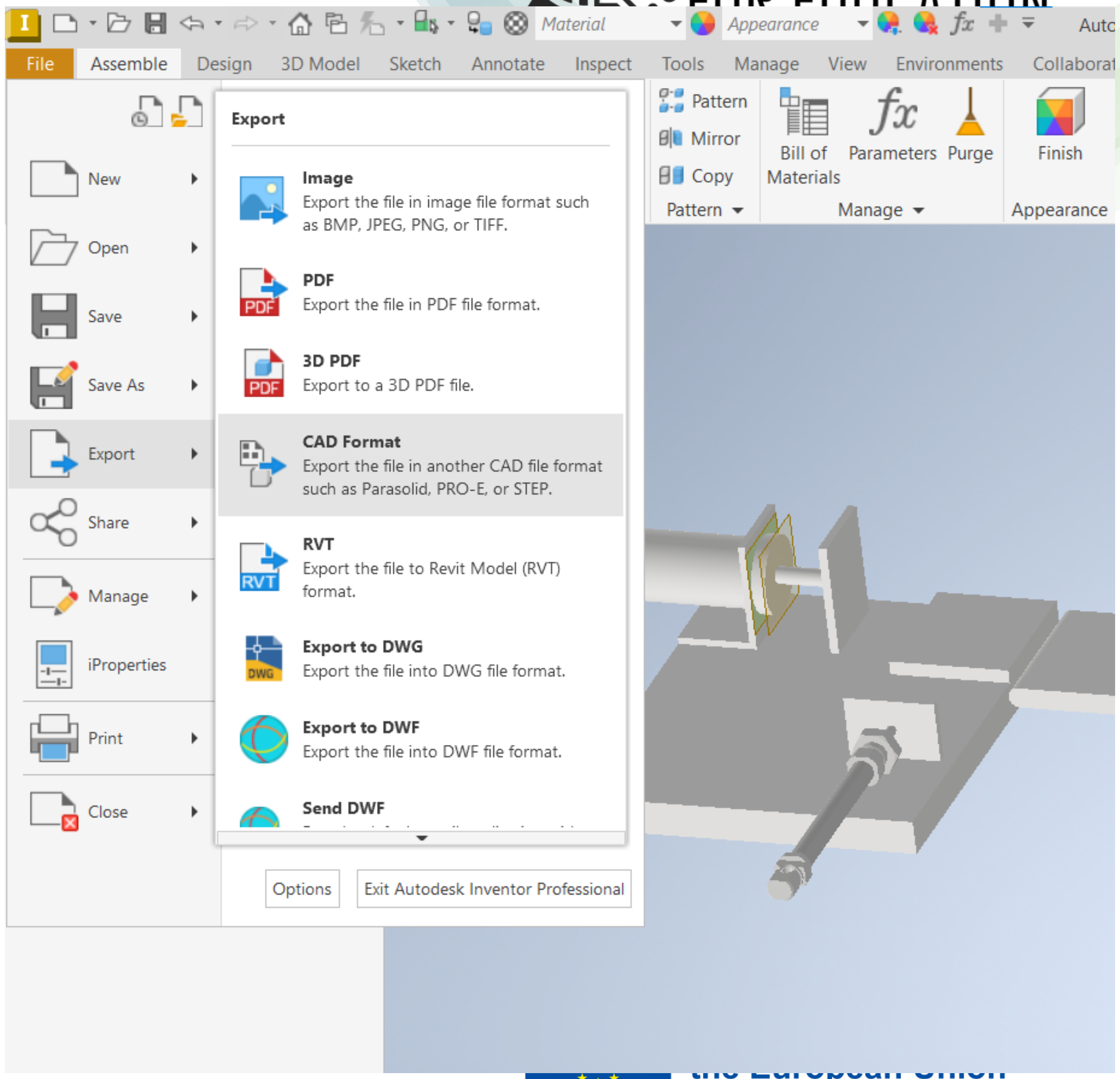
Export to STEP

- Activate Main assembly
- Export the project
  - Export to STEP

In the FILE tab, choose Export / CAD Format



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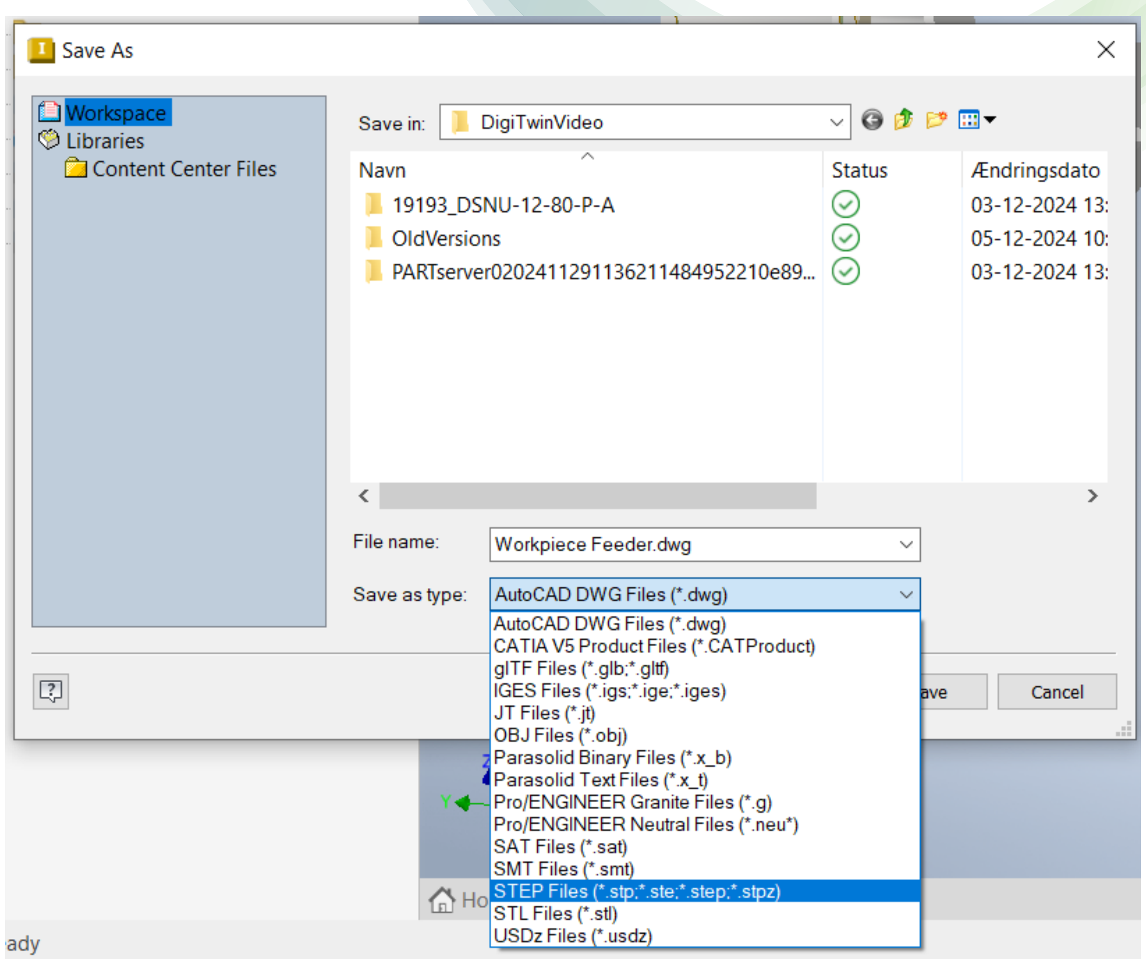
Export to STEP

- Activate Main assembly
- Export the project
    - Export to STEP

Name the Export file

Choose the location for the file

Choose the file type and press save



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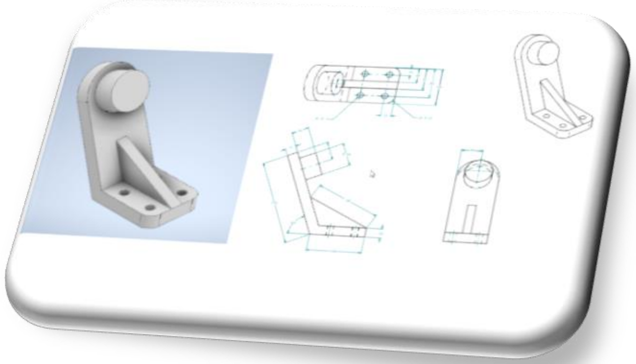
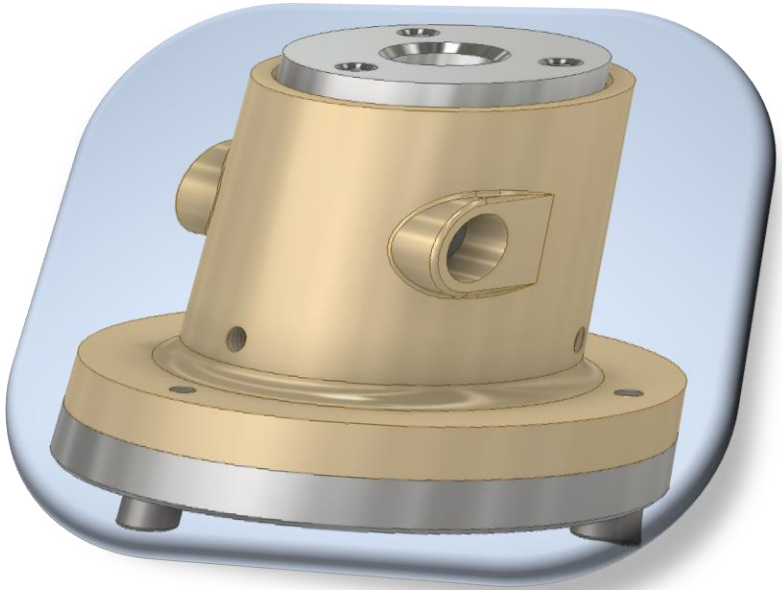
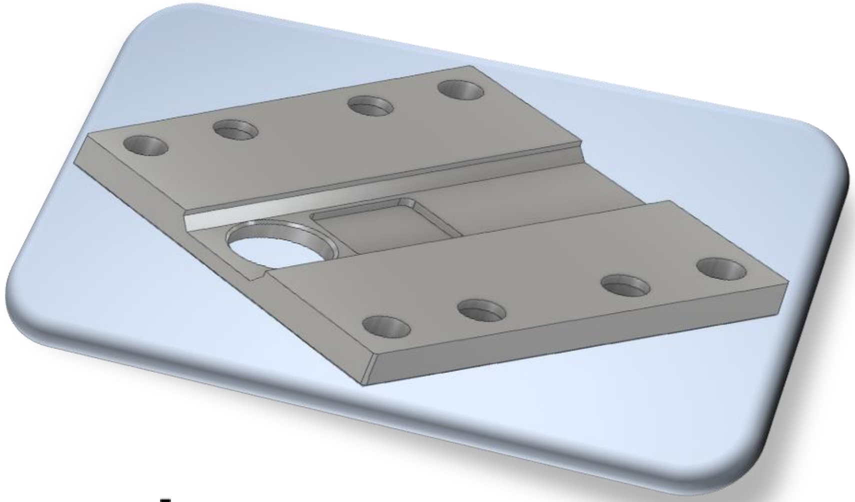


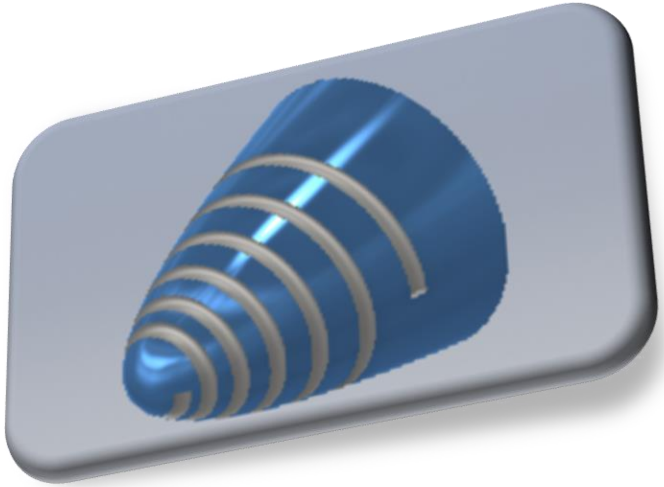


Want to learn more?

- Inventor tutorials in the Welcome Screen
- Youtube

Thank you





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