# CREATE A BASE CABINET

Tutorial

From 3D objects to a marketable base cabinet

The 3DVIA Documentation
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## INTRODUCTION

Cabinets are the basic elements of a kitchen: they content objects, they support the sink, or they integrate an oven or a hood. There are four types of cabinets: base, wall, high and corner cabinet. Each type of cabinet can be customized to arrive to a choice of hundreds of different models for the customer.

Thanks to this documentation, you will know how to create a sample base cabinet and be able to create various models from it.

For our sample case, let's assume that our Range contains as best-seller a classical base cabinet with a black wood facade, a white knob, a natural wood top, four legs and a shelf inside. That this cabinet exists in two widths, with three possible worktops, a left or right-opening door, and a vertical handle in option, all this up to the customer.

We have to create the different parts of this cabinet, enable and define the variations, assemble the components to a piece of furniture matching with the real one, set the pricing, and finally add the cabinet to a catalog.

This documentation is the first of a series of tutorials. It is also available online.

Note: The aim of this step-by-step documentation is to help you create a base cabinet manually. This might be a long proceeding if you want to add several cabinets and their variations to the database.

There is a way to deal with large catalogs, reduce delays and ease the global creation process by automating the most of it: by using the ByMe API and our libraries of 3D assets, assemblies and materials. Contact us for more information about this.

## PRE-REQUISITES

To follow the complete process described hereinafter, you must have access to the applications with the appropriate rights.

You also need the following:

- Range information: Products, the corresponding 3D models, the 2D and 3D representations of them, a description, pricing and information about their behavior.
- Information about the Legal Entity to which the range belongs.
- An Application Distribution: This distribution is the way used by the platform to launch the planner in a certain context (e.g. for a specific language, a specific country, a specific set of catalogs).

## OVERVIEW

The diagram below is an overview of the workflow to create a base cabinet, from the 3D assets until it is visible in the Company's Kitchen Planner for the customer.

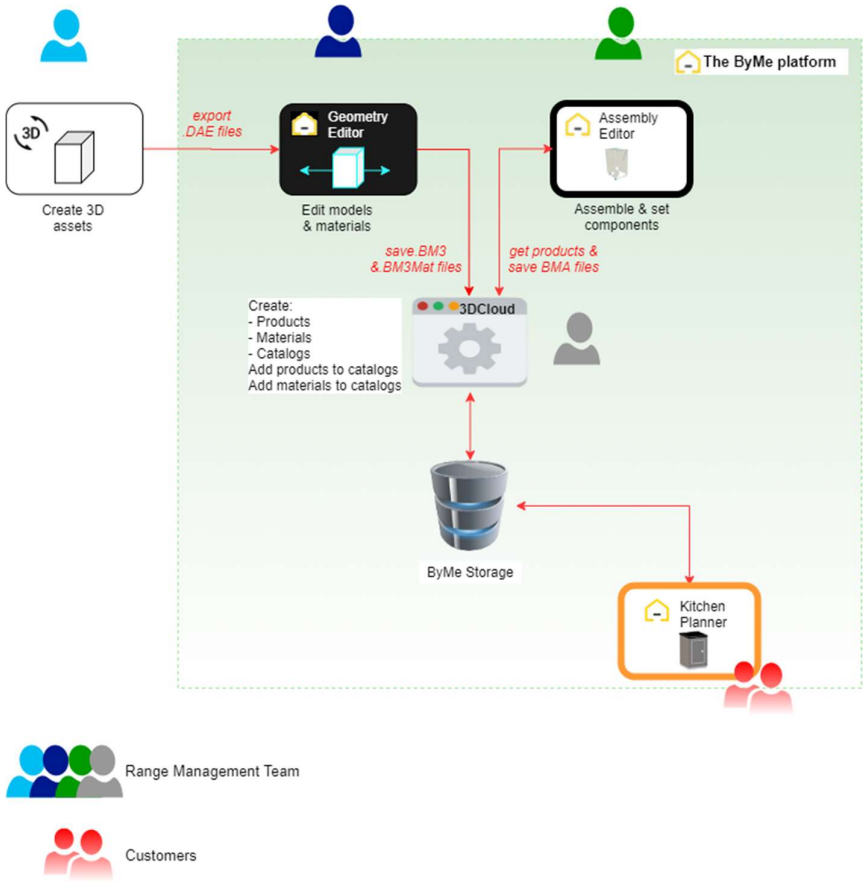



*Figure 1 – Workflow of the creation of a base cabinet*

Although Geometry Editor, Assembly Editor, 3DCloud and the Kitchen Planner are inside the same solution, their user is not unique. Each ByMe application requires specific skills.

- Geometry Editor requires Designer skills;
- Assembly Editor requires CAD skills;
- 3DCloud, which is the front-end of the ByMe database, requires transversal skills: a Range manager with publishing workflow skills;

In the case of automated processes, the dialog with the database via the ByMe API should be done by a developer.

## TERMINOLOGY

⇨ Refer to the Reference documentation of the ByMe platform for the complete list of the terms and concepts used in the applications.

Below is a short list of terms used in Geometry Editor and Assembly Editor. The table is ordered from content to container.

| | |
|---|---|
| **Geometry** | The shape of an object, represented as a mesh of triangles. |
| **Texture** | A bitmap file contributing to the definition of a material. |

| | |
|---|---|
| **Material** | The representation of the visual properties of an object, such as color, shininess, reflexivity, etc. This representation is a BM3MAT asset.<br>It can include several textures representing each of these properties.<br>Linear products (worktops, cornices, strips, etc.) using materials as resources are generated by the application depending on the context. |
| **Model** | The combination of a geometry with one or several materials. It is stored in Geometry Editor as a BM3 binary asset file. |
| **Product** | The datasheet of a model in 3DCloud, which will be stored in the database with a unique ID. This sheet includes text descriptions, tags, an image that will be used as a thumbnail, a link to the BM3 or a BM3MAT or a BMA file, various parameters, etc. |
| **Component** | A building element of Assembly Editor, calling a product and its parameters. This component will be set to define additional or overloading parameters, and its positioning in the space. |
| **Assembly** | Leveraging traditional CAD notions, an Assembly is the aggregation of several components, specifying their relative positioning through precise parameters and relations. It is created and managed by Assembly Editor as a BMA asset file. |

Note that the names change throughout the whole process, to illustrate their state change, as illustrated below:

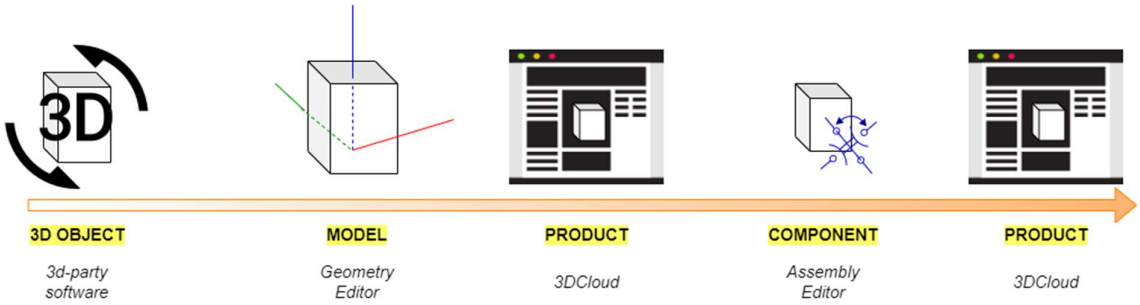
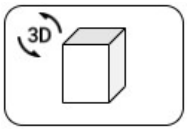



*Figure 2 – Name change and state change*

## CONVENTION

This “cabinet” icon indicates that the information relates specifically to the sample cabinet.




# STEP 1 – MODELING

The modeling is made in a third-party application. However, there are some recommendations about it.

## Prepare the Assets

The first step consists in creating all the assets that will result in the base cabinet offered for sale.

This work should be done according to the compatibility of the 3D modelling software with Geometry Editor, and the concordance of the software features with those of Geometry Editor.

Note: We recommend to use Solidworks 2018 as modelling software. The interoperability with Geometry Editor is native using the 3DCloud Solidworks plug-in.

The level of support provided by Geometry Editor is the intersection of three aspects:

- Is the 3D Modelling software compatible with Geometry Editor?
- Does the software provide an export format supported by Geometry Editor?
- Does the software support every geometric features that are required by Geometry Editor?

⇨ Refer to “Appendix 3” for detailed information on these questions and make the best choice for your configuration.

## Create the Objects

Create all the 3D objects needed to build a base cabinet. The final object “cabinet” is a combination of many 3D objects, such as a door and its handle, and of materials.

For our sample base cabinet, we will need to create the following models: a box, a door front, a knob, a handle, a leg, and a shelf. In addition, we will create a material: the worktop.

# STEP 2 – WORK WITH GEOMETRY EDITOR

The second step consists in importing the 3D object files into Geometry Editor, edit them (i.e. define which areas will be scalable and/or act on materials among others) and then, convert them to the Geometry file format.

These files are preferably in .DAE format, but it may be:

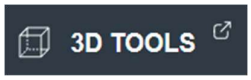
- .GLB files;
- A folder containing either a .BIN + a .GLTF + the required textures;
- A folder containing an .OBJ file + one or more .MLT files + one or more textures;
- A folder containing a .DAE file + an image folder.

⇨ See “Appendix 3 / Analyze the supported features” for further detail.

## Import 3D files into Geometry Editor

To import files, proceed as follows:

1. Connect to **3DCloud**, the UI of the ByMe database.
2. Scroll down to **3D Tools** and right-click to open Geometry Editor in a new tab.

   
   
3. Open Finder or the Windows Explorer to search for the .DAE file corresponding to the first model that you want to open in Geometry Editor.
4. Drag and drop it to the **Editor** tab. The 3D representation of the model is now visible, you can manipulate and edit it.

## Quick Overview of the Interface

⇨ Refer to the “ByMe Geometry Editor Reference” documentation for detailed information on the User Interface.

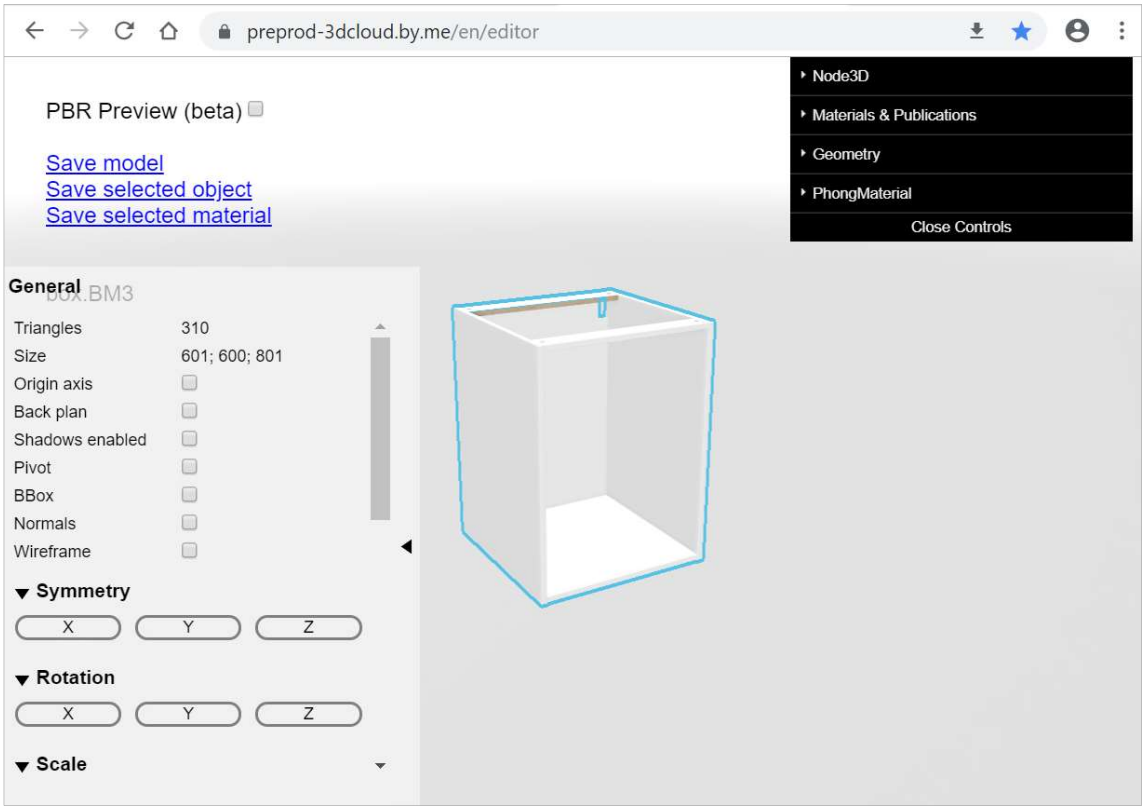



*Figure 3 – The Geometry Editor workspace*

The workspace is organized into four areas.

- The scene, which is the 3D representation area where you can manipulate the model.
- A left grey panel, where you can visualize and edit various options for the whole geometry.
- A right black panel, where you can visualize and edit options for the selected 3D node or material.
- A **Save** menu, depending on what is selected in the scene.

The Black Panel

The black panel is where you define material variations.

Select a node on the model to expend the menu, then expend each menu item to access the options.

Some options on the black panel have borders: options with a red border allow you to select a file; with green borders to enter text; with blue borders to enter numeric values.

The content of the black panel varies depending on the type of material (Phong or PBR) that is selected on the scene.

The Grey Panel

The grey panel is where you define scalable areas.

You can customize the workspace by selecting some of the settings in the **General** area. For example, check the **Origin axis** box to display the X, Y and Z-axis.

The Save Menu

This menu offers three options:

- **Save model:** Save the whole model as a .BM3 file.
- **Save selected object**: Save the selected node as a .BM3 file.
- **Save selected material**: Save the material (or the material of the selected node) as a .BM3MAT file.

## Edit 3D Models

The ByMe platform allows the dynamic modification of models (i.e. the model displayed to the user can be different from the 3D model file) by a Range Manager in 3DCloud, or by the customer in the Kitchen Planner.

Editing a model is essential if you want to create product variations. These variations might be to change the dimensions of the product or to change the material of the product.

| Variation | Name | Description |
|---|---|---|
| Enable size variation | Scaling | Increase or decrease the size of the objet while keeping the ratio between the three dimensions (height, width, depth). |
| | Smart scaling | Make an area be stretchable on an axis. |
| Change the color/material | Publication | Allows defining a face (i.e. a group of triangles) by an ID to be able to change applied material or color on this face via a parameter.<br>Applied material shall be in .BM3MAT format. |

Smart-Scaling

⇨ Refer to the "ByMe Geometry and Materials Reference" documentation for detailed information on scaling, especially "Preparing geometries and materials for variability".

It is possible to scale geometries, either uniformly on each axis (default scaling) or non-uniformly, the smart-scaling working independently on each axis. However, smart-scaling is incompatible with curved or round shapes.

You can set a smart-scaling to allow the customer make changes in the Kitchen Planner such as enlarging the cabinet.

Important: To avoid scaling artifacts, a model supporting smart-scaling *must* have mesh edges all along the limit between stretchable and non-stretchable areas.
Meshes that have been decimated / simplified will not guarantee this property and thus, will work poorly with smart-scaling.

For our sample cabinet, we will have to define a non-uniform scaling to allow the customer choose between two widths. The box, the door front and the shelf will be stretchable on the X-axis.

Material Publication

⇨ Refer to the "ByMe Geometry and Materials Reference" documentation for detailed information on preparing 3D assets to work properly and materials to remain homogeneous after publication.

Changing a material on an object or a part of it can be triggered by a parameter value change originating in 3DCloud (or the API), a product rule or the product edit panel of the Kitchen Planner.

A publication is a two-step action:

1. In Geometry Editor: Assign a publication to editable nodes of the product.
2. In 3DCloud: Add the matching material variation parameters.

For our sample cabinet, we can define a publication for the knob and the shelf.

## Save the Files

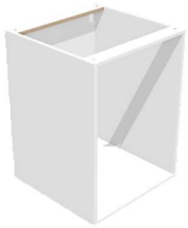
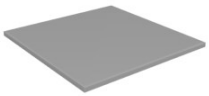
Saving models and materials in Geometry Editor generates the following file formats.

| | |
|---|---|
| BM3 | The native format for 3D assets, containing a mesh of vertexes, parametric scale where appropriate, geometries, embedded materials and textures.<br>In addition, save as models (.BM3 files) the items that do not require 3D nor variations, such as the legs. |
| BM3MAT | The native format for materials. It stores materials definition and textures, as well as ByMe additional concepts like UVOperators. |

Note: Models and materials are editable in the database through the 3DCloud User Interface.

## Sample Cabinet

To create our sample cabinet, you should have the following objects.

| Model & Material | File format | Sample |
|---|---|---|
| A box | BM3 |  |
| A front door | BM3 | |
| A handle | BM3 | |
| | | |
| A leg | BM3 | |
| A shelf | BM3 |  |
| A worktop | BM3MAT | |

## Check List before Editing

If you want to duplicate a model to change its material or color, we highly recommend to save them as different models and not as a unique model with variations. For example, we will create a white knob from the metal one to have two knob models.

| Model & Material | Scaling | Description |
|---|---|---|
| Box | ✔ Yes | Enable scaling for further width change |
| Door | ✔ Yes | Enable scaling for further width change |
| Knob | ✘ No | Changing the material to create a second knob |
| Metal Handle | ✘ No | |
| Leg | ✘ No | |
| Shelf | ✔ Yes | Enable scaling for further width change |
| Worktop | ✘ No | Linear material |

Note: You do not have to start with a product in particular; there is no hierarchical order between the products in Geometry Editor. However, there is a tree structure in Assembly Editor.

## About the Axes

Geometry Editor offers a 3-dimensional view of the model. Scaling will be made on these axes.

You can display the axis on the scene by selecting the **Origin axis** check box in the **General** area.

Important: The origin point has been defined when creating the 3D object. In our sample, object are centered on the origin point. This will be important in Assembly Editor, when defining the positions of the different components.

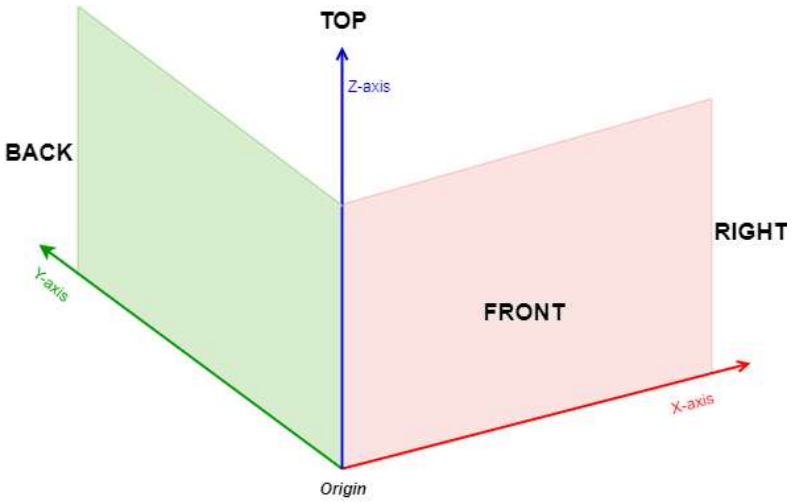



*Figure 4 – The X, Y and Z-axes*

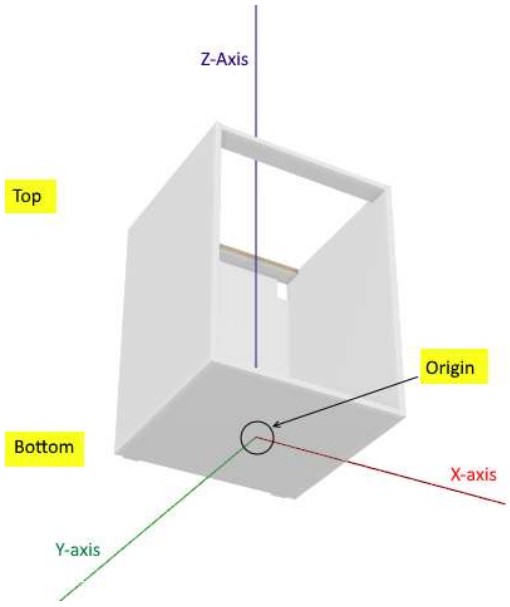



*Figure 5 – Axial positioning of an object in Geometry Editor*

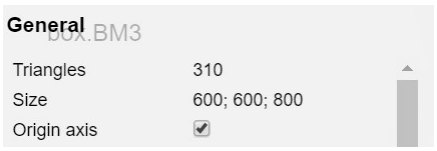
Note: Select the **Back plan** option to be sure that the model is well located in the space.

## About the Dimensions

The **General** area on the left panel gives essential information about the object:

- **Triangles**: The number of meshes of the 3D object.
- **Size**: The dimensions of the object, according to its positioning on the axes.

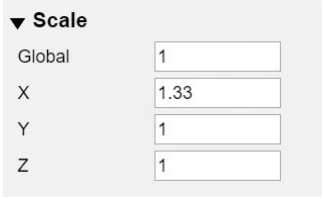
| | | | |
|---|---|---|---|
| **Value** | 600 | 600 | 800 |
| **Dimension** | Width | Depth | Height |
| **Axis** | X | Y | Z |

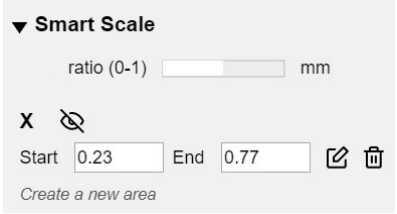



⇨ For performance reasons, the ByMe platform encourages low poly assets. Refer to the "Geometry and Materials Reference" documentation for detailed information on "Large Models, LOD and Simplification".

## About the Ratio

The **Scale** fields, used to stretch a whole axis and not a part of it, are expressed as a ratio. By default, the **Smart Scale** fields are expressed as a ratio too.



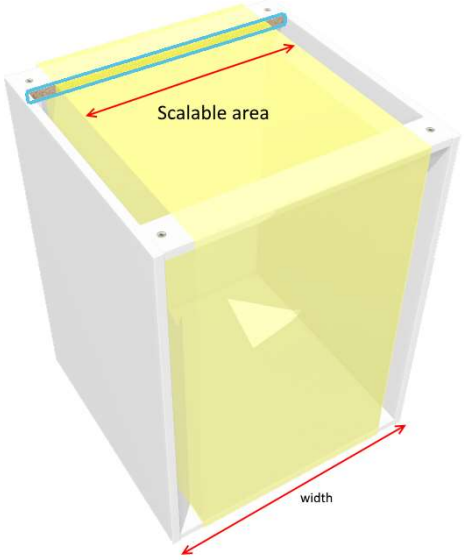
The ratio is calculated from the actual size of the object, which is “1”. Therefore, to double a dimension on an axis, enter 2; to reduce by a half, enter 0.5.

When defining a Smart Scaling area, choose to work with millimeter values by switching the white **Ratio** cursor to the right.

About the Scalable Areas

When you stretch uniformly a whole object, the stretch area includes the side panels in the case of a box, or the molding in the case of a door.

To keep the size of these parts, you have to define scalable areas. This will define non-scalable areas by exclusion.




This illustration shows a basic cabinet with top-holes to fix the worktop on it and back-holes to fix the cabinet to the wall. The scalable area should exclude:

- The left and right panels of the box
- The top and the back-holes

The illustration shows in yellow the area that should be defined as scalable. Outside this yellow area, all will be excluded from the scaling.

This means that you only have to enter the start and end positions of the scalable area.

Move the object on the scene to be sure to locate all the parts to exclude.

Important: Define the positions of the scalable area carefully to ensure the behavior in Assembly Editor.

Edit the Box to Define Scaling

Our work on the box is to define scaling to allow the widening of the box, which default size is 60 cm, to 80 cm.

Important: The box will not be stretched uniformly (this would increase or reduce the whole box including the side panels among others) but on the X-axis only *and* inside the concerned area to keep the thickness of the panels. That is why the dimensions will be set in the **Smart Scale** area.

To define the scaling, proceed as follows:

1. Scroll down in the left panel to the **Smart Scale** area.
2. Choose to express the dimensions in millimeters. By default, the fields are specified with the dimensions of the box.
3. Click the **View** button to display the scalable area in yellow. By default, the whole box is included in the area.

Now you have a choice between entering the values manually and using the mouse.

### *Enter the Values Manually*

If you already know the values, enter them in the **X** input fields. The area should exclude the side panels, the holes on the top and the holes on the back. For our example:

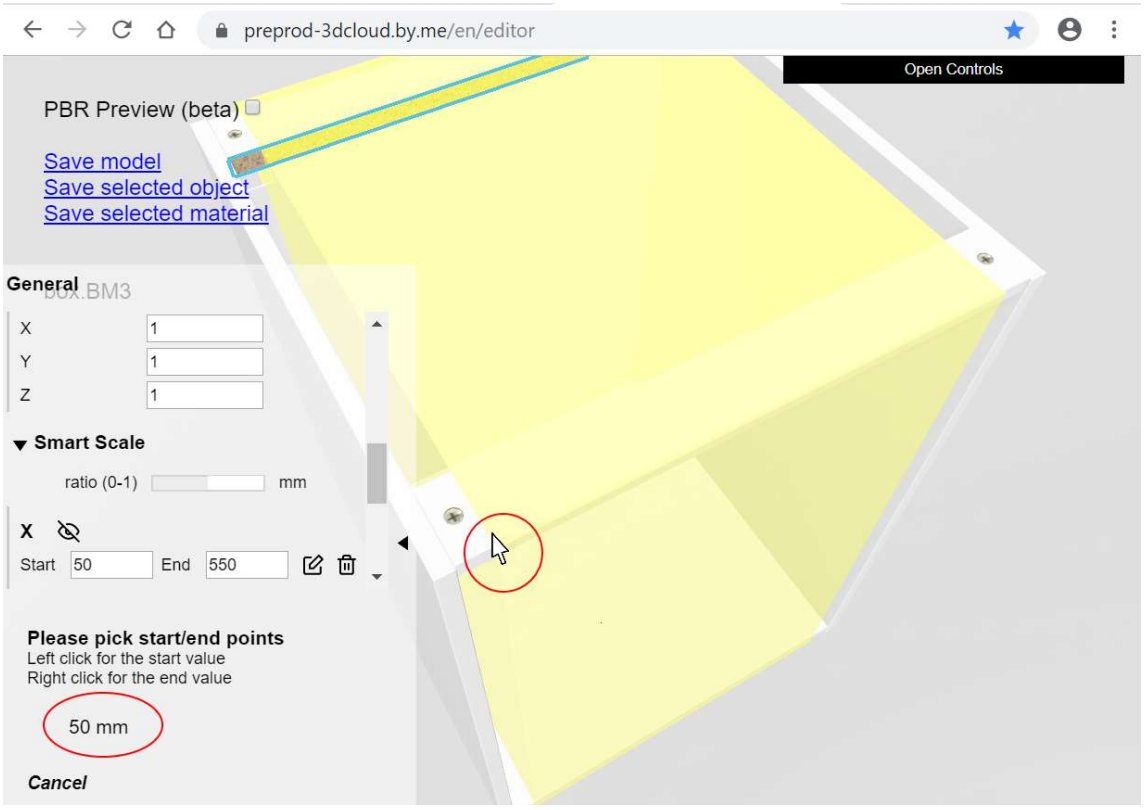
1. Enter 71 in the **Start** field. This is where our scalable area starts on the X-axis.
2. Enter 534 in the **End** field. This is where our scalable area ends on the X-axis.

### *Click to Define the Values*

If you do not already know the values, you can define them by clicking them on the scene.

1. Use the mouse wheel to zoom in the object on the scene.
2. Click the **Edit** icon on the right. This opens a **Pick up** area.

3. Click with the left button of the mouse where you want the scalable area to start.
4. Click with the right button where you want the scalable area to finish.

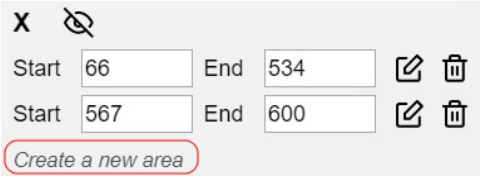



5. Click **Cancel** to exit from the pick-up area. The new values are taken into account.

Note: Using the mouse to define the area can be very precise: the exact position in millimeters is displayed while moving the mouse.

However, it is recommended to adjust values by rounded, symmetric and coherent numbers.

In the case you want to define several scalable areas on an axis, you have to add lines to the **X** area by clicking **Create a new area**.




### *Test the Scaling*

You can test the scaling by clicking **Test Smart Scale** on the bottom of the grey panel.

Go to **Scale** and enter a value in the **X** field, e.g. 1.5 to stretch to 50% more, or use the arrows to increment the value.

### *Save the Changes*

Click **Save model** to save the box and its scaling parameters as a BM3 file.

Note: The variation from 60 cm to 80 cm will be defined in 3DCloud.

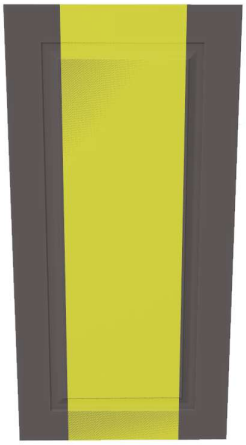
## Edit the Door Front to Define Scaling

Our work on the door front is to define scaling to enable a width variation of the central panel on the X-axis, excluding *de facto* the frame and the moldings.

The dimensions of our sample door are width 598 / depth 20 / height 798 mm.

To define the smart scaling, proceed as follows:

1. Scroll down in the left panel to the **Smart Scale** area.
2. Choose to express the dimensions in millimeters and click the **Eye** icon. By default, the whole surface will be in the area.
3. Click the **Edit** button to pick up the values to reduce this area.
4. Zoom in and click left to define the start value along the left molding. In our case, the value is 94.
5. Click right to define the end value along the right molding. In our case, the value is 305.
6. Click **Cancel** to exit from the pick-up area. The new values are taken into account.
7. Test the smart scaling if you want.

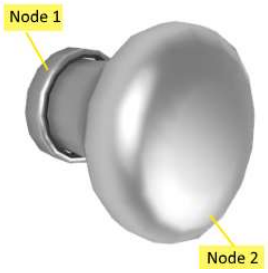


### *Save the Changes*

Click **Save model** to save the door and its scaling parameters as a BM3 file.

## Edit the Knob to Create a Duplicate

You may want to create a variation of a model, e.g. the same knob with another material and another color. The knob is made of two nodes (the nodes are defined during the modeling) and we will make a publication on the node corresponding to the button.



To create a duplicate of the metal knob, proceed as follows:

1. Click the button on the scene to select it as Node3D.
2. Specify a name if you want (press **Enter** to validate).
3. Go to **Materials & Publications**.
4. Click **Load material** to open the Finder or the Windows Explorer and search for the BM3MAT file corresponding to the new material. For example, a white acrylic material file.
5. Click **Open** to load the file and add it to the list of choices named **Scene materials**.
6. Rename the material if need be by specifying the **Name** field in the **PhongMaterial** area (press **Enter** to validate).

The **Scene material** drop-down list now contain a White Acrylic variation:

If you want to change the material of the ring, select the corresponding node and then proceed as for the button.

### *Save the Changes*

Click **Save the model** to save the changed knob and its parameters as a .BM3 file.

Now, you have two different knobs at disposal.

## Edit the Handle to Save it as Model

There is no scaling or publication work to do in Geometry Editor with the metal handle.

But you have to save it as a .BM3 file in order to import it in 3DCloud at the next step. Click **Save model**.

## Edit the Shelf to Define Scaling

The work on the shelf is to enable the scaling on the X-axis, as for the box and the door. In addition, you might want to change its color.

Because there are no areas to exclude for the shelf, you can define a simple scaling.

1. Scroll down in the left panel to **Scale**.
2. Specify the **X** field with a ratio of 1.33 to stretch the shelf to 33% more on the X-axis.

Note: You can change the color of the shelf only if the model is a .SVG file or if the material has no texture. Else, change the material (see above the proceeding with the knob).

To change the color, proceed as follows:

1. Select the object on the scene, to select the node on the black panel.
2. Scroll down to **PhongMaterial**.
3. Roll over the **Diffuse color** field to display a color palette.
4. Pick up the wished color.
5. Change the **Specular color** in addition if need be.

### *Save the Changes*

Click **Save the model** to save the shelf and its parameters as a .BM3 file.

## Edit the Worktop

The worktop is a linear material, i.e. a .BM3MAT file. As a linear, it is stretchable by definition and there is no scaling to define.

The substitution of one worktop by another will be set in 3DCloud.

You might want to change the tiling operator. In this case, scroll down the black panel to **UVOperator** and change the **ScaleX** and **ScaleY** values to adapt them to your needs. The tiling is expressed in millimeters.

### *Save the Changes*

Click **Save selected material** to save the worktop as a .BM3MAT file.

## Save the Leg as BM3

There is no edition work to do on the legs, but it need to be converted into a .BM3 file to be used as product and then component in 3DCloud and Assembly Editor.

Drag and drop the leg 3D model in Geometry Editor and then, click **Save as model**.

# STEP 3 – CREATE PRODUCT DATASHEETS

⇨ For detailed information on the 3DCloud user interface, see the "3DCloud" documentation.

The next step consists in creating in 3DCloud one datasheet per model (.BM3 file). Once the datasheet is created, the model becomes a product.

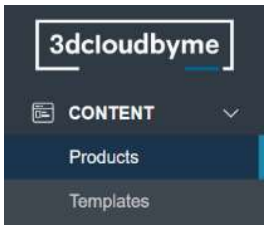
Each datasheet will define the name of the product and the descriptions to display in the Kitchen Planner (both in various languages); link the model to a type and a brand; define its period of availability and its pricing; define various tags to reference it; contain a 2D and a 3D representation if need be; contain size parameters and options if need be.

Important: This step is mandatory to make the products available in Assembly Editor, including the articles that have not been edited in Geometry Editor, such as the legs.

## Create Product Datasheets

To create a *product* datasheet, proceed as follows:

1. Open **3DCloud** and go to **Content** / **Products**.
2. Click **New product** in the **Products** tab.




At this step, the database automatically assigns an ID to the product. To choose the ID of the product, you have to use the API.

For our sample cabinet, you have to create datasheets for each product (box, door front, handle, shelf and the legs). As best practice, do not create any material datasheet, because materials should have been applied in the .BM3 3D model.

## About the Language

The possible languages are defined in the application distribution. The languages available for the **name** field and the **Commercial information** drop-down list depend on these settings.

## About the Product Types

Selecting a product type is mandatory because it will trigger the applicative rules.

If you hesitate between several similar types, use the type that describes the destination of your product. For example: if you have a choice between several types of doors, do not use "doors" but specify that the door is the front of the cabinet and select "front doors".

⇨ See Configuration Guide / Predefined Applicative Rules in the Reference documentation.

## About Descriptions

Although the description fields are not mandatory, they are highly recommended.

The commercial description and the short description will be displayed to the customer in the product page of the Kitchen Planner.

You can write as many words as wished in the **Commercial description** area. If the text exceed the text area, a scroll bar will appear.

However, the number of words is limited in the **Short description** text area. A best practice is to limit the short description to five words (keep in mind that description will be translated).

## About Tags

Tags are not mandatory for Kitchen, because they are not used as filters in the Kitchen Planner. However, you can specify that your product is intended for the "kitchen" room.

## About Parameter and IDs

You will have to define parameters in the datasheets and give them an ID.

They make the link between the 3D representation (the .BMA file) and the product containing this 3D representation. The mapping is done by creating parameters using the same name (case sensitive) in 3DCloud *and* in Assembly Editor.

Parameters are defined at three levels:

1. At "simple product level", when products do not belong to an assembly yet.
2. At "sub-assembly level", when the product are combined in a sub-assembly.
3. At "top-assembly level", when products and sub-assemblies are grouped together into a final assembly.

Important: The size parameters (width, depth, height) are mandatory at each level.

### Simple Product Level Parameters

You will define size parameters for all the products, referring to the dimensions of the 3D model.

Select **dimension variations** to define the fixed or variable width, height and depth for each product.

Additionally, you have to define 'type' options for the fronts and handles.

For our sample cabinet, you will define at this step the parameters of the first level, that is to say the dimensions of the box, the door front, the handles, the shelf and the legs. In addition, the options of the door front and the handles.

### Sub-Assembly Parameters

It is a best practice to create product combinations, such as "door front and handle" and "box and legs". These combinations are named sub-assemblies.

⇨ See below "Step 5 – Datamodel" for detailed information.

These sub-assemblies require specific parameters (i.e. options) to define their behavior.

### Top-Assembly Parameters

Once all the products and sub-assemblies are grouped together into a single product in Assembly Editor, the result is named "top-assembly" and must have its own product datasheet with new parameters.

Among these parameters, you will define options to enable the worktop in the Kitchen Planner.

⇨ See below "Step 6 – Save the Cabinet as Product".

Writing Parameter IDs

Below are some rules and recommendations regarding these IDs.

- IDs are case sensitive.
- Respect the camel case convention: capitalize the first letter of the second and following words. E.g. backLegPosition.
- Never use these characters: *, \, /, @, &, {, }, [, ], %, £, $.
- Never use vowels with accents or umlauts.
- Use generic words.
- To ensure that the desired behavior will happen, the parameter needs to have the same ID as the BM3 material publication or BMA material parameter. The same for the width, depth, height parameters of the BM3.
- In some cases, they should match expected parameter IDs in the application (e.g. worktopOption) as defined in the parameter dictionary.

⇨ Refer to the “Parameter Dictionary” for detailed information on parameters.

## About Pricing

Pricing depends on your catalog. For some catalogs, the pricing is specified for each single product, and so the price of the cabinet is the sum of each products, for others it is defined at the top-assembly level for each combination of the cabinet. In your case, you have to choose.

For our sample cabinet, we will define the pricing at the top-assembly level. Leave these fields blank during this step.

## Datasheet Information Overview

Below is an overview of the information that should be specified, depending on the level.

Note: Current Step 3 only deals with single products. Else, see Step 5 and Step 6.

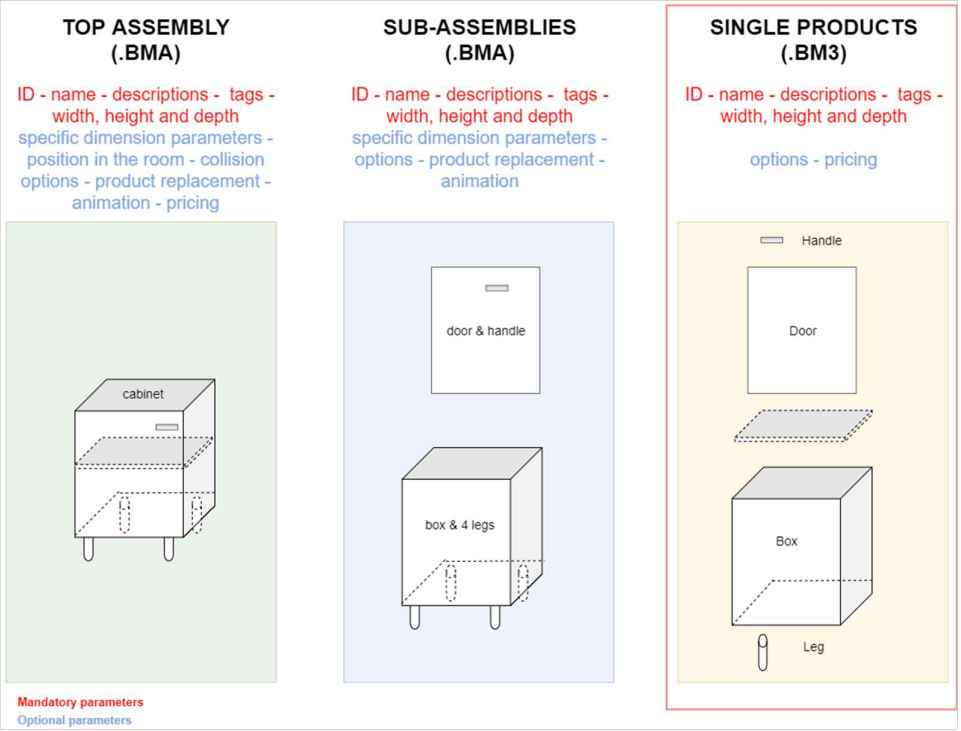
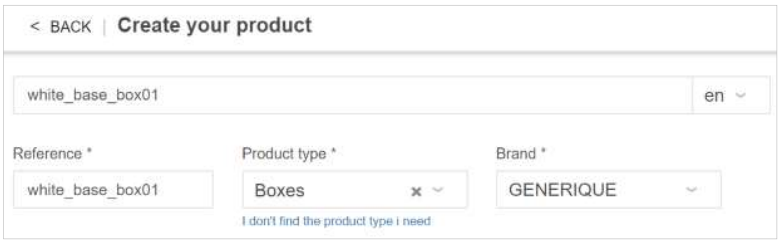



*Figure 6 – Datasheets parameters and options*

## Create the Box Datasheet

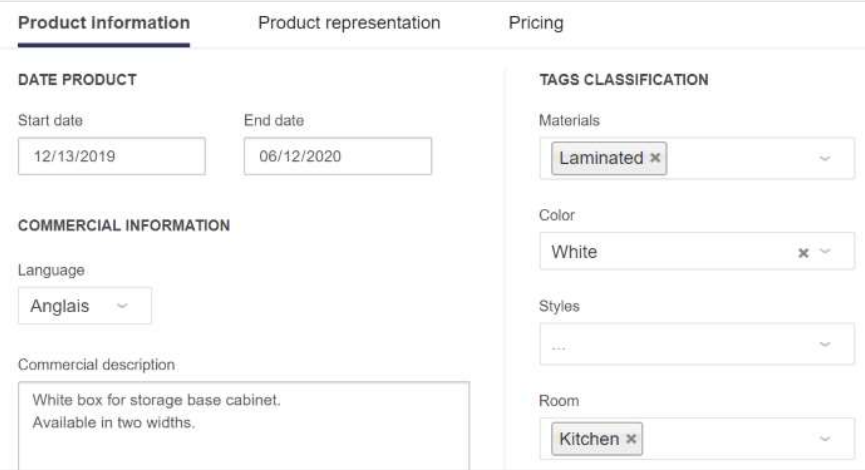
To create the datasheet for the box, follow the steps illustrated below.

General Information




1. Enter the name of the product: select a language in the drop-down list and then, enter a name. Repeat this step for each language. Note that this name will be displayed to the customer.
2. Enter your reference for the product. This reference may be displayed to the customer and is used in Assembly Editor as well.
3. Link the product to a product type, i.e. **Boxes** because it is a box.
4. Link the product to your Brand.

## Product Information



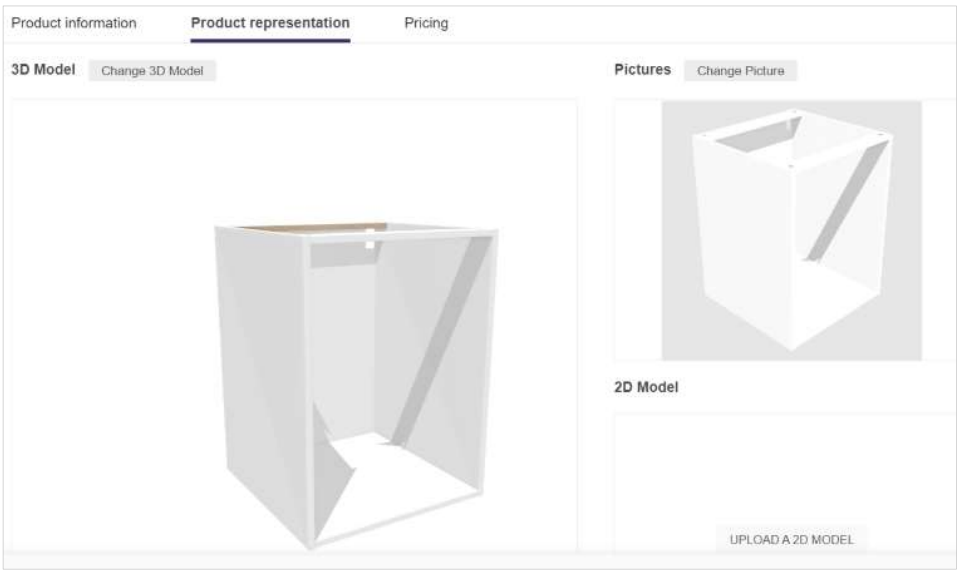

1. Define the period of availability, if need be. Note that these dates always overwrite the dates set in the catalog.
2. Choose a language to use in the Kitchen Planner.
3. Enter the commercial description and a short description of the box.

## Product Representation

The **Product Representation** tab is essential when creating the datasheet, because:

- It is where you upload the model, i.e. the .BM3 file;
- It is where you set up variations, options and mapping parameters.

### *3D and 2D Representations*



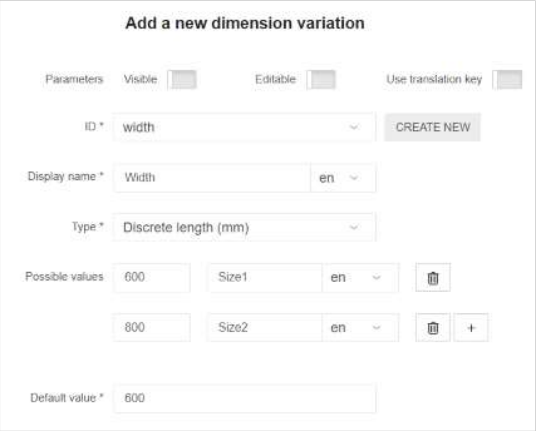

1. Add the BM3 file by clicking **Upload a 3D model**.
2. Add a bitmap picture of the box by clicking **Upload a picture**. It will be used as thumbnail in the product list in 3DCloud and in the Kitchen Planner.

### *Technical Information*

Not applicable.

### *Variable Dimension*

We have defined a smart scaling on our box to enable its widening in the Kitchen Planner. Thus, specify the dimensions by clicking **Add variable dimension**.

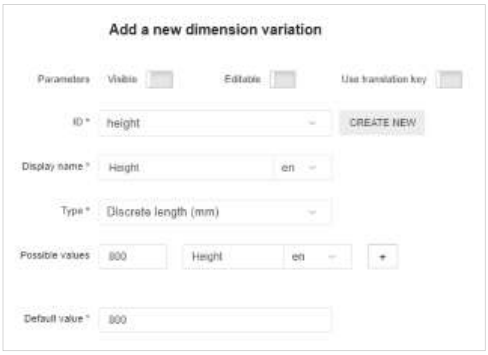



1. Select **width** in the **ID** drop-down list (or create it if need be).
2. Enter a label for the ID in the **Display name** field. Capitalize the first letter to avoid confusion with the ID.
3. Select **Discrete length (mm)** in the **Type** list. This will allow the customer to *choose* between different possible values, while the **Continuous length** would allow the customer to *enter* a value between two limits.
4. Enter the possible values in the corresponding fields, with a name to display to the customer. For our box, enter **600** and **800**.
5. Specify the default value, i.e. **600**.
6. Click **Validate** to save the variation.

The dimension variation is now defined and visible in the **Variations** area.

### *Fixed Dimensions*

Repeat the proceeding to specify the height and the depth of the box.



Now the box datasheet contain the following variations.

Save

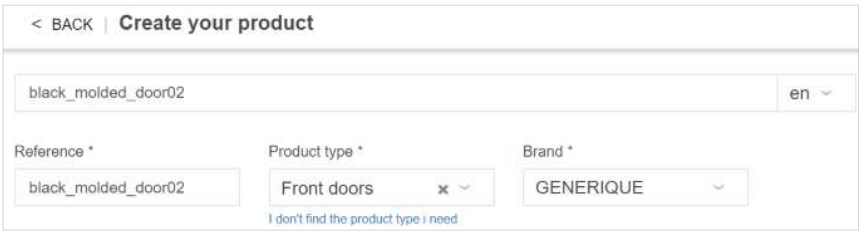
Click **Save** to upload the model and its datasheet in the 3DCloud storage. Click **Products** to see the new product in the list.

When the product is successfully added (a delay is possible), the **3D model** column displays **Yes** in the product list.

## Create the Door Front Datasheet

To create the datasheet for the door front, follow the steps illustrated below.

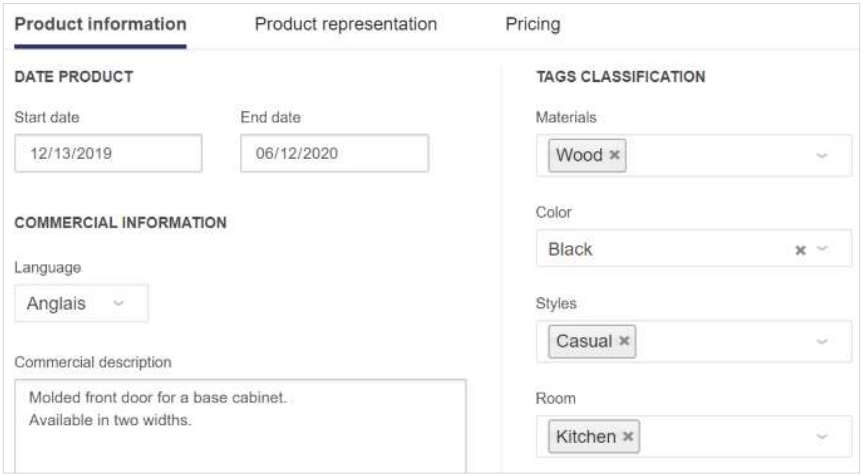
## General Information




1. Enter the name of the product: select a language in the drop-down list and then, enter a name. Repeat this step for each language. Note that this name will be displayed to the customer.
2. Enter your reference for the product. This reference may be displayed to the customer and is used in Assembly Editor as well.
3. Link the product to a product type, i.e. **Front doors** because it is a front for the door of the future base cabinet.
4. Link the product to your Brand.

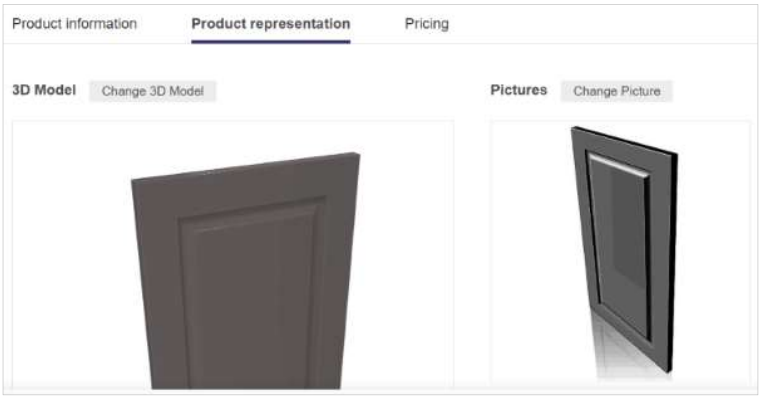
## Product Information

1. Define the period of availability, if need be. Note that these dates always overwrite the dates set in the catalog.
2. Choose a language to use in the Kitchen Planner.
3. Enter the commercial description and a short description of the door.
4. Define tags to ease the classification of the door in the Kitchen Planner.




## Product Representation

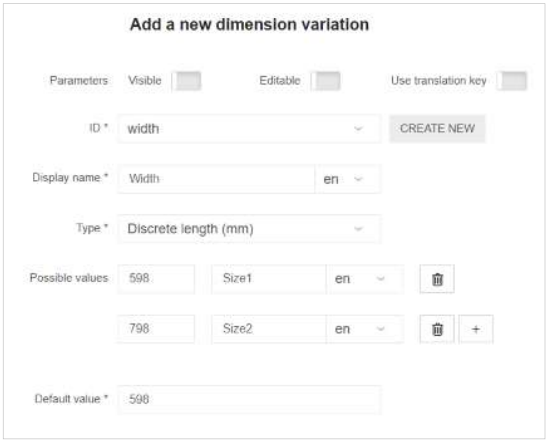
Start by adding the 3D and 2D representations of the product.




1. Add the .BM3 file by clicking **Upload a 3D model**.
2. Add a bitmap picture of the door by clicking **Upload a picture**. It will be used as thumbnail in the product list in 3DCloud and in the Kitchen Planner.

### *Variable Dimension*

We defined a smart scaling on our door to enable its widening in the Kitchen Planner. Thus, specify the dimensions by clicking **Add variable dimension**.




1. Select **width** in the **ID** drop-down list (or create it if need be).
2. Enter a label for the ID in the **Display name** field. Capitalize the first letter to avoid confusion with the ID.
3. Select **Discrete length (mm)** in the **Type** list. This will allow the customer to *choose* between different possible values, while the **Continuous length** would allow the customer to *enter* a value.
4. Enter the possible values in the corresponding fields, with a name to display to the customer. For our box, enter **598** and **798**.
5. Specify the default value, i.e. **598**.
6. Click **Validate**.

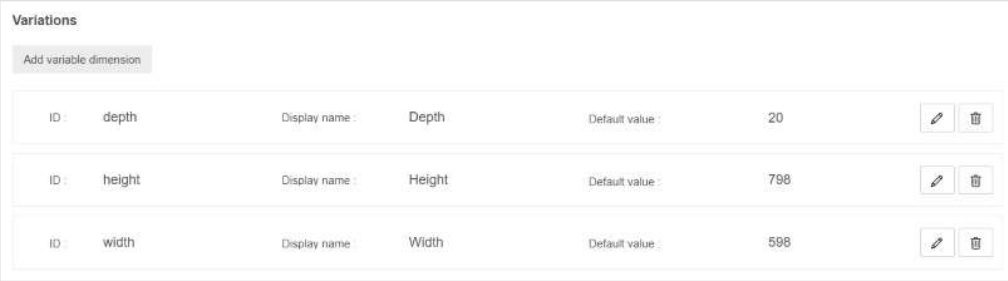
### *Fixed Dimensions*

Specify the height and the depth of the box by clicking **Add variable dimension** (even if the dimension will not be variable).

1. Select **height** in the **ID** drop-down list (or create it if need be).

2. Enter a label for the ID in the **Display name** field. Capitalize the first letter to avoid confusion with the ID.
3. Select **Discrete length (mm)** in the **Type** list. This will allow you to *enter* a value.
4. Enter the possible values in the corresponding fields, with a name to display to the customer.
5. Specify the default value.
6. Click **Validate**.
7. Repeat the proceeding to specify the depth.

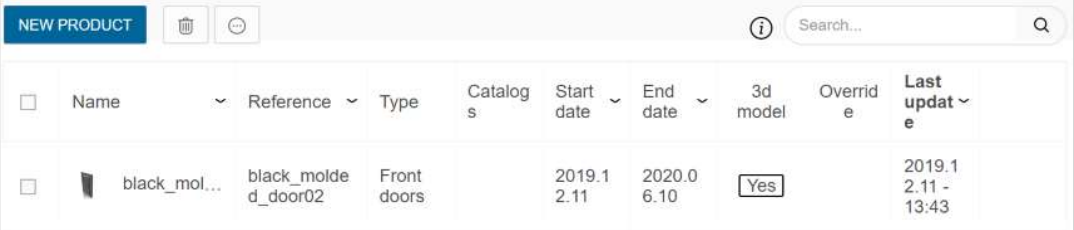
Now the door datasheet contains the following variations.




Save

Click **Save** to upload the model and its datasheet in the 3DCloud storage. Click **Products** to see the new product in the list.

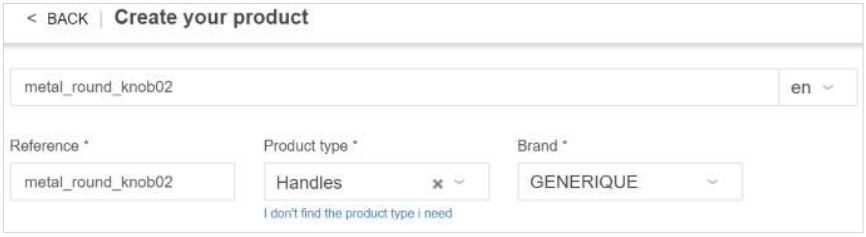
When the product is successfully added (a delay is possible), the **3D model** column displays **Yes** in the product list.




## Create the Knob Datasheet

To create the datasheet for the knob, follow the steps illustrated below.

General Information

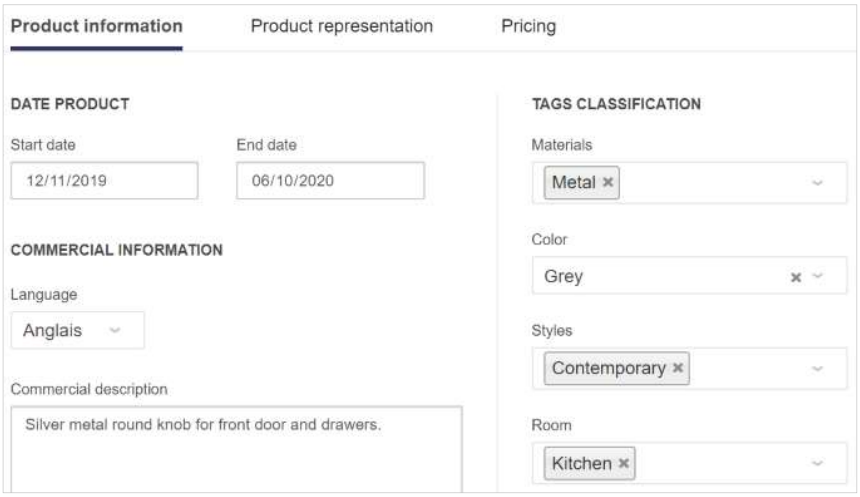



1. Enter the name of the product: select a language in the drop-down list and then, enter a name. Repeat this step for each language. Note that this name will be displayed to the customer.

2. Enter your reference for the product. This reference may be displayed to the customer and is used in Assembly Editor as well.
3. Link the product to a product type, i.e. **Handles** because a knob is a type of handle.
4. Link the product to your Brand.

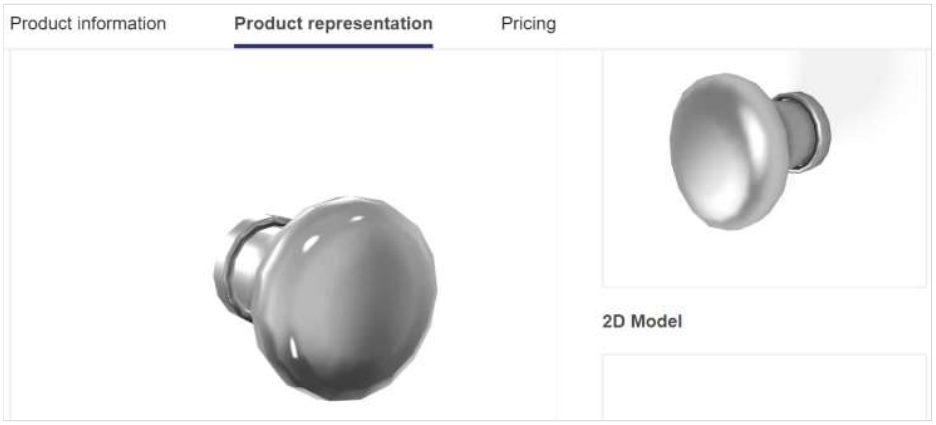
## Product Information

1. Define the period of availability, if need be. Note that these dates always overwrite the dates set in the catalog.
2. Choose a language to use in the Kitchen Planner.
3. Enter the commercial description and a short description of the knob.
4. Define tags to ease the classification of the knob in the Kitchen Planner.




## Product Representation

Start by adding the 3D and 2D representations of the product.




1. Add the BM3 file by clicking **Upload a 3D model**.
2. Add a bitmap picture of the knob by clicking **Upload a picture**. It will be used as thumbnail in the product list in 3DCloud and in the Kitchen Planner.

### *Type Option*

These variation parameters will complete the publication made in Geometry Editor.

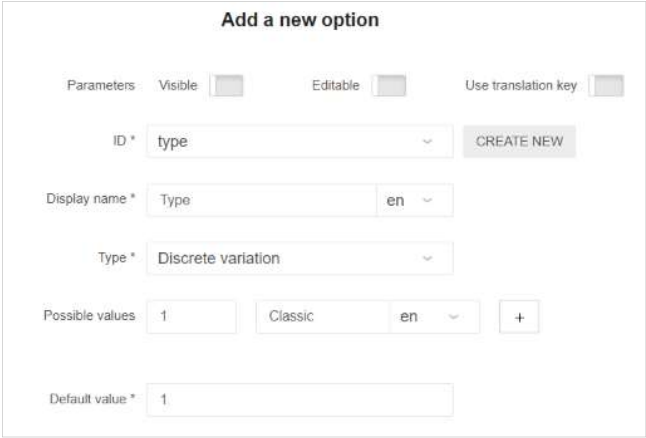
⇨ See Appendix 2 for the list of possible parameters for a base cabinet.

Handles can be of three different types:

- Type 1: A classic handle, to fix on the front door
- Type 2: An edge handle, to fix on the edge of the door
- Type 3: A drop handle

This type parameter is mandatory, unless you will not be able to place the knob on the door front.

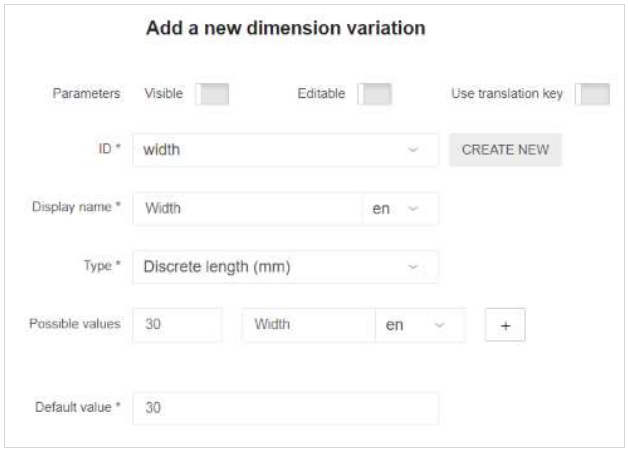
Click **Add option**.



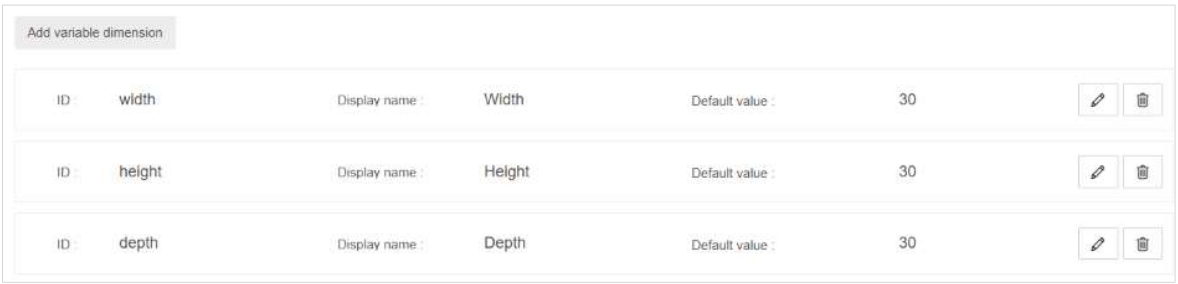

1. Click **Create new** to specify **type** as the ID. Click the **+** to add it to the **ID** field.
2. Enter a label for the ID in the **Display name** field. Capitalize the first letter to avoid confusion with the ID.
3. Select **Discrete variation** in the **Type** list. This will allow you to *enter* a value.
4. Enter **1** and **Classic** as unique possible value.
5. Specify the default value, i.e. **1**.
6. Click **Validate**.

### *Fixed Dimensions*

Define the dimensions of the handle by clicking **Add variable variations**. This variation is used for any dimension specification, even if it will not vary.




7. Select **width** in the **ID** drop-down list (or create it if need be).
8. Enter a label for the ID in the **Display name** field. Capitalize the first letter to avoid confusion with the ID.
9. Select **Discrete length (mm**) in the **Type** list. This will allow you to *enter* a value.
10. Enter the size in the **Possible values** fields.
11. Specify the default value, i.e. **30**.
12. Click **Validate**.
13. Repeat the proceeding to set the height and depth parameters with the same dimension.




### *Material Replacement*

Important: You will define a material replacement in the datasheet only if it has not been defined in the BM3 model itself in Geometry Editor and on condition that the other material *has* a material datasheet.

As a best practice, we duplicated the knob in two models after a material publication in Geometry Editor. That is why a material replacement is not required in 3DCloud.

### *Product Replacement*

Important: Product replacement is only used on sub-assemblies, or top-assemblies.

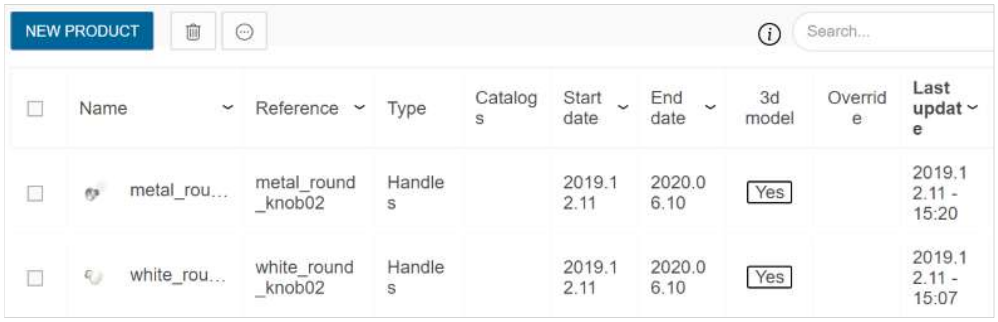
To allow the customer to choose between our three possible handles (two versions of the knob and the metal handle), we will define a product replacement in the datasheet “front and handle” sub-assembly.

⇨ See “Step 5 / Save the Front and Handle Sub-Assembly”.

Save

Click **Save** to upload the model and its datasheet in the 3DCloud storage. Click **Products** to see the new product in the list.

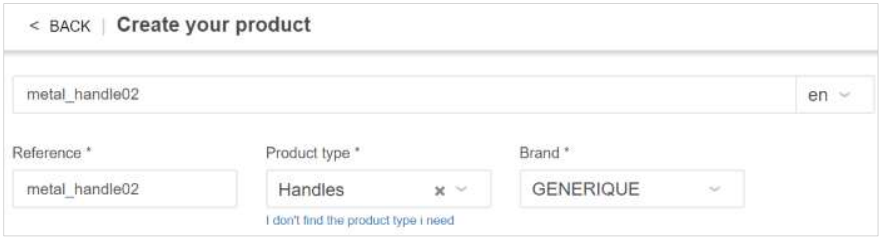
When the product is successfully added (a delay is possible), the **3D model** column displays **Yes** in the product list.




## Create the Metal Handle Datasheet

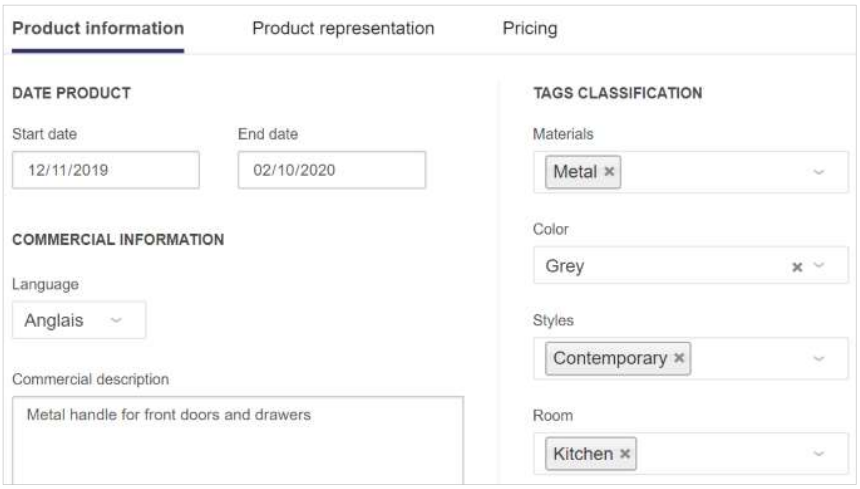
To create the datasheet for the metal handle, follow the steps illustrated below.

General Information




1. Enter the name of the product: select a language in the drop-down list and then, enter a name. Repeat this step for each language. Note that this name will be displayed to the customer.
2. Enter your reference for the product. This reference may be displayed to the customer and is used in Assembly Editor as well.
3. Link the product to a product type.
4. Link the product to your Brand.

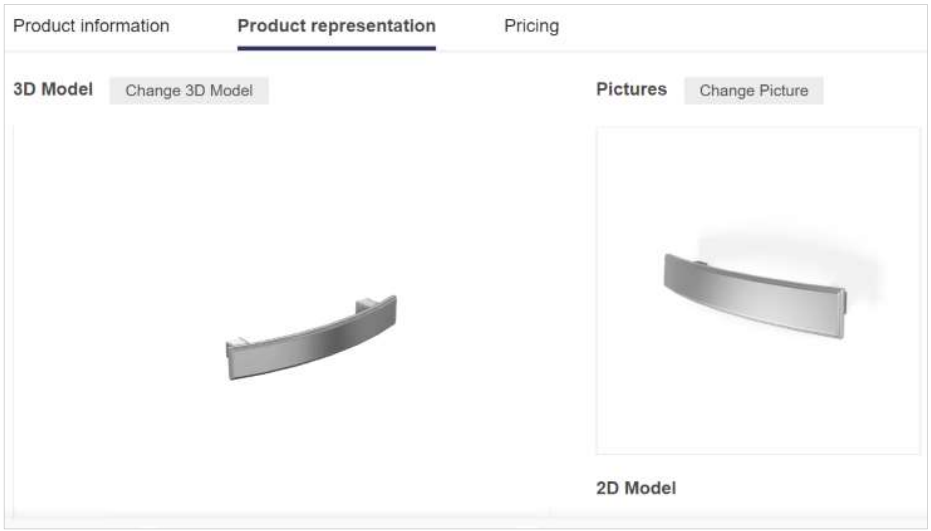
## Product Information




1. Define the period of availability, if need be. Note that these dates always overwrite the dates set in the catalog.
2. Choose a language to use in the Kitchen Planner.
3. Enter the commercial description and a short description of the handle.
4. Define tags to ease the classification of the handle in the Kitchen Planner.

## Product Representation

Start by adding the 3D and 2D representations of the product.




1. Add the BM3 file by clicking **Upload a 3D model**.
2. Add a bitmap picture of the handle by clicking **Upload a picture**. It will be used as thumbnail in the product list in 3DCloud and in the Kitchen Planner.

### *Type Option*

These variation parameters will complete the publication made in Geometry Editor.

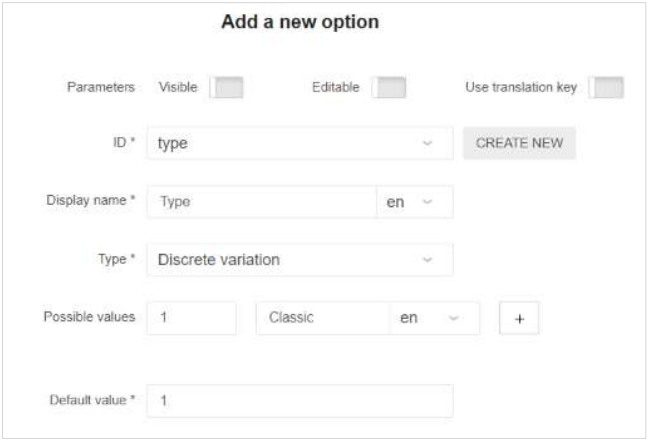
⇨ See Appendix 2 for the list of possible parameters for a base cabinet.

Handles can be of three different types:

- Type 1: A classic handle, to fix on the front door
- Type 2: An edge handle, to fix on the edge of the door
- Type 3: A drop handle

This type parameter is mandatory, unless you will not be able to place the handle on the door front.

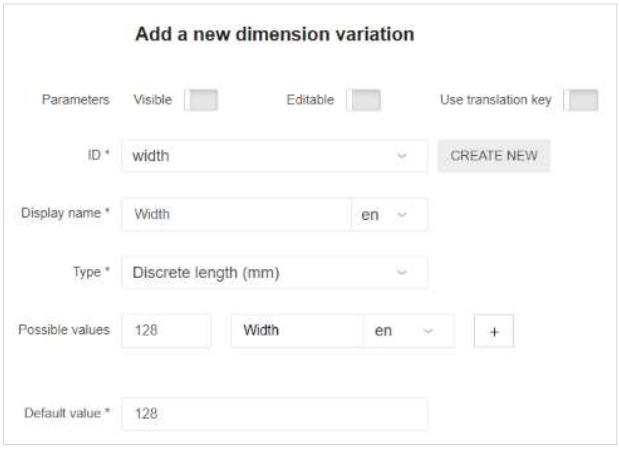
Click **Add option**.




1. Click **Create new** to specify **type** as the ID. Click the **+** to add it to the **ID** field.
2. Enter a label for the ID in the **Display name** field. Capitalize the first letter to avoid confusion with the ID.
3. Select **Discrete variation** in the **Type** list. This will allow you to *enter* a value.
4. Enter **1** and **Classic** as unique possible value.
5. Specify the default value, i.e. **1**.
6. Click **Validate**.

### *Fixed Dimensions*

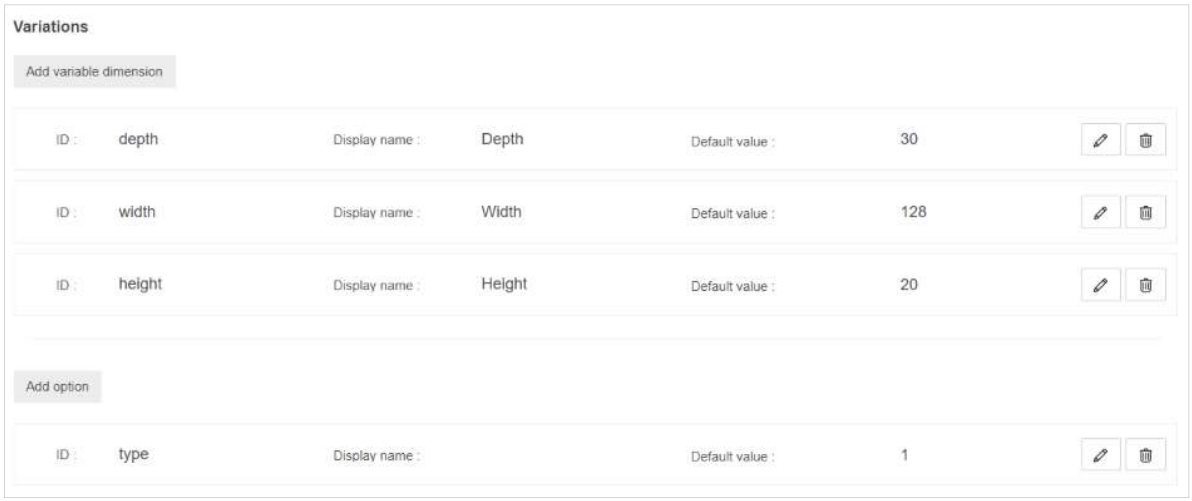
Define the dimensions of the handle by clicking **Add variable variations**. This variation is used for any dimension specification, even if it will not vary.




1. Select **width** in the **ID** drop-down list (or create it if need be).

2. Enter a label for the ID in the **Display name** field. Capitalize the first letter to avoid confusion with the ID.
3. Select **Discrete length (mm)** in the **Type** list. This will allow you to *enter* a value.
4. Enter the size in the **Possible values** fields.
5. Specify the default value, i.e. **30**.
6. Click **Validate**.
7. Repeat the proceeding to set the height and depth parameters with the same dimension.

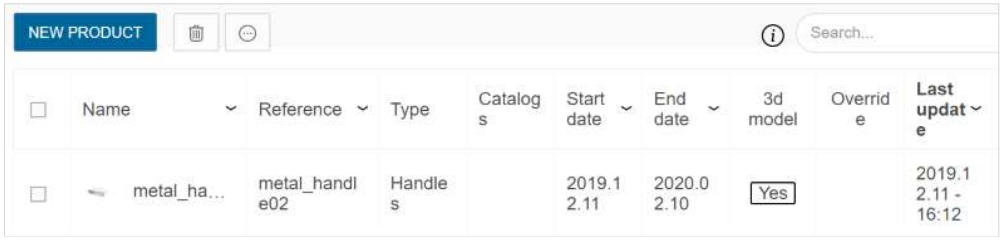
Now the **Variations** area contains the following:




Save

Click **Save** to upload the model and its datasheet in the 3DCloud storage. Click **Products** to see the new product in the list.

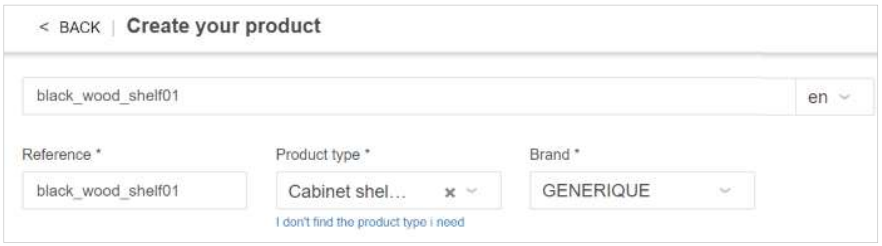
When the product is successfully added (a delay is possible), the **3D model** column displays **Yes** in the product list.




## Create the Shelf Datasheet

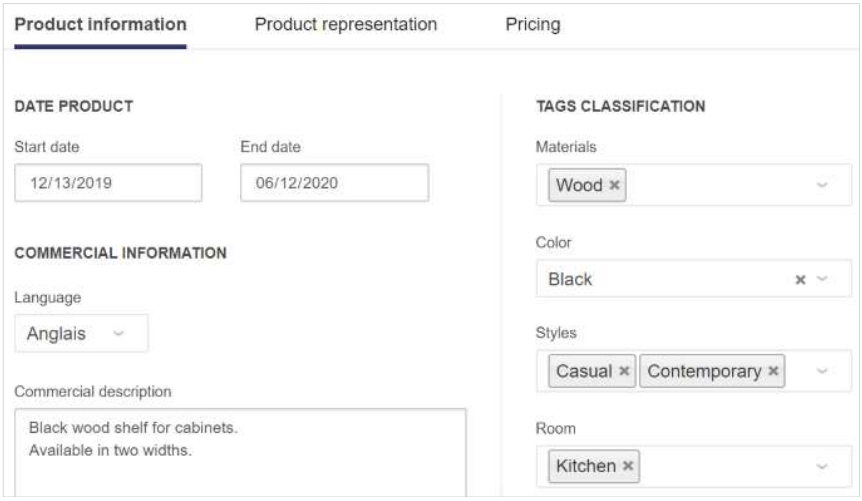
To create the datasheet for the shelf, follow the steps illustrated below.

## General Information




1. Enter the name of the product: select a language in the drop-down list and then, enter a name. Repeat this step for each language. Note that this name will be displayed to the customer.
2. Enter your reference for the product. This reference may be displayed to the customer and is used in Assembly Editor as well.
3. Link the product to a product type, i.e. **Cabinet shelfves** because this shelf belongs to a cabinet.
4. Link the product to your Brand.

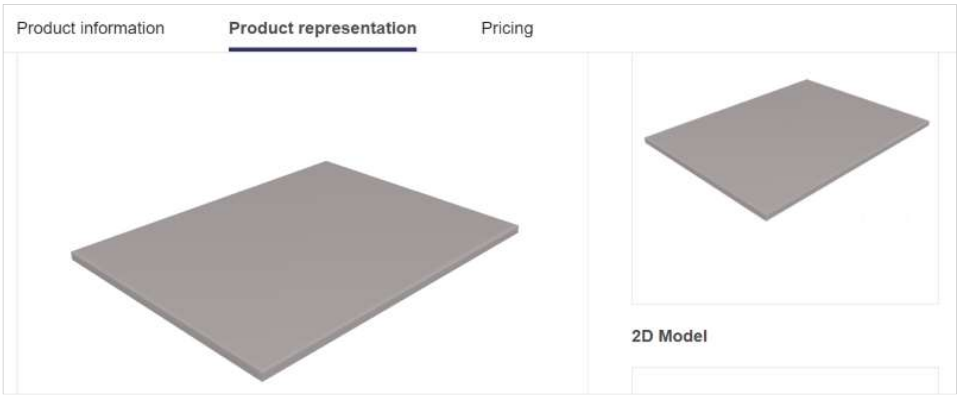
## Product Information




1. Define the period of availability, if need be. Note that these dates always overwrite the dates set in the catalog.
2. Choose a language to use in the Kitchen Planner.
3. Enter the commercial description and a short description of the box.
4. Define tags to ease the classification of the box in the Kitchen Planner.

## Product Representation

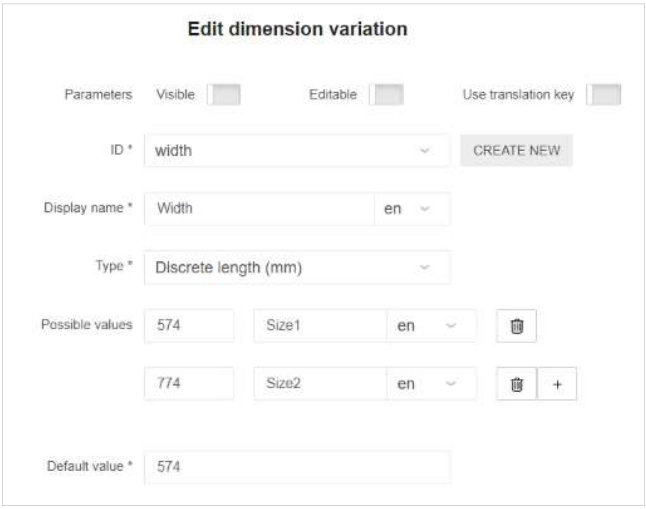
Start by adding the 3D and 2D representations of the product.




1. Add the BM3 file by clicking **Upload a 3D model**.
2. Add a bitmap picture of the box by clicking **Upload a picture**. It will be used as thumbnail in the product list in 3DCloud and in the Kitchen Planner.

### *Variable Dimension*

We defined a smart scaling on our shelf to enable its widening in the Kitchen Planner. Thus, specify the dimensions by clicking **Add variable dimension**.



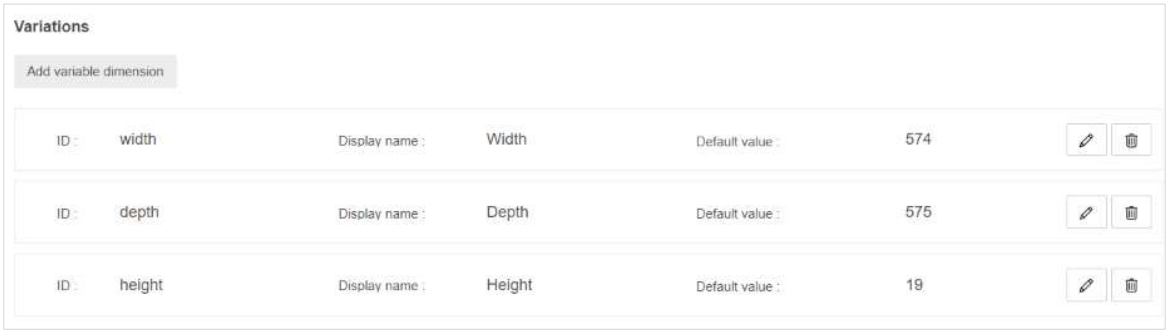

1. Select **width** in the **ID** drop-down list (or create it if need be).
2. Enter a label for the ID in the **Display name** field. Capitalize the first letter to avoid confusion with the ID.
3. Select **Discrete length (mm)** in the **Type** list. This will allow the customer to *choose* between different possible values, while the **Continuous length** would allow the customer to *enter* a value.
4. Enter the possible values in the corresponding fields, with a name to display to the customer. For our box, enter **574** and **774**.
5. Specify the default value, i.e. **574**.
6. Click **Validate** to save the variation.

The dimension variation is now defined and visible in the **Variations** area.

*Fixed Dimensions*

Repeat the proceeding to specify the height and the depth of the box with only one possible value.

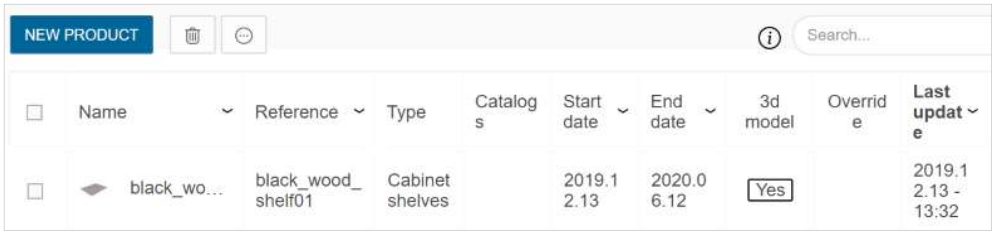
Now the **Variations** area contains the following:




Save

Click **Save** to upload the model and its datasheet in the 3DCloud storage. Click **Products** to see the new product in the list.

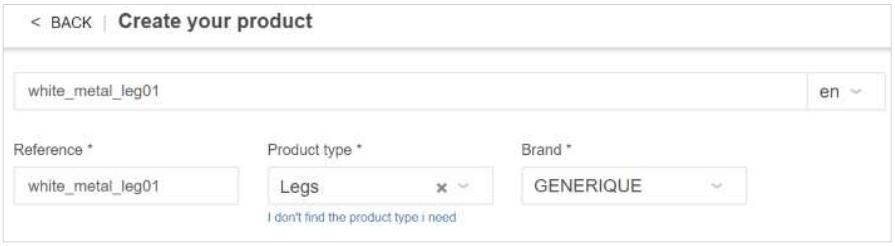
When the product is successfully added, the **3D model** column displays **Yes** in the product list.




## Create the Leg Datasheet

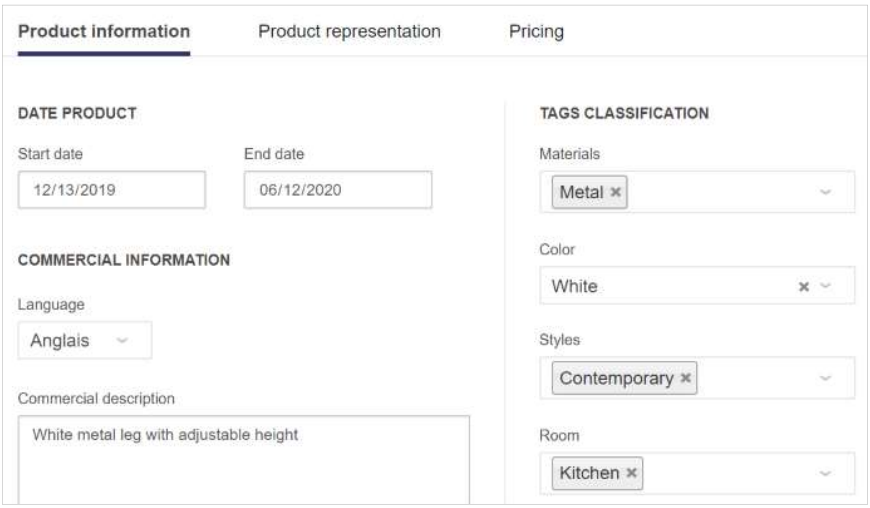
To create the datasheet for the leg, follow the steps illustrated below.

General Information



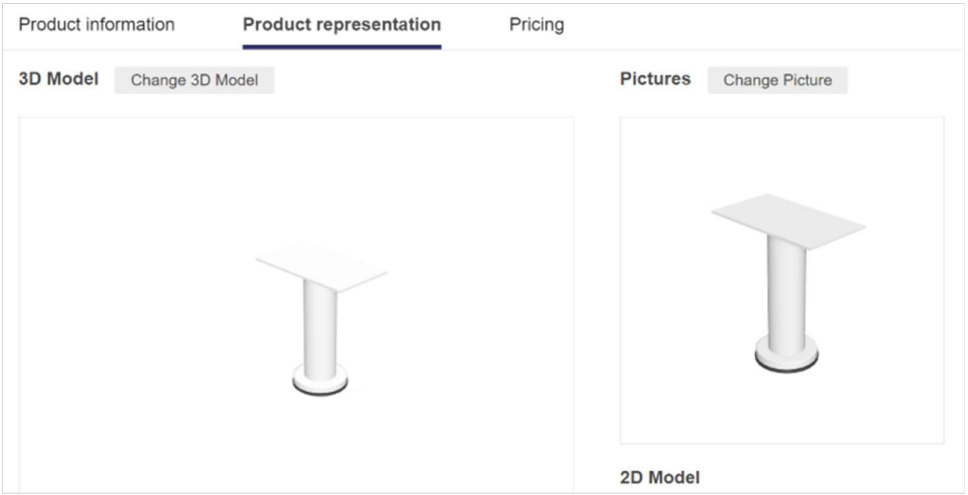

1. Enter the name of the product: select a language in the drop-down list and then, enter a name. Repeat this step for each language. Note that this name will be displayed to the customer.
2. Enter your reference for the product. This reference may be displayed to the customer and is used in Assembly Editor as well.
3. Link the product to a product type.
4. Link the product to your Brand.

## Product Information




1. Define the period of availability, if need be. Note that these dates always overwrite the dates set in the catalog.
2. Choose a language to use in the Kitchen Planner.
3. Enter the commercial description and a short description of the leg.

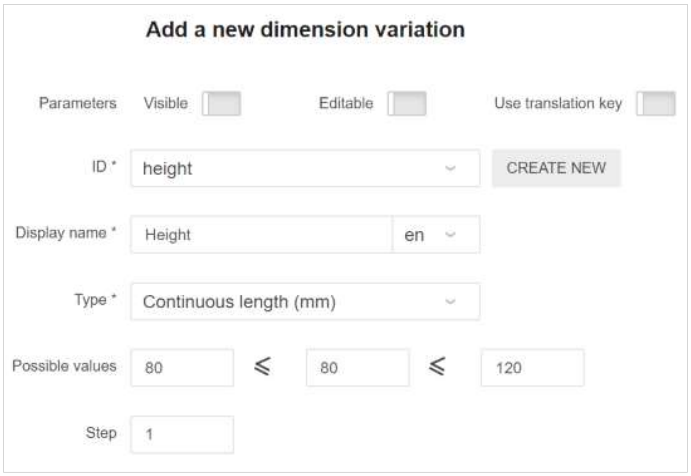
## Product Representation




1. Add the BM3 file by clicking **Upload a 3D model**.
2. Add a bitmap picture of the handle by clicking **Upload a picture**. It will be used as thumbnail in the product list in 3DCloud and in the Kitchen Planner.

### *Variable Dimension*

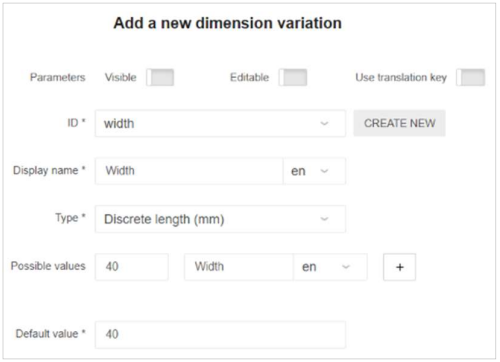
Click **Add variable dimension** to allow the customer change the height of the leg in the Kitchen Planner.

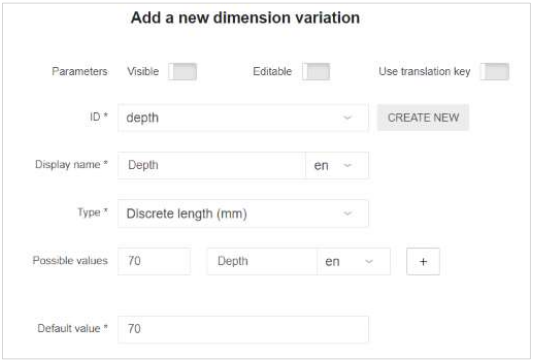



1. Select **height** in the **ID** drop-down list (or create it if need be).
2. Enter a label for the ID in the **Display name** field. Capitalize the first letter to avoid confusion with the ID.
3. Select **Continuous length (mm**) in the **Type** list. This will allow the customer to *enter* a value.
4. Enter the possible values in the corresponding fields, with a name to display to the customer. The value in the center is specified automatically with the height object.
5. Specify the default value, i.e. **80**.
6. Click **Validate**.

### *Fixed Dimensions*

Define the other dimensions of the leg by clicking **Add variable variations** again. This variation is used for any dimension specification, even if it will not vary.

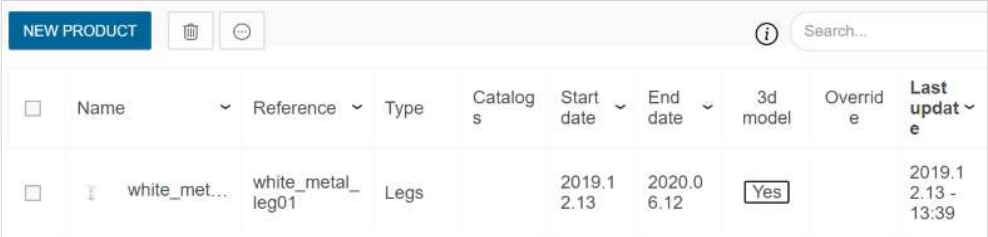



## Save

Click **Save** to upload the model and its datasheet in the 3DCloud storage. Click **Products** to see the new product in the list.

When the product is successfully added (a delay is possible), the **3D model** column displays **Yes** in the product list.

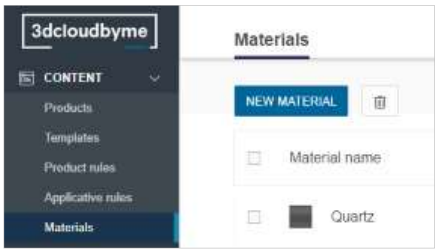



## Create the Worktop Material Datasheet

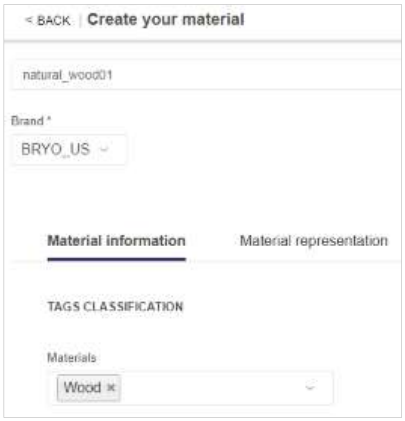
The worktop is a material (a .BM3MAT file) and needs to be recorded in a material datasheet.

The proceeding is similar to the creation of products with few information to specify.

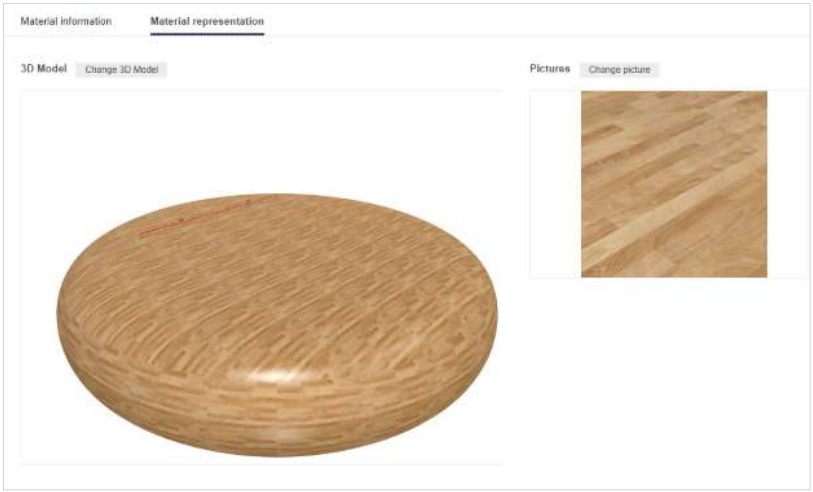
Click **Materials** in the **Content** menu and then, click **New material**.




1. Enter a name for the material.
2. Link the material to a brand. This is mandatory.
3. Specify a Materials tag.




Click **Material representation** to add the 3D model and the picture that will be used as thumbnail.



Click **Save** to save the material datasheet.

Note: Create as many worktop materials as the number of worktop products. Afterwards, create the corresponding product datasheets.

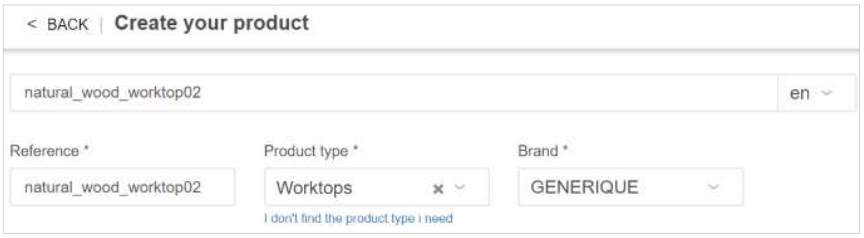
## Create the Worktop Datasheet

Worktops are generated dynamically in the Kitchen Planner from the worktop products available in the database, and from the settings made in 3DCloud and Assembly.

At this stage, you have to create several worktop products to create a choice of worktops for the customer.

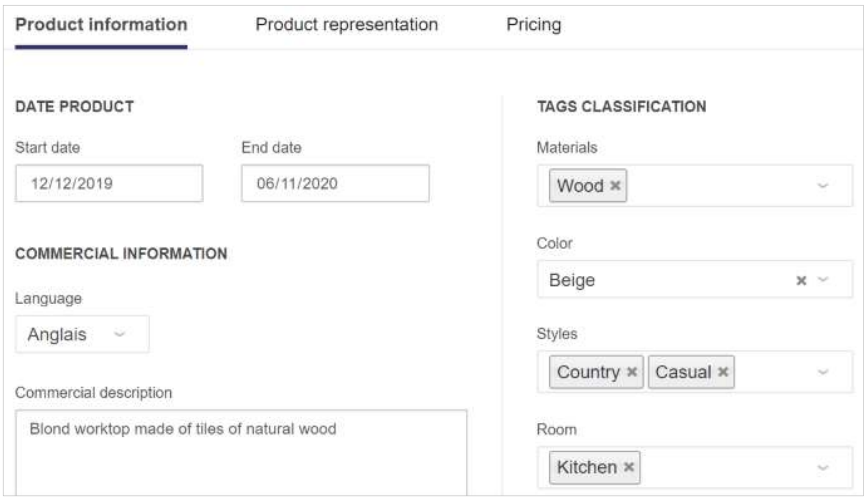
⇨ See “Step 5 / Prepare the Worktop” and “Step 6 / Worktop Option and Worktop Type” for detailed information about the worktop settings.

General Information




1. Enter the name of the product: select a language in the drop-down list and then, enter a name. Repeat this step for each language. Note that this name will be displayed to the customer.
2. Enter your reference for the product. This reference may be displayed to the customer and is used in Assembly Editor as well.
3. Link the product to a product type, i.e. **Worktops** because it is a worktop.
4. Link the product to your Brand.

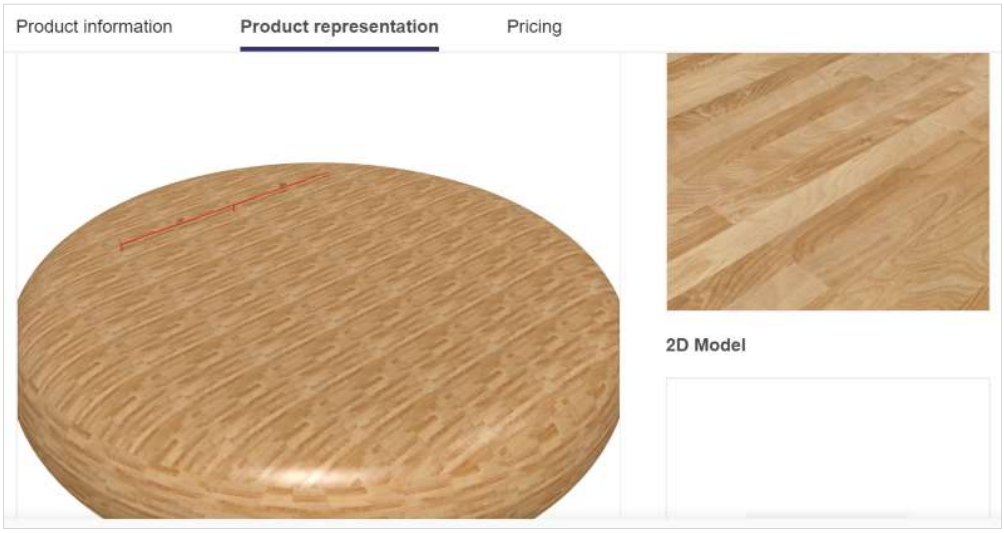
Product Information




1. Define the period of availability, if need be. Note that these dates always overwrite the dates set in the catalog.
2. Choose a language to use in the Kitchen Planner.

3. Enter the commercial description and a short description of the worktop.

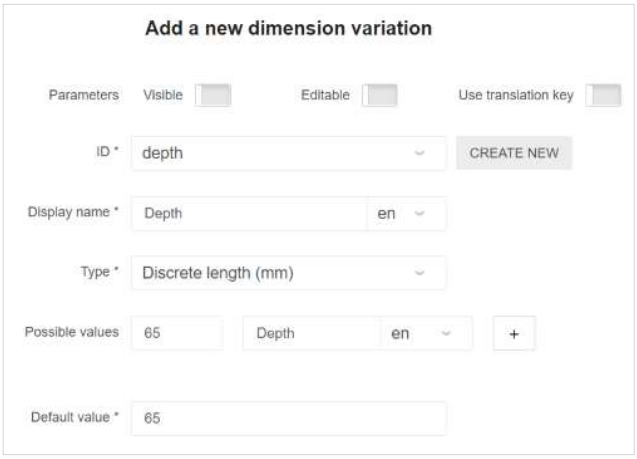
Product Representation




1. Add the BM3MAT file by clicking **Upload a 3D model**.
2. Add a bitmap picture of the worktop by clicking **Upload a picture**. It will be used as thumbnail in the product list in 3DCloud and in the Kitchen Planner.

### *Fixed Dimensions*

Click **Add variable dimension** to define the depth of the worktop. Our worktop is a precut one, and only the width of our sample worktop is stretchable to be consistent with the stretching of the cabinet.

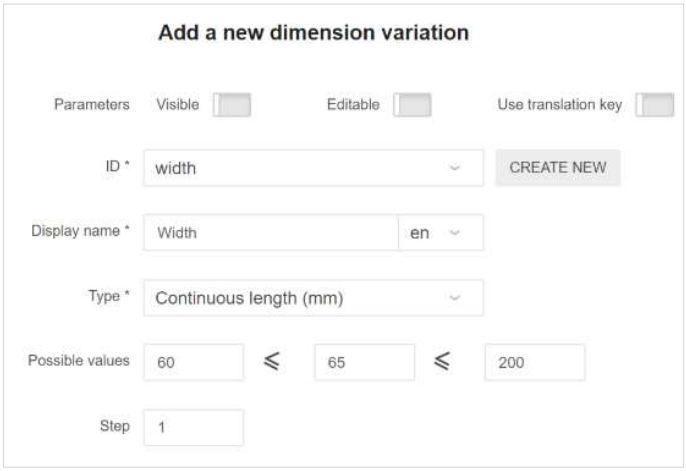



1. Select **depth** in the **ID** drop-down list (or create it if need be).
2. Enter a label for the ID in the **Display name** field. Capitalize the first letter to avoid confusion with the ID.
3. Select **Discrete length (mm**) in the **Type** list. This will allow you to enter the dimension.
4. Enter **65** and **Depth** as unique possible value.
5. Enter **65** as default value.
6. Click **Validate** to add the dimension.

7. Repeat the proceeding to add the height (40 mm) of the worktop.

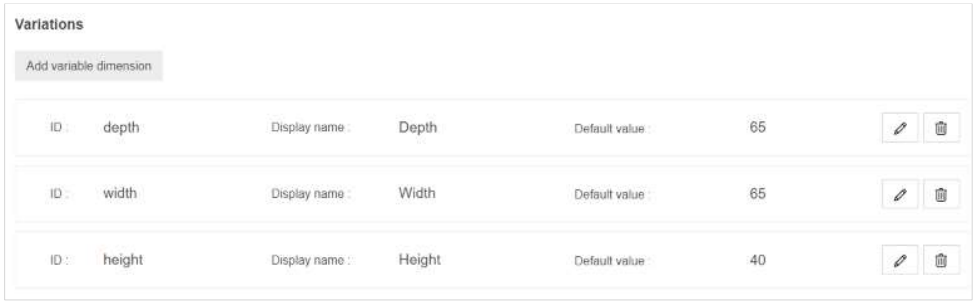
### *Variable Dimension*

Now click **Add variable dimension** to define the width.



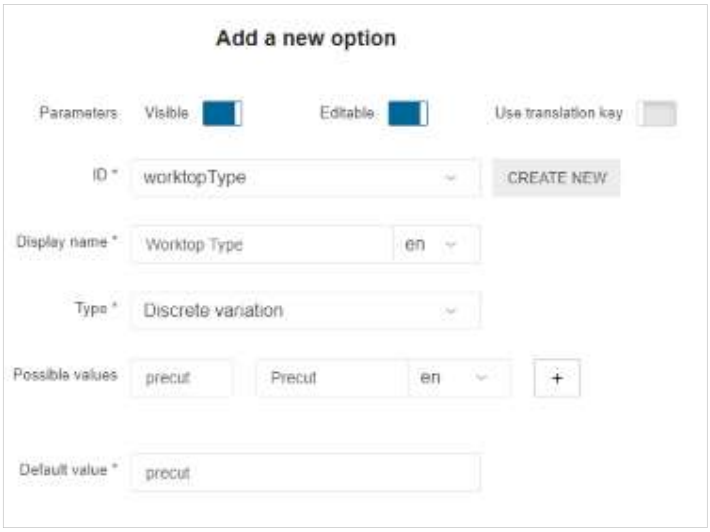

1. Select **width** in the **ID** drop-down list (or create it if need be).
2. Enter a label for the ID in the **Display name** field. Capitalize the first letter to avoid confusion with the ID.
3. Select **Continuous length (mm**) in the **Type** list. This allows you to enter a range of possible values.
4. Enter **65** as default value.
5. Click **Validate** to add the dimension variation.

Now the **Variations** area shows the following:




### *worktopType*

This option will determine whether the worktop is sold with predefined dimensions (“precut”), or at the meter (“custom”). Click **Add option**.




1. Check the **Visible** and **Editable** options to make the worktop editable by the customer in the Kitchen Planner.
2. Click **Create new** to specify **worktopType** as the ID and click **+** to move it to the **ID** field.
3. Enter a label for the ID in the **Display name** field. Capitalize the first letter to avoid confusion with the ID.
4. Select **Discrete variation** in the **Type** drop-down list. This will allow you to enter a value.
5. Enter **precut** and **Precut** as unique possible value.
6. Specify the default value, i.e. **precut** for our sample worktop which width will be variable.
7. Click **Validate** to add the option.

## Pricing

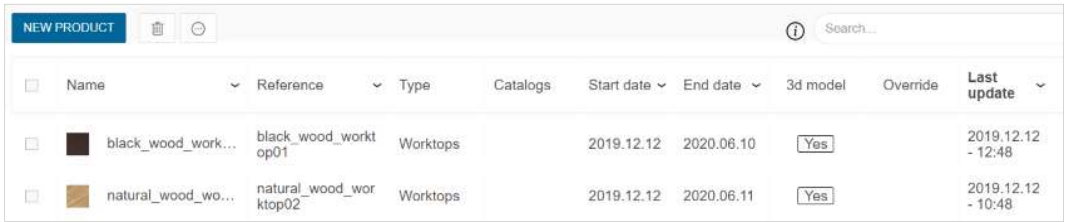
Worktops require a pricing mapping, which is not defined in this datasheet.

⇨ Refer to the “Content Publisher Guide / Pricing” for detailed information about the pricing.

## Save

Click **Save** to upload the model and its datasheet in the 3DCloud storage. Click **Products** to see the new product in the list.

When the product is successfully added (a delay is possible), the **3D model** column displays **Yes** in the product list.



## STEP 4 – ADD THE PRODUCTS TO A CATALOG

When the product datasheets are created, you have to link them to a catalog unless they will not appear in Assembly Editor.

This means that you can add as components in Assembly Editor only the products that are registered in catalogs.

Product catalogs are a required link between a product and the Kitchen Planner. This catalog must belong to an application distribution to be visible in the Kitchen Planner. Below is an illustration of these relationships.

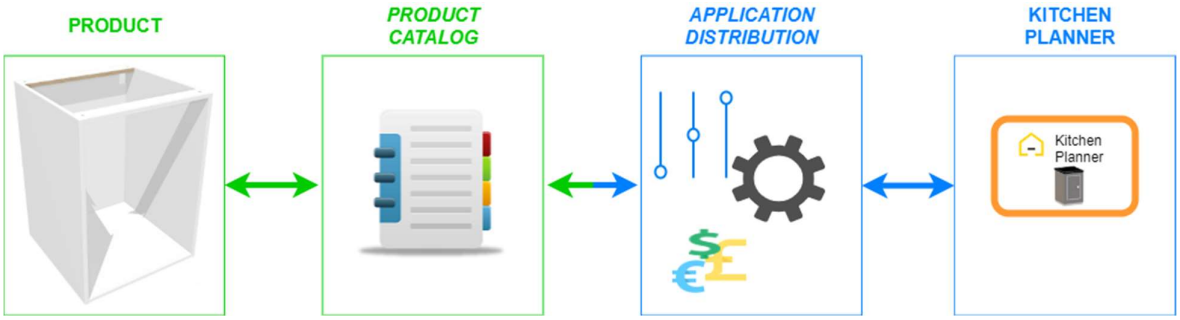
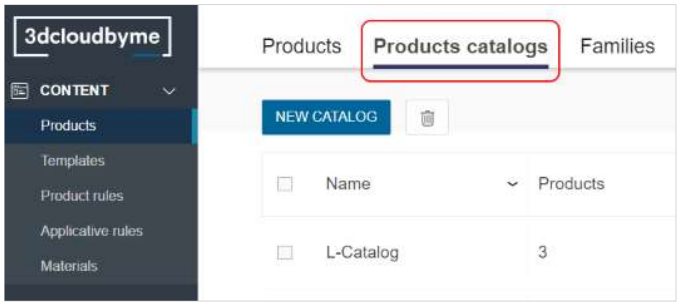



*Figure 7 – Product and catalog relationship*

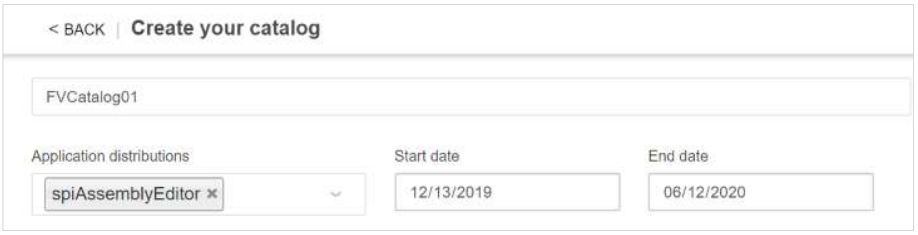
We will create a unique catalog to contain all the products of our sample cabinet. However, the best practice is to add only top-assemblies to the catalog.

Click the **Products catalogs** tab and then, **New catalog** to start the proceeding.




### General Information

⇨ See “Core Platform Concepts” in the Reference documentation for a definition of “Application distribution”.



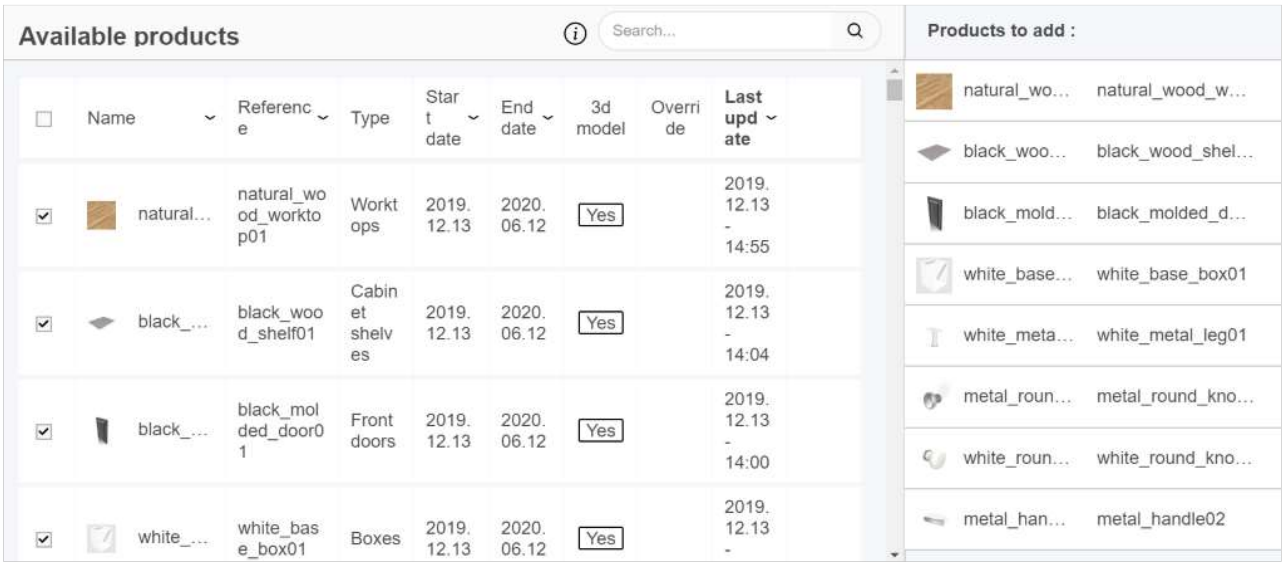

1. Give a name to the catalog. This is an internal name; it will not be displayed to the customer.
2. Select at least one **Application distribution**.
3. Specify the period of viability of the catalog. Note that product dates overwrite catalog dates.

## Catalog Information

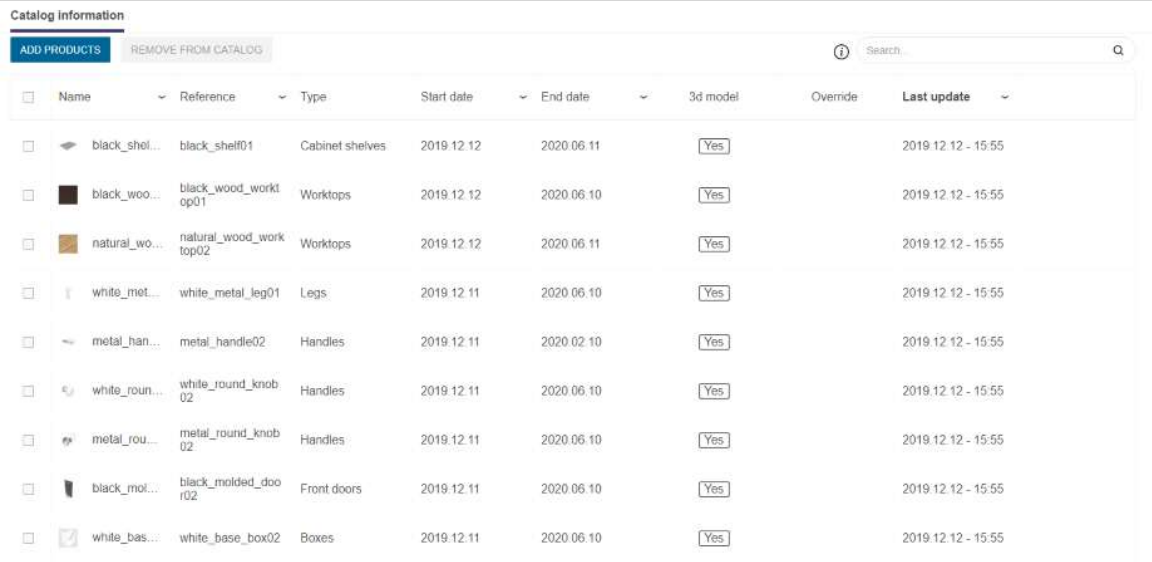
Products are added to the catalog from this page. Even if the catalog is created, product cannot be linked to a catalog from the product pages.

Click **Add products** to select the products to link to the catalog. You can add several products at a time.

For our sample cabinet, select all the products created at step 4.



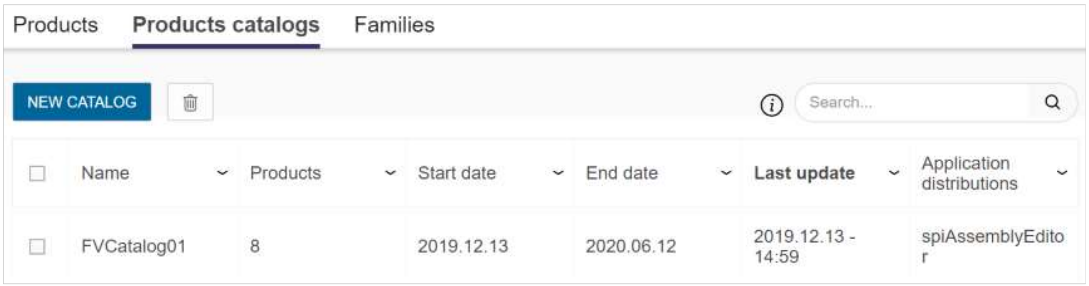

Click **Validate** to add the product to the catalog. They are now listed in a table in the **Catalog information** area.




To remove a product from the catalog, select the corresponding check box and then, click **Remove from catalog**.

## Save

Click **Save** to upload the catalog and its content in the 3DCloud storage. Click **Products** and then, **Products catalogs** to see the new product in the list (a delay is possible).



# STEP 5 – ASSEMBLE THE PRODUCTS INTO A CABINET

The aim of Assembly Editor is to group together the components (3D products and non-3D items, with their parameters) into one finished product that can be displayed to the customer, while taking their relative placement and orientation into account. This finished product will be offered to the customer in the Kitchen Planner with a 3D experience that will allow her/him to manipulate the product, zoom in and out, try various combinations, and eventually edit the product to match it to her/his wishes.

The native format of Assembly Editor is .BMA. A .BMA file is a list of components that are placed according to a position mapping.

Assembly Editor also allows you to add kinematics (such as opening a door in the Kitchen Planner), manage component positioning (absolute or relative), manage component publication, manage output sets (for cut-out or worktop generation) and manage anchor points (possible positions of components in editable furniture).

Note: Assembly Editor allows you to work with .BM3 and .BMA files. The condition is that they have a datasheet in 3DCloud.

## Vocabulary and Concepts

The aggregation of the components is named “assembly” hereinafter.

“Components” are products registered in 3DCloud.

The properties of each component (position, orientation, own parameters, etc.) can be linked (or "binded") to parameters and/or relations of the assembly: that is why these assemblies are named “parametric assemblies”.

About Parameters

There are many parameters in Assembly Editor: they are the duplication of all the parameters defined in the 3DCloud datasheets. There are component parameters, size parameters, material parameters, etc. each parameter having a different type. A parameter can be a dimension (mm), a Boolean, an integer, a material, or a component.

Some parameters may be visible or editable by the customer in the Kitchen Planner, or by the Range Manager in 3DCloud.

About Relations

The default dimensions, for example, are recovered from the .BM3 file. They can be overloaded by dynamic values obtained through relations.

A relation make it possible to define calculated positioning of components – instead of fixed, absolute positioning. For example, legs are always positioned under the cabinet with the given offset; the relation can determine the position of the legs, whatever the width of the cabinet is.

Relations are expressions used to calculate dynamic values from parameters or other relations. A relation supports mathematic operators (such as **+**, **-** and *****) or logical tests (using a dot as separator). The result can be assigned to component properties, output set properties and anchor point properties. Relations

are defined internally and cannot be overloaded, whether in 3DCloud by a Range Manager or in the Kitchen Planner by a user, to avoid structure problems.

⇨ Refer to the "Product and Assembly reference" documentation, section "Assemblies" for detailed information on the relations.

For our sample, we will define many relations to position the legs, the handle or the door front dynamically.

About Component Positioning

The positioning of the components can be of three types:

- Absolute, relating to the axes;
- Relative, via expressions.

⇨ Refer to the "Assembly Editor Reference" documentation for a complete information on anchor points.

About Kinematics

You can define in Assembly Editor the kinematics that can be played in the Kitchen Planner. These kinematics behaviors are either of rotation or of translation type.

Important: The scope of the animation cannot be on a single component of the assembly to which you want to add a kinematics, because it will apply on all the components. Therefore, you have to create the kinematics on the right level.

In our case, we want the door (i.e. front with the fixed handle) to rotate: we will put the kinematics on the sub-assembly containing the two components front and handle.

About Output Sets

An output set is a linear information required in specific cases to handle application features requiring 2D as an input. Specific cases are every linear products: worktop, plinths, cornices…

For our sample, we will define output sets to place the worktop on the base cabinet.

⇨ Refer to the "Assembly Editor Reference" documentation for a complete information on output sets.

About Sub- and Top-Assemblies

Sub-assemblies are a way to ease the work and make common configurations reusable. Each sub-assembly is then registered as a product and linked to a catalog in 3DCloud in order to be used either as such or as a component in a top-assembly.

For example, it is highly recommended to create a "legs and box" sub-assembly or a "front and handle" sub-assembly.

A top-assembly is the final assembly built from simple components and sub-assembly components. The top-assembly is then registered as a product and linked to a catalog in 3DCloud.

## Data Model

Below is an overview of the data model, where components are grouped into sub-assemblies before creating the final assembly.

We recommend creating sub-assemblies of the door front with the handle, and of the box with the legs. Then, add them to the top-assembly.

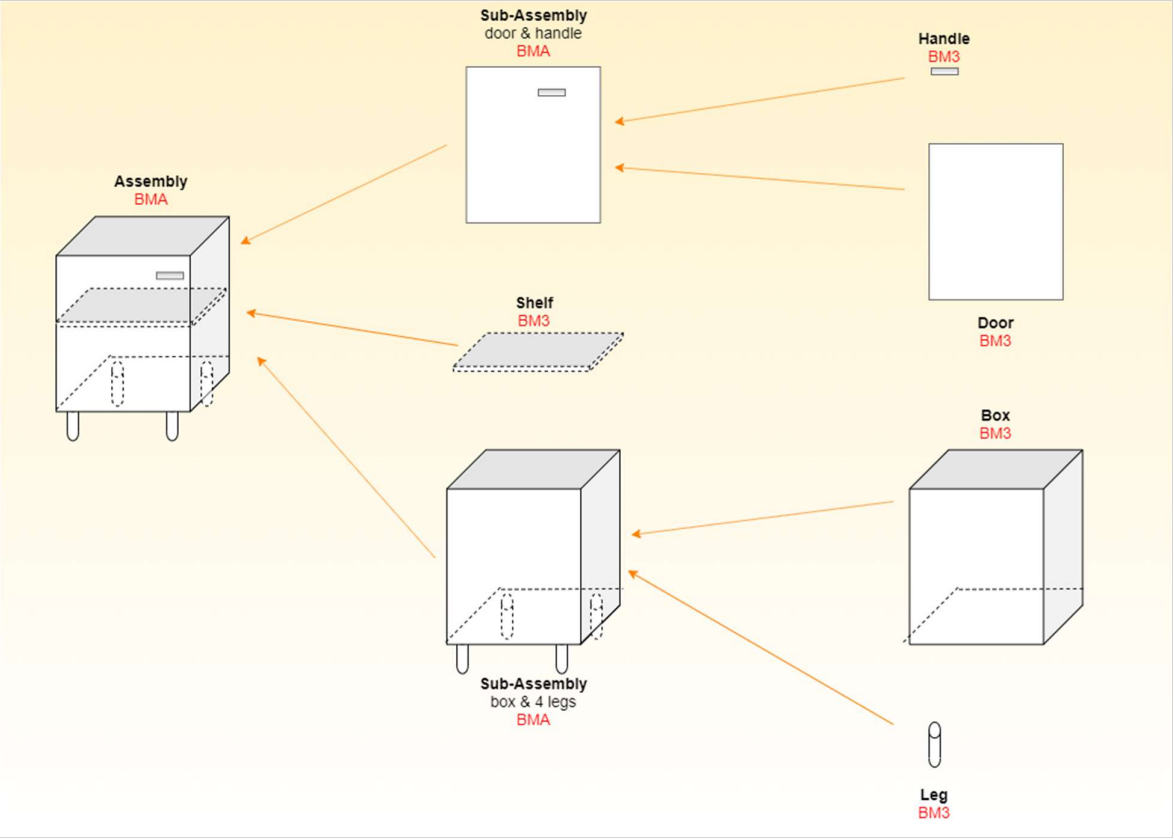
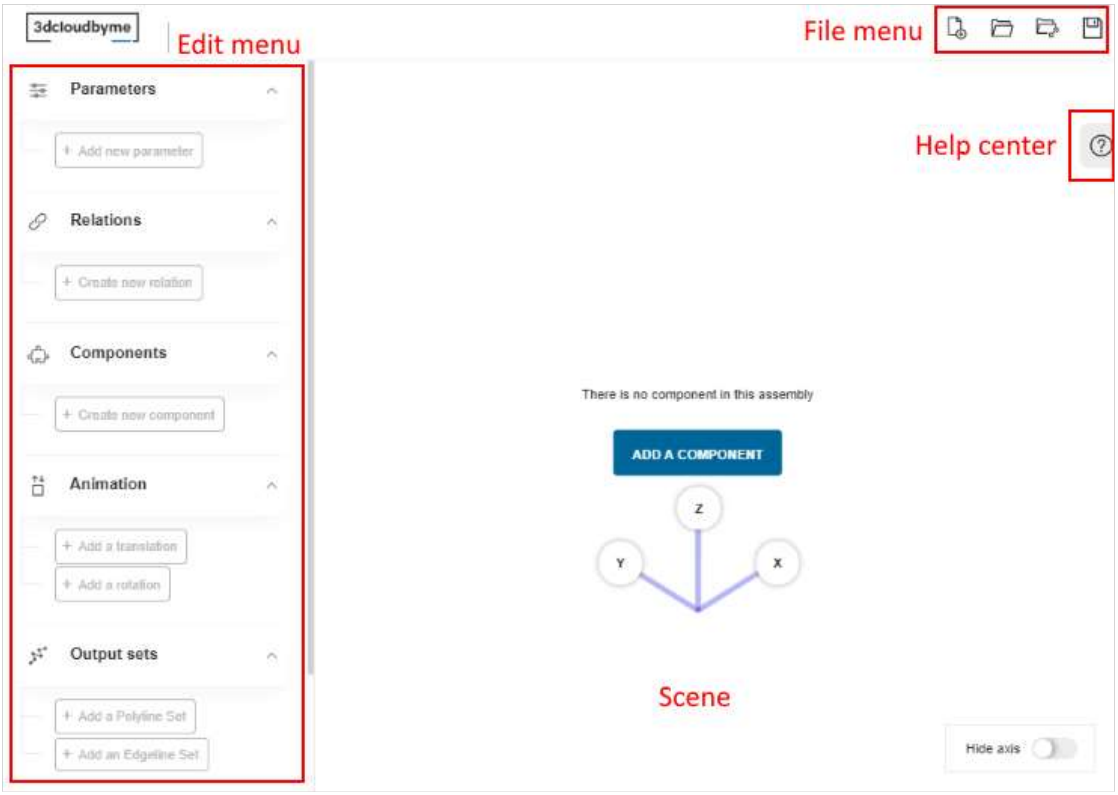



*Figure 8 – Assembly and sub-assemblies*

## Quick Overview of the Interface

⇨ Refer to the "Assembly Editor Reference" documentation for detailed information on the User Interface.




The workspace is organized into three areas.

- The **File** menu, from where you can create, open and save an assembly.
- The **Edit** menu, where you create the components, parameters, relations etc. for the assembly.
- The **Scene**, where you can view and manipulate the assembly.
- The **Help center**, which checks the validity of the assembly.

The **File** menu contain the following functions:

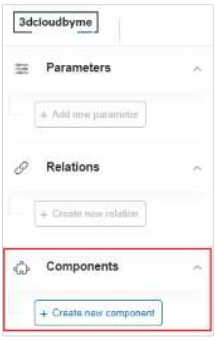
| | | |
|---|---|---|
| | **New assembly** | Click this icon to create a new assembly.<br>This will close the current assembly, if applicable. |
| | **Open from disk** | Click this icon to open an assembly from your disk.<br>This will close the current assembly, if applicable. |
| | **Open from URL** | Click this icon to open an assembly from an URL.<br>This will close the current assembly, if applicable. |
| | **Save** | Click this icon to save the current assembly, even if it is not finished.<br>This will create a zip file containing a root.BMA file, to save on your disk. |

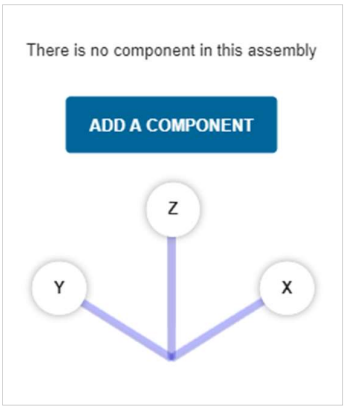
## Create the "Box and Legs" Sub-Assembly

When starting a new assembly, the left menu is empty, as illustrated.

## Add the Components

Click **Create new component** on this menu, or click **Add a component** on the scene.

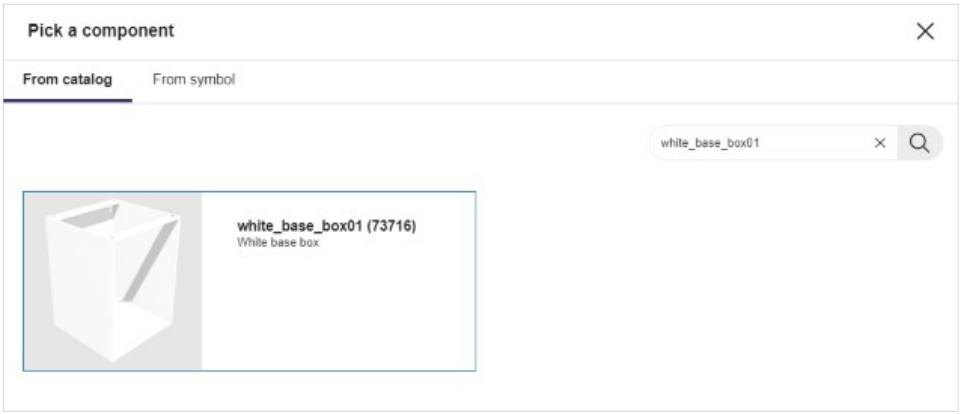


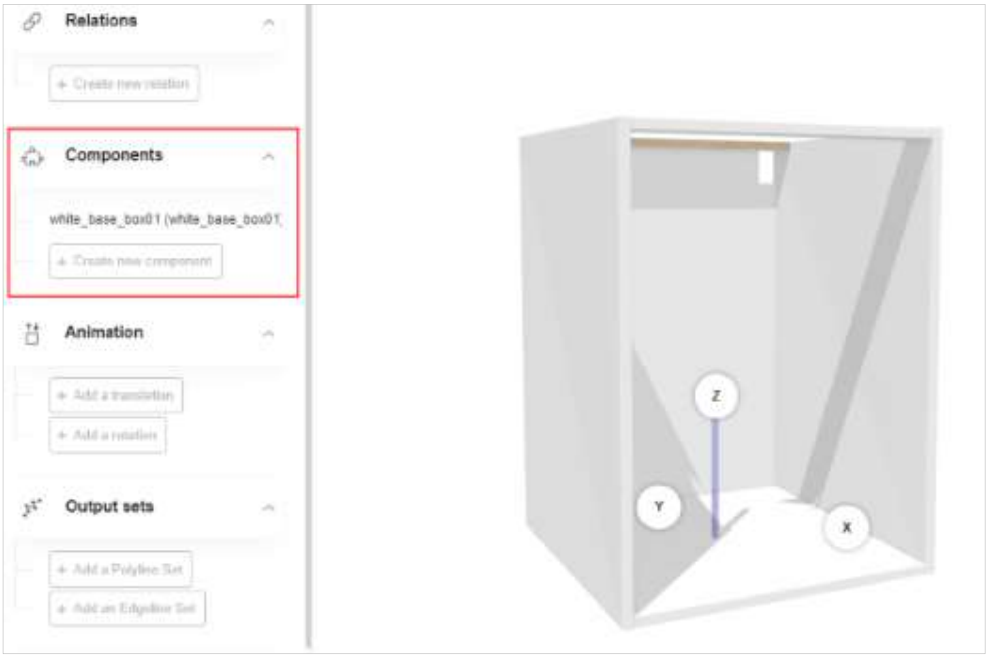

Browse the catalog by specifying the name of the product in the search field. Use either the complete name as defined in the 3DCloud datasheet or the type e.g. “box” or “front”. If you want to enter only the first letters of the name, add an asterisk to the name*. For example: “white_*”.

Press **Enter** to start the search.

Important: If you do not find your product, check that it is linked to the catalog.

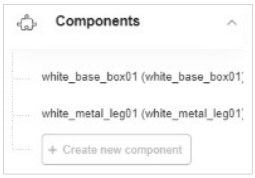



Click the thumbnail of the product to add it to the list of components and display it on the scene.

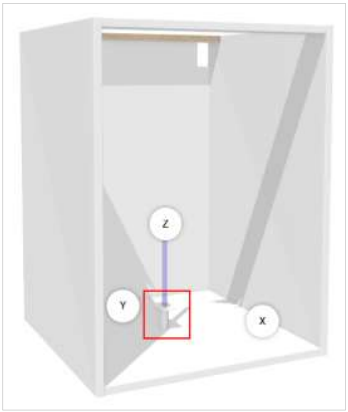



Repeat the proceeding to add the leg.

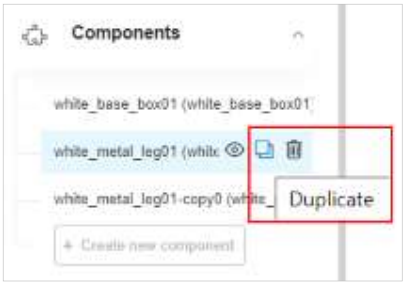
You can show or hide a component on the scene by clicking the eye icon next to the component name.




Note that all the components are centered by default on the origin of the axes. Their final position is not yet defined.



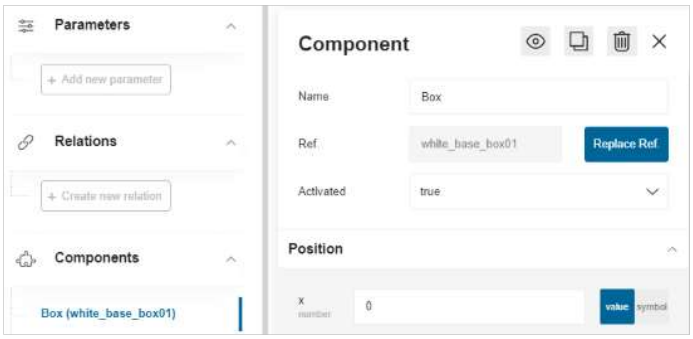

Duplicate the leg three times by clicking the **Duplicate** icon, to obtain a set of four legs. At this stage, they receive a default name.



## Rename the Components

Select a component in the list or on the scene to display its properties.

This area allows you among others to rename the component and define its position.




Rename the box component into “Box”.

Important: Always capitalize the first letters of the words in a compound name to avoid confusion with parameter names. For example, a component name would be “CoverPanel” and a parameter name would be “coverPanel”.

Rename each leg according to the diagram below:

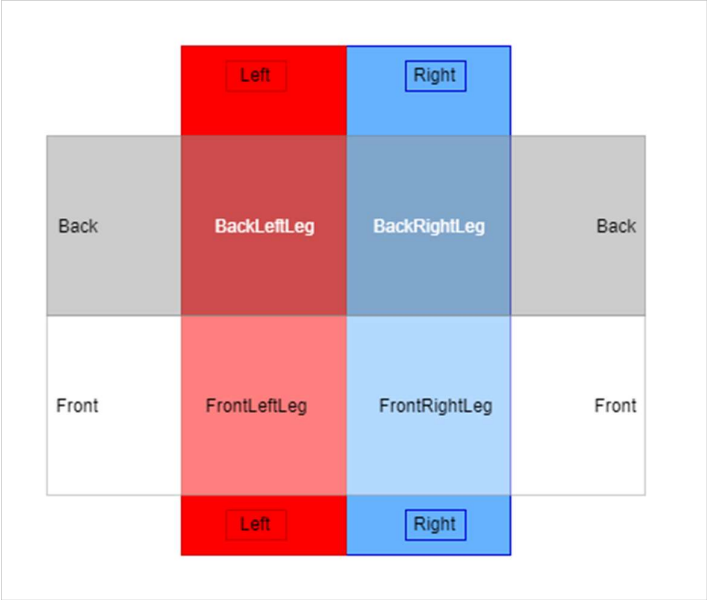
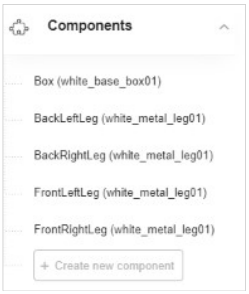



*Figure 9 – Rename the legs*

Check that the components of the "box and legs" sub-assembly are all renamed.




## Create Parameters

We highly recommend that you create parameters as things progress; avoid creating all possible parameters "once for all" to prevent confusions.

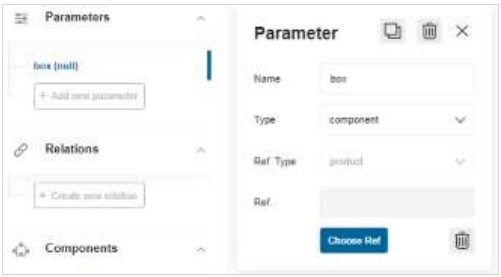
### *Create the Component Parameters*

Important: Use camel case convention when renaming the parameters. Try to create meaningful names, as in our examples. Parameters always starts with a lowercase.

You have to create one parameter per component. Duplicated components are considered as a whole. For example, you have to create *one* parameter for *the set of four* legs.

To create a parameter, proceed as follows:

1. Click **Add new parameter** in the **Parameters** area.
2. Click the parameter to display its properties.
3. Replace the default name by one matching the naming convention, e.g. "box".
4. Select **component** in the **Type** drop-down list. The associated reference type is specified automatically.
5. Changes are saved automatically. Click the cross icon to close the **Parameter** panel.

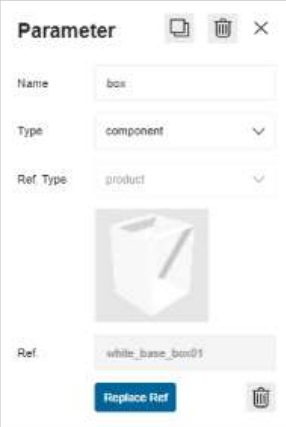


For the "box and legs" sub-assembly of our sample cabinet, create the following parameters: "box" and "leg".

### *Link the Parameters with the Components*

The next step consists in linking each parameter to its component.

1. Click the first parameter e.g. **box** to access its properties.
2. Click **Choose ref** to select a product reference to link with the parameter.
3. Search for the product using its 3DCloud name. Then, click the product thumbnail to add it as reference.




4. Repeat the proceeding to link the leg parameter with the leg product.

Now the component reference appears after the name, in parentheses, instead of “null”.

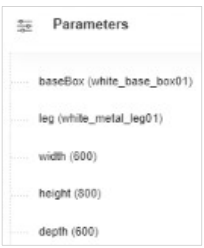
### *Create the Size Parameters*

Important: These size parameters are mandatory.

The next step consists in creating the size parameters of the box that will be useful for the sub-assembly.

1. Click **Add new parameter** in the **Parameters** area.
2. Click the parameter default name to access its properties.
3. Rename the parameter into **width**.
4. Select **number** in the **Type** drop-down list.
5. Enter a value, i.e. 600 for the box.
6. Repeat the proceeding with the height (800) and the depth (600).

The **Parameters** area now contains the following parameters:




### *Link the Size Parameters to the Box*

This step consists in linking the size parameters with the dimensions of the components.

The box must be linked dynamically to the size parameters to prepare the scaling of the base cabinet.

Click the **Box** component in the list to consult its properties. Then, scroll down the panel to reach the **Overloads** field. The size parameters displayed in this area are recovered from the 3DCloud datasheet.

1. Select the check box regarding the **depth** of the component to enable the modifications.

2. Move the cursor to **Symbol** to display the list of parameters declared in Assembly.
3. Select the **depth** parameter. The depth parameter of the "Box" product is now overloaded by the depth parameter of the "box" component.

Repeat the proceeding with the width and height of the box. The **Overloads** area now indicates that the three parameters have been overloaded.

## Define the Position of the Box in the Room

Each component is placed relating to the absolute coordinate system (origin and axis).

Click the box in the **Components** to display its properties. Go to the **Position** area to access the coordinates.

Keep **0** in the **X** and **Y** fields and enter **80** in the **Z** field. This value corresponds to the height of the leg. The box is now 80 mm higher from the floor.

## Place the Legs

To position the legs under the furniture you have to consider the following:

- The size of the box;
- The positioning of the front legs on the box (may depend on whether the box will have a plinth or not);
- The positioning of the back legs on the box (the distance may not be the same as for the front legs);

Therefore, the position of the legs is managed through relations.

In our sample case, there is no plinth and the distance is the same for all the legs.

### *Create Leg Offset Parameters*

Before creating the relations, you have to create offset parameters to specify the position of each leg from the edge of the bottom of the bow. Then, these parameters will be used by the relations.

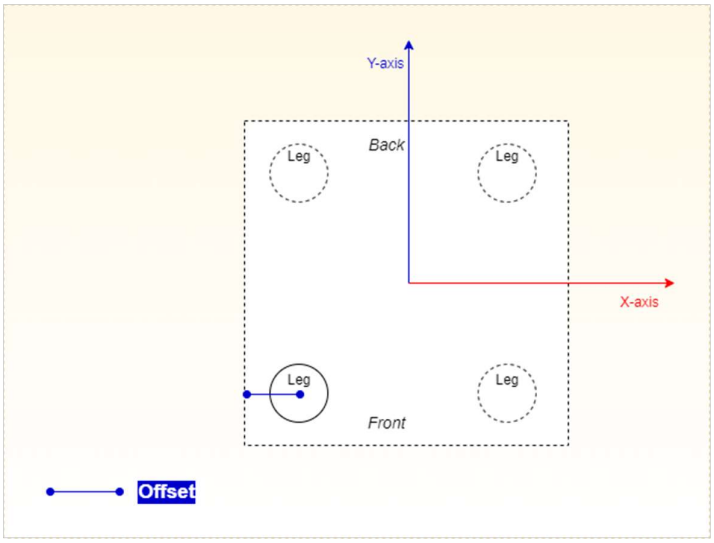



*Figure 10 – Leg offset*

As illustrated above, the offset is the distance between the center of a leg and the edge of the box.

Note: Remember that our sample 3D objects are centered on the origin point.

There are four legs, thus you have to define four offset parameters:

- xFrontLegOffset: the offset of the front leg on the X-axis;
- xBackLegOffset: the offset of the back leg on the X-axis;
- yFrontLegOffset: the offset of the front leg on the Y-axis;
- yBackLegOffset: the offset of the back leg on the Y-axis;

To create these parameters, proceed as follows:

1. Click **Add new parameter** in the **Parameters** area.
2. Click the default name of the new parameter to display its properties.
3. Replace the default name with **xFrontLegOffset**.

4. Select **number** in the **Type** drop-down list.
5. Enter a value, e.g. 100 if you want the center of the leg to be place 100 mm from the left edge of the box.
6. Repeat the proceeding to create the three other parameters for the three other legs.

Now you have the following four leg parameters that you can use in relations.

### *Create Offset Relations*

Note: Use the camel convention to rename relations; do not capitalize the first letter to avoid confusion with the components.

To create the relations, you do not have to consider the sides (front / back) but the axes (X and Y). Remember that the offset parameters are defined on these axes.

Thus, you will create four relations:

- Two relations to place the leg on the X-axis
- Two relations to place the leg on the Y-axis

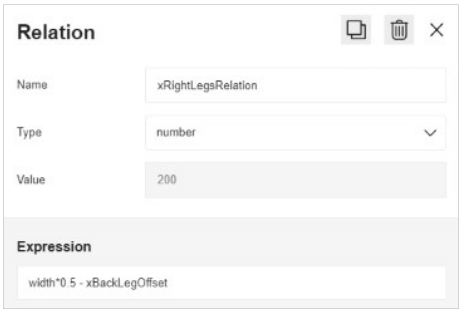
These relations will get a value calculated dynamically through a formula. This is important to ensure that the legs are aligned on a given axis.

Click **Create new relation** on the left menu to start creating the relations.

1. Select the default name of the relation to display its properties and start defining the relation.
2. Change the default name with an appropriate one: **xLeftLegsRelation**.
3. Select **number** in the **Type** drop-down list.
4. Specify the **Expression** field with a formula referring to the main width parameter and specifying the distance in millimeters from the edge of the box: **-width*0.5 + xFrontLegOffset**.
5. Press **Enter** to save the expression and update the **Value** field.

Repeat the proceeding to create the second relation for the X-axis.

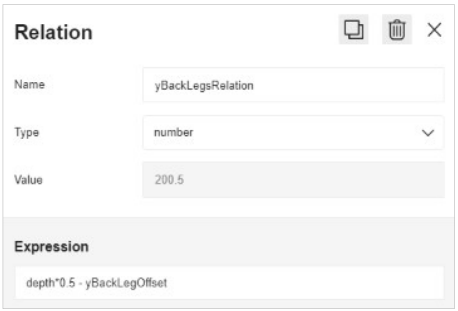
1. Name the relation **xRightLegRelation**.
2. Enter **width*0.5 – xBackLegOffset** in the **Expression** field.




Create now the relations on the **Y**-axis with the same proceeding. Note that the expression uses the **depth** parameter instead of the width, to be consistent with the Y-axis. Define them as follows:

**yBackLegsRelation**

Use yBackLegOffset in the expression:




**yFrontLegsRelation**

Use yFrontLegOffset in the expression:

### *Define the Position of each Leg*

Now that the relations between the legs on a same axis and the box are created, you can specify the XYZ position of each feet from the floor.

Click the **BackLeftLeg** component in the **Components** list to display its properties. Go to the **Position** area.

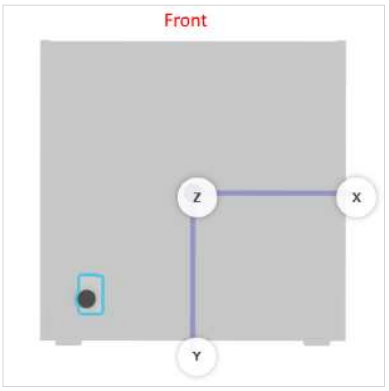
Note: For the case you would define a fixed position for the legs, remember to take both the width of the box and the offsets into account. In addition, remember that the components should be distributed *from* the origin point.

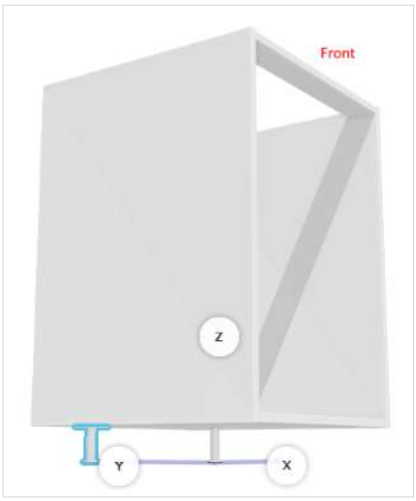
With the perspective of the future scalability of the cabinet, we will define dynamic positions of the legs, using the relations we have created previously.

For our sample cabinet, which width and depth are 600 and 600 respectively, enter the following:

- **X** position: Move the cursor to **Symbol** and select the **xLeftLegsRelation**.
- **Y** position: Move the cursor to **Symbol** and select the **yBackLegsRelation**.
- **Z** position: Enter **0** (legs are on the floor).

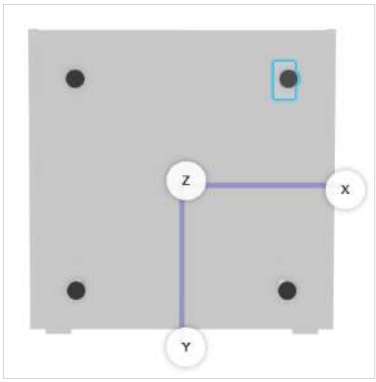
The back left leg is now placed as illustrated below:

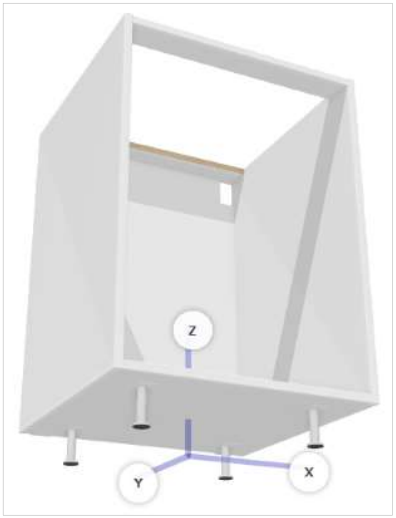



Repeat the proceeding to place the three other legs and obtain the following result.

| | Back Left Leg | Back Right Leg | Front Left Leg | Front Right Leg |
|---|---|---|---|---|
| **X position** | xLeftLegsRelation | xRightLegsRelation | xLeftLegsRelation | xRightLegsRelation |
| **Y position** | yBackLegsRelation | yBackLegsRelation | yFrontLegsRelation | yFrontLegsRelation |
| **Z position** | 0 | 0 | 0 | 0 |




## Prepare the Worktop

The availability of the worktop is ensured by the combination of the worktop option and output sets. Should one of these be missing, the worktop could not be generated in the Kitchen Planner.

Important: Do not add the worktop as material or component in Assembly Editor, this is useless. The worktop is added dynamically to the base cabinet in the Kitchen Planner only. See below “Step 7 – Display the cabinet in the Kitchen Planner”.

### *Worktop Option*

The worktop option is set in the top-assembly datasheet.

⇨ See “Step 6 / worktopOption” for the proceeding.

### *About Output Sets*

An output set is a linear information required to place the worktop on the base cabinet. In the Kitchen Planner, output sets are used to define the 2D properties of the worktop (and plinths, wall panels, decorative strips, cornices paths as well).

Output sets are of two types:

- A polyline set, which is limited to lines defined by points only;
- An edgeline set, which allow lines and circles defined by points and radiuses.

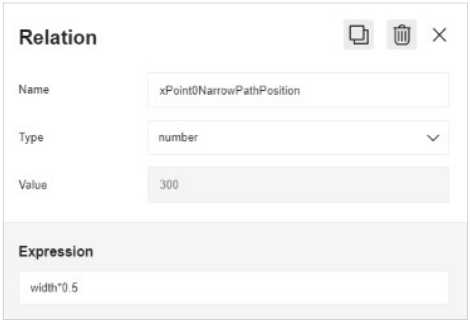
To define the position of our worktop, we will use polyline sets.

### *Create the Relations*

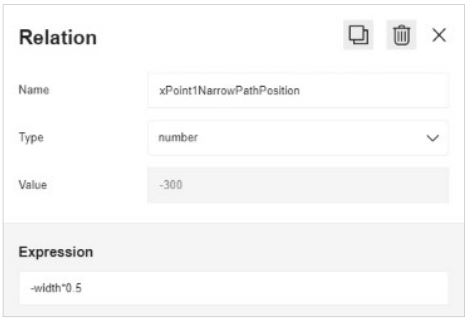
Polylines have either fixed coordinates, or variable coordinates using relations, in the perspective of a scaling.

Place the starting point of the polyline, through a first relation.

1. Click **Create new relation**.
2. Click the default name to display the properties.
3. Rename the relation into **xPoint0NarrowPathPosition**.
4. Select number in the **Type** drop-down list.
5. Go to the **Expression** field.
6. Enter a formula referring to the width parameter **width*0.5** as illustrated below. Press **Enter** to save the expression and update the **Value** field above.



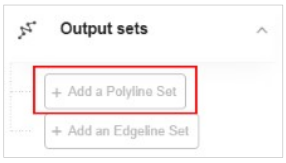

Then, place the destination point through a second relation named **xPoint1NarrowPathPosition**, which formula is **-width*0.5**.




### *Create the Output Set*

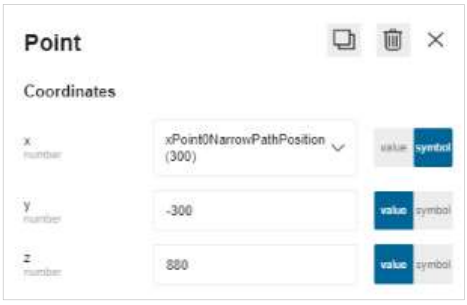
The points are created through the relations; you will now create the lines between these points and place them.

Scroll down the left menu to **Output Sets** and click **Add a polyline set** to create the first output set.




1. Click the default name to access the properties and rename it into **narrowTopPath**. “Top” means that this path concerns the top of the base cabinet; “narrow” means the front of the cabinet.
2. Click **Add a polyline** to specify that you will draw a line. You cannot rename the polyline.

3. Click **Add a point** to specify the start point of the line, and repeat this step to create the final point of the line. You cannot rename the points.
4. Click the first **Point (0, 0, 0)** to access the coordinates of the start point. You could enter fixed coordinates but we will use the relations created previously.
5. Move the cursor to the right of the **X** field to **Symbol**, to display the list of relations.
6. Select **xPointONarrowPosition** in the list.
7. Enter values in the **Y** and **Z** fields, because the scaling of the cabinet is limited to the X-axis:
    a. Enter **-300** in the **Y** field**,** this value being the half of the depth of the box;
    b. Enter **880**, in the **Z** field, this value being the total height of the cabinet (box and legs).



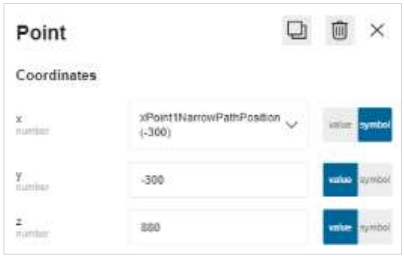

Do not worry if a line appears across the box. The second point is not defined yet.

Click this second point and repeat steps 5 to 7 to define it with the following values:

**X** field: The **xPoint1NarrowPathPosition** relation

**Y** field: **-300**

**Z** field: **880**

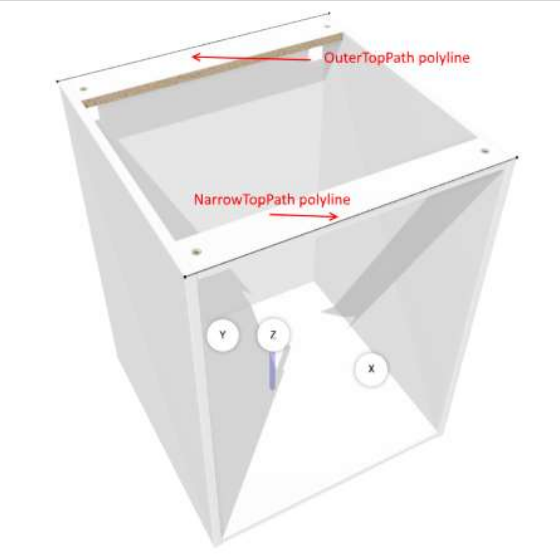



The first outline is now specified.

Repeat the whole proceeding to define the second polyline.

1. Scroll down the left menu to **Output Sets** and click **Add a polyline set** to create a second output set.
2. Rename it into **outerTopPath**. “outer” means the back of the cabinet.
3. Click **Add a polyline**.
4. Click **Add a point** twice to add two points. You cannot rename the points.
5. Click the first **Point (0, 0, 0)** and specify the following coordinates.
    - **X** field: select the **xPoint0NarrowPathPosition** relation
    - **Y** field: enter **300** (i.e. the half of the depth of the box)
    - **Z** field: enter **880** (i.e. the height of the box plus the legs)
6. Click the second **Point (0, 0, 0)** and specify the following coordinates.
    - **X** field: select the **xPoint1NarrowPathPosition** relation
    - **Y** field: enter **300** again
    - **Z** field: enter **880** again

The output sets are now defined as illustrated below. The box is ready to receive a worktop in the Kitchen Planner.




## Save the Box and Legs Sub-Assembly

Following the data model, (see above Data Model) the box and the legs must be saved as a sub-assembly. This is recommend to make it reusable and necessary to make them work as a group in further animations.

### *Save the Work*

Click the **Save my work** icon on the up right corner to save the sub-assembly as a .BMA file. Assembly Editor produces a zip file containing a root.BMA file. Move the zip file to your project location and rename the file into box_leg.BMA.

| Name | Date modified | Type | Size |
|---|---|---|---|
| D036A935-8F63-4B78-A02F-410ECA1EA382.ZIP | 07/01/2020 17:11 | zip Archive | 1 KB |
| box_legs.BMA | 07/01/2020 17:11 | BMA File | 2 KB |

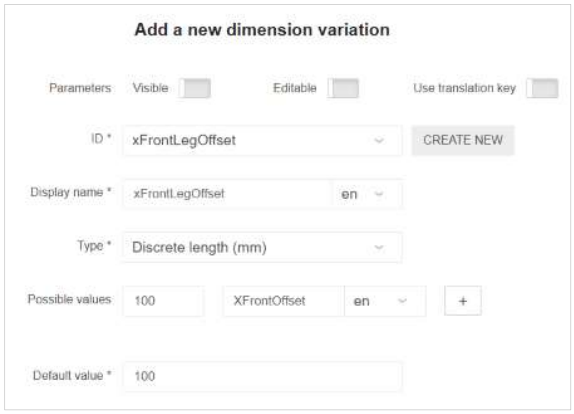
### *Create a Datasheet*

Then, create a product datasheet in 3DCloud for this sub-assembly. Import the .BMA file as 3D Model representation.

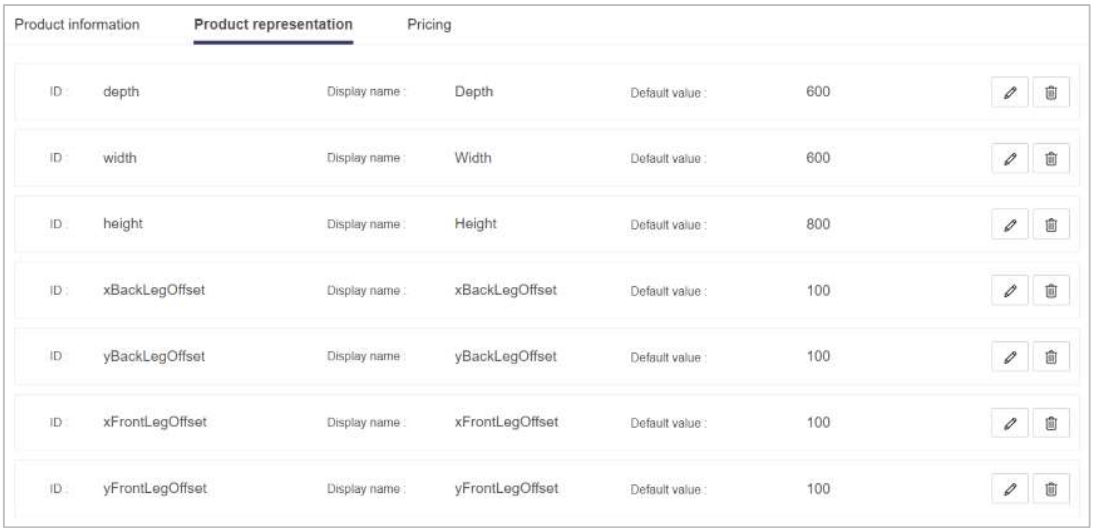
You have to define in the 3DCloud datasheet all the parameters that are defined in Assembly.

For our “box and legs” sub-assembly, this means that you have to create the size parameters (W600 / D600 / H800) and the offset parameters.

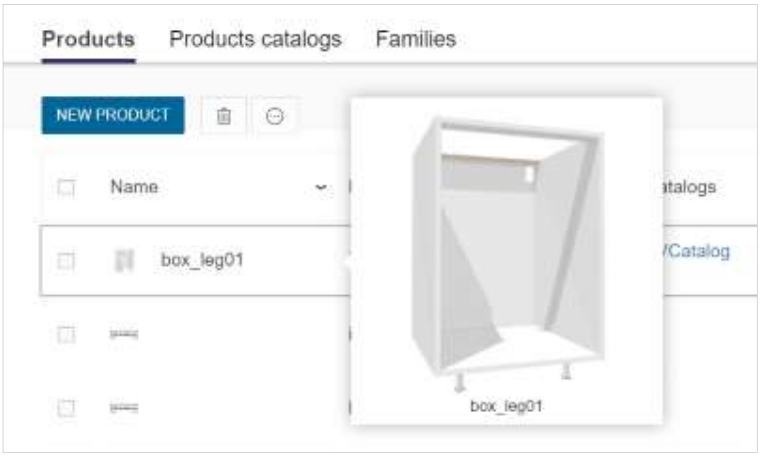
Add the size parameters, then click **Variable dimension** again to add the offset parameters.




1. Click **Create New** and enter **xFrontLegOffset**. Press **Enter** to add it to the **ID** field.
2. Specify the **Display name** field with the same.
3. Select **Discrete length (mm**) to enter a value.
4. Enter **100** in the **Possible values** field and as default value.
5. Click **Validate**.
6. Repeat the proceeding to create the **xBackLegOffset**, **yFrontLegOffset** and **yBackLegOffset**.




Finally, add the new product to the catalog.




Whereby, you will be able to import it as a component in Assembly Editor to continue building the base cabinet.

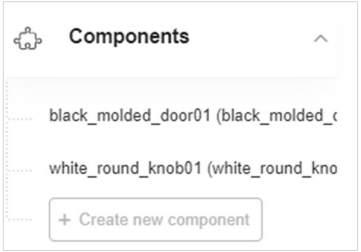
## Create the "Front and Handle" Sub-Assembly

Start from new to create the second sub-assembly.

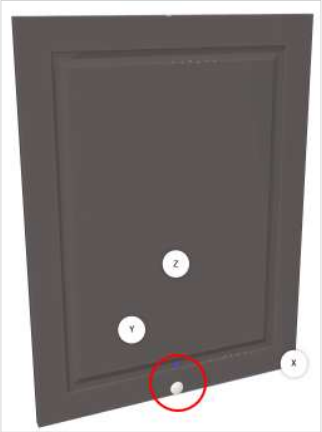
### Add the Components

1. Click **Create new component** on the left menu, or click **Add a component** on the scene.
2. Browse the catalog by specifying the name of the product in the search field. Use either the complete name as defined in the 3DCloud datasheet or a term of the type e.g. "box" or "front". Press **Enter** to start the search.

3. Click the thumbnail of the product to add it to the list of components and display it on the scene.
4. Repeat the proceeding to add the handle (i.e. the white knob). Now you have the following two components.



Remember that the components are centered by default on the origin of the axes. Their final position is not yet defined.



## Rename the Components

Select a component in the list or on the scene to display its properties.

Rename the door front component into **DoorFront** and the knob into **Handle**.

Important: Always capitalize the first letters of the words in a compound name to avoid confusion with parameter names.

## Component Parameters

Note: You create parameters as needed; this is not a batch process with the aim of creating all possible parameters just in case.

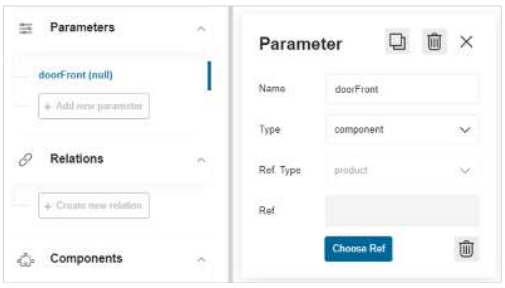
### *Create the Component Parameters*

Important: Use camel case convention when renaming the parameters.

You have to create one parameter per component.

To create a parameter, proceed as follows:

1. Click **Add new parameter** in the **Parameters** area.
2. Click the parameter to display its properties.
3. Replace the default name by one matching the naming convention.
4. Select **component** in the **Type** drop-down list. The associated reference type is specified automatically.
5. Changes are saved automatically. Click the cross icon to close the **Parameter** panel.

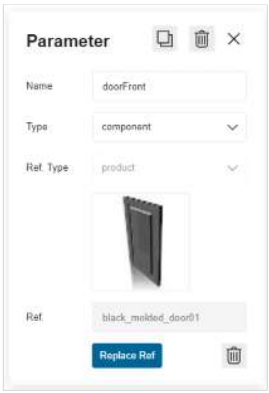


For the “front and handle” sub-assembly of our sample cabinet, create the following parameters: doorFront and handle.

### *Link the Parameters with the Components*

The next step consists in linking each parameter to its component.

1. Click the first parameter e.g. **doorFront** to access its properties.
2. Click **Choose ref** to select a product reference to link with the parameter.
3. Search for the product using its 3DCloud name. Then, click the product thumbnail to add it as reference.




4. Repeat the proceeding to link the handle parameter with the handle product and get the following. Note that the component reference has replaced “null” in the parentheses.

## Size Parameters and Relations

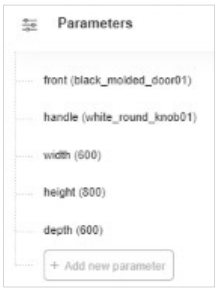
The next step consists in creating main size parameters identical to the size parameters of the whole cabinet. Relations will refer to these parameters to calculate the real dimensions of the door front dynamically.

Important: These size parameters are mandatory.

### *Main Size Parameters*

1. Click **Add new parameter** in the **Parameters** area.
2. Click the parameter default name to access its properties.
3. Rename the parameter into **width**.
4. Select **number** in the **Type** drop-down list.
5. Enter a value, i.e. **600** for the door front.
6. Repeat the proceeding with the height (800) and the depth (600).

The **Parameters** area now contains the following parameters:



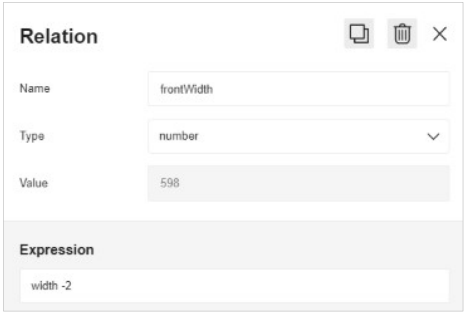

### *Size Relations*

The “front and handle” sub-assembly will use relations to define dynamically some dimensions of the front and the handle.

You have to create a relation to link the width of the door front to the main width parameter while considering its real size (remember that the width of the door front is set to 598 mm in its datasheet).

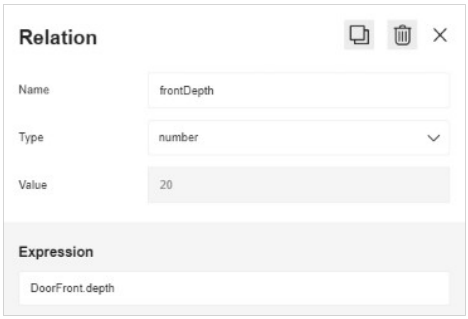
Start creating the width relation for the front.

1. Click **Create new relation**.
2. Click the default name to display the properties of the relation.
3. Rename the relation into **frontWidth**.
4. Select **Number** as type.
5. Enter **width –2** in the **Expression** field and press **Enter** to update the **Value** field above. The width of the door front is now **598** mm.




Continue by creating the depth relation.

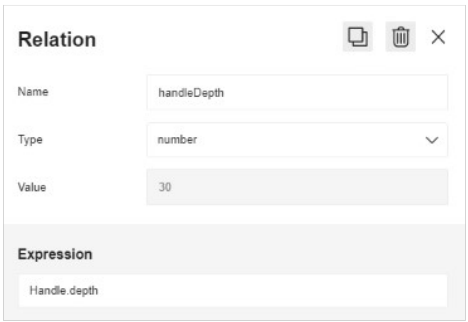
1. Click **Create new relation**.
2. Click the default name to display the properties of the relation.
3. Rename the relation into **frontDepth**.
4. Select **Number** as type.
5. Specify the **Expression** field with the following formula, where the dot means to get the value of the depth parameter for the DoorFront component: **DoorFront.depth** and press **Enter** to update the value.




Note: This type of expression cannot be “undefined” (i.e. =”null”). Do not forget to test the value.

Finally, create a handle size relation.

1. Click **Create new relation**.
2. Click the default name to display the properties of the relation.
3. Rename the relation into **handleDepth**.
4. Select **Number** as type.
5. Enter an expression relating to the handle component and the depth parameter: **Handle.depth** and press Enter to update the value.



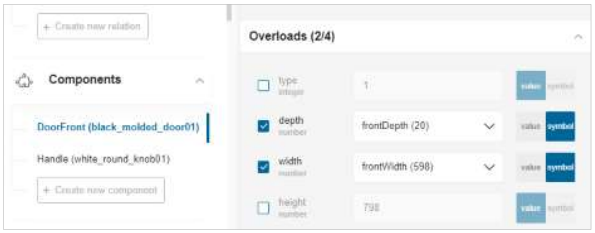

### *Link the Size Parameters to the Door Front*

This step consists in defining the size of the component using the size relations.

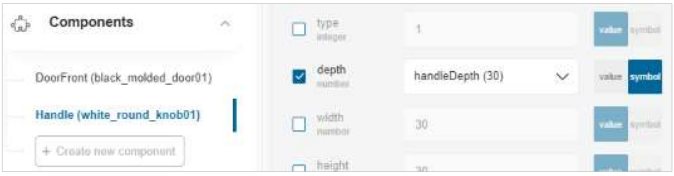
The front width must vary while stretching the cabinet; front depth is defined dynamically for the case when the customer changes the door with one thicker or thinner, to ensure that the handle remains well placed.

1. Click the **DoorFront** component in the list to access its properties.
2. Scroll down to the **Overloads** field. The size parameters displayed in this area are recovered from the 3DCloud datasheet.
3. Select the check box regarding the **width** of the component to enable the modifications.
4. Move the cursor to **Symbol** to display the list of parameters declared in Assembly.
5. Select the **frontWidth** relation.
6. Select the depth check box, move the cursor to **Symbol** and select the **frontDepth** relation.

Those calculated via the relation now overload the width and depth of the “DoorFront” component.




Repeat the proceeding with the depth of the handle. Select the **Handle** component and link it to the **handleDepth** relation.



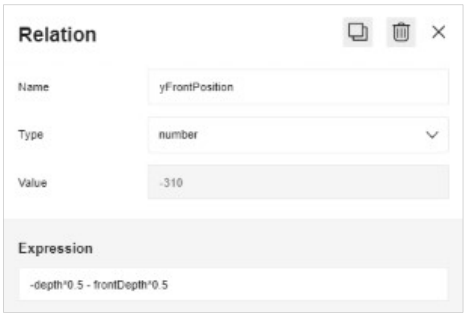

## Front Position

By default, the door is placed on the scene at the origin point like any component. Now we will place it on the front of the box.

### *Create Position Relations*

The position of the front on the Z-axis must be calculated dynamically to follow the stretching of the whole cabinet.

1. Click **Create new relation**.
2. Click the default name to display the properties of the relation.
3. Rename the relation into **yFrontPosition**.
4. Select **Number** as type.
5. Enter **-depth*0.5 - frontDepth*0.5** in the **Expression** field. The formula is based on the half of the main depth and the half of the front depth.
6. Press **Enter** to update the **Value** field above. The value is now -310.



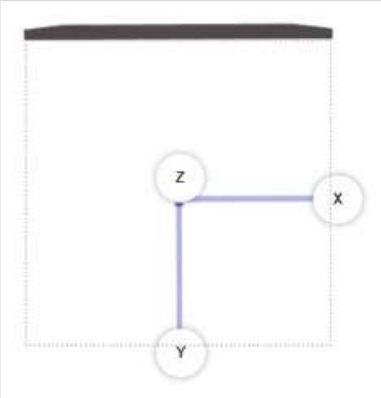

### *Door Front Position*

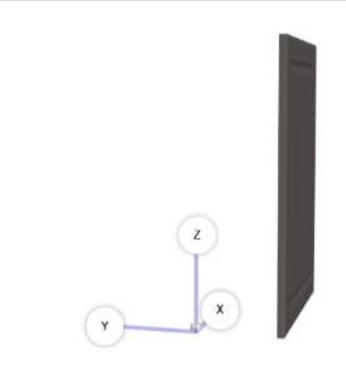
The final step consists in overloading the position of the door front component with the front position relation.

1. Click the **DoorFront** component to access its properties.
2. Go to **Position** and move the cursor to **Symbol** for the Y position.

3. Select **yFrontPosition** in the list of relations.

The door is now placed as illustrated below. Note that the handle is still on the default position.




## Opening Side Parameter

The next step for this sub-assembly is to define a side parameter, as in the 3DCloud datasheet, to specify the opening side of the combination door front and handle.

This parameter is in relation with the future animation of the base cabinet: opening the door.

1. Click **Add new parameter** and click the default name to access the properties.
2. Rename it into **side**.
3. Select **number** (or **integer**) in the drop-down list.
4. Enter **-1** in the **Value** field because the door front is left opening (1 for a right opening door).
5. Press **Enter** to validate the parameter.

Note: To be valid, the side parameter must be used in two relations: the x-position of the handle and the rotation axis. See hereinafter.

## Handle Position

The position of the handle must remain constant even if the door is stretched horizontally. The relation will be calculated from the main size parameters and offsets to specify the position from the edge of the front.

For our sample cabinet we will use an offset of 160 mm from the top and side edges of the door to place the knob.

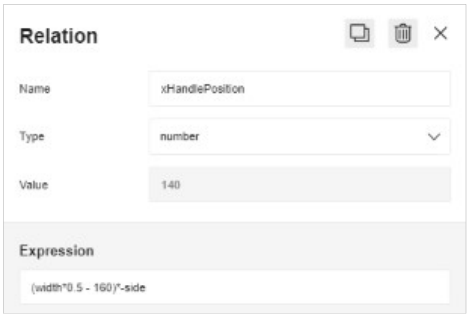
### *Create Position Relations*

Start by creating the position on the X-axis.

Note that this relation will use the side parameter that we have just created.

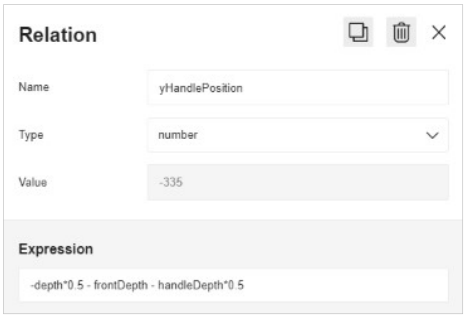
1. Click **Create new relation**.
2. Click the default name to display the properties of the relation.
3. Rename the relation into **xHandlePosition**.

4. Select **Number** as type.
5. Enter **(width*0.5 - 160)*-side** in the **Expression** field.
6. Press **Enter** to update the **Value** field.



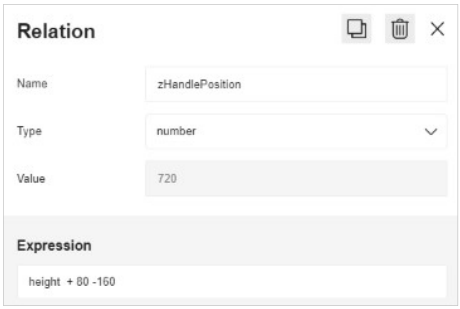

Continue by creating the position on the **Y**-axis. The formula is based on the depth parameter, the depth of the front and the depth of the handle.

1. Click **Create new relation**.
2. Click the default name to display the properties of the relation.
3. Rename the relation into **yHandlePosition**.
4. Select **Number** as type.
5. Enter **-depth*0.5 - frontDepth - handleDepth*0.5** in the **Expression** field.
6. Press **Enter** to update the **Value** field.




Finally, create the position on the Z-axis. The formula calculates the position from the main height parameter, the height from the floor (the legs actually) and the offset.

1. Click **Create new relation**.
2. Click the default name to display the properties of the relation.
3. Rename the relation into **zHandlePosition**.
4. Select **Number** as type.
5. Enter **height + 80 -160** in the **Expression** field (where 80 is the height from the floor and 160 is the offset of the handle from the edge of the door front).
6. Press **Enter** to update the **Value** field.

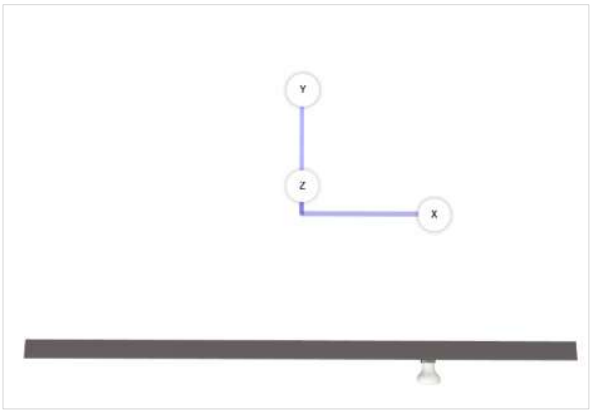



### *Handle Position*

Now that the position relations are defined, you have to overload the position of the handle with these relations.

1. Click the **Handle** component to access its properties.
2. Go to **Position** and move the cursor to **Symbol** for the **X** position.
3. Select **xHandlePosition** in the list of relations.
4. Overload the **Y** value with **yHandlePosition** and the **Z** value with **zHandlePosition**.

The handle is now placed as illustrated below:



## Add an Animation

Animations make it possible to define kinematic behaviors in the Kitchen Planner, such as opening the door of the base cabinet.

Animations - either a rotation or a translation – apply to the components of an assembly *as a whole*. Thus, do not create the door animation at the top-assembly level because it would not be possible to rotate only the door front component within the whole assembly. Create the door front animation on the “front and handle” sub-assembly.

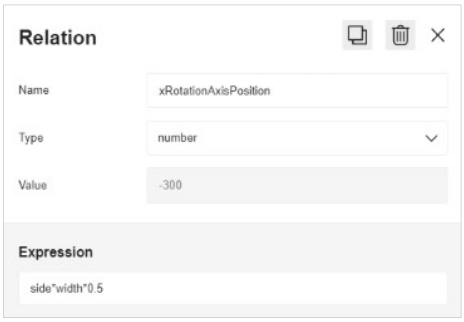
The animation of opening a door is a rotation: the door front will rotate on the Z-axis. A translation would consist for example in opening a drawer from a cabinet on the Y-axis.

### *Create a Relation*

To give the customer the possibility to change the direction of the door opening in the Kitchen Planner, you have to define the position of the rotation axis dynamically.

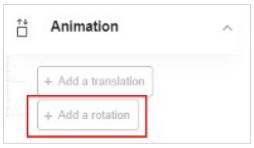
1. Click **Create new relation**.

2. Select the default name to access its properties and rename it into **xRotationAxisPosition**.
3. Enter the following formula in the **Expression** field, using the side and width parameters: **side*width*0.5**.
4. Press **Enter** to update the value.

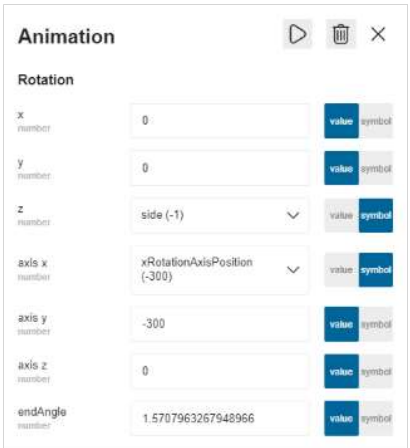



### *Create the Rotation*

Scroll down the left menu to **Animation** and click **Add a rotation**.



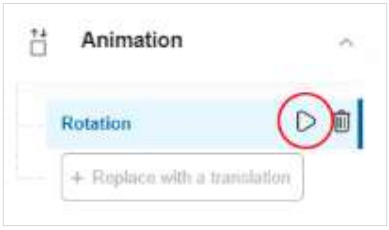

1. Click the rotation to display its properties.
2. Specify the direction of the rotation: move the cursor to **Symbol** for the **Z** field and select the **side** parameter in the list.
3. Specify the position of the rotation axis:
    a. Select **Symbol** for the **axis x** field and then, **xRotationAxisPosition**.
    b. Enter -300 (which is the half of the box depth) in the **axis y** field to place the rotation axis on the front.
    c. Leave the **axis z** blank.
4. Keep the default value in the **endAngle** field.

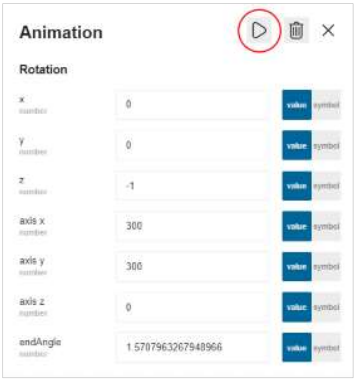


### *Test the Animation*

Test the animation by clicking the **Play** icon either on the left menu or on the property panel.

Also test the possibility of a right opening door by changing the value of the side parameter to **1** and clicking the **Play** icon.




## Save the Front and Handle Sub-Assembly

Following the data model (see above Data Model) the door front and the handle must be saved as a sub-assembly. This is recommend to make it reusable and necessary if you want to add an animation that will open the door front (handle and door front must work as a group).

### *Save the Work*

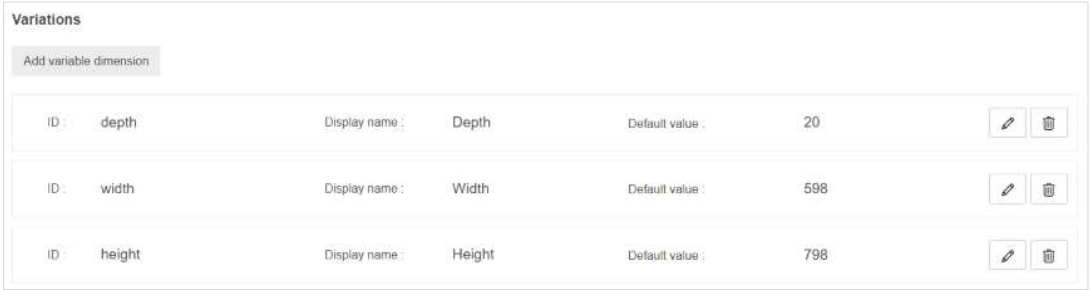
Click the **Save my work** icon on the up right corner to save the sub-assembly as a .BMA file. Assembly Editor produces a zip file containing a root.BMA file. Move the zip file to your project location and rename the file into front_handle.BMA.

### *Create a Datasheet*

Then, create a product datasheet in 3DCloud for this sub-assembly. Import the .BMA file as 3D Model representation.

You have to define in the 3DCloud datasheet all the parameters that are defined in Assembly.

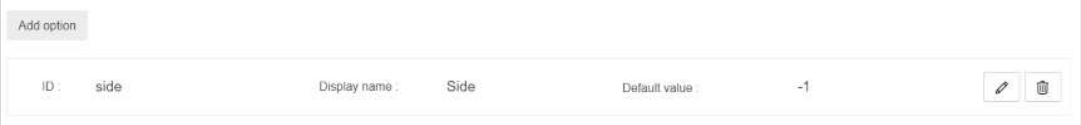
Click **Variable dimension** to add the size parameters of the door front, as illustrated below:




Then, add an option to specify the side opening of the door. Our door front is a left opening door.

1. Click **Create new** and enter **side** in the text field.

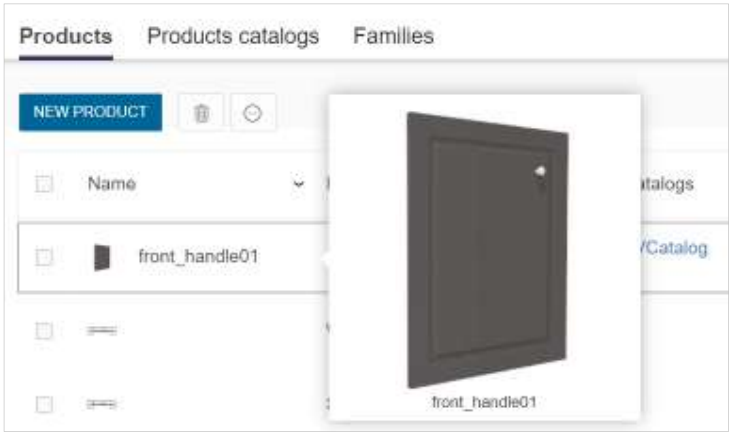
2. Click the cross to add it to the **ID** field.
3. Enter **Side** as display name.
4. Select **Discrete variation** to enter a choice of values.
5. Enter **-1** and **Left** as first possible value.
6. Enter **1** and **Right** as second possible value.
7. Enter **-1** as default value.
8. Click **Validate**.




Add a **Product replacement** option to create a choice of handles in the Kitchen Planner.

1. Click **Create new** and enter **handle** in the text field.
2. Click the cross to add it to the **ID** field.
3. Enter **Handle** as display name.
4. Select **Allow empty value** and **Allow any value** to open the choice to any existing handle in the catalog .
5. Select the ceramic knob, i.e. **white_round_knob01** as default product.
6. Click **Validate**.

Finally, add the new product to the catalog.



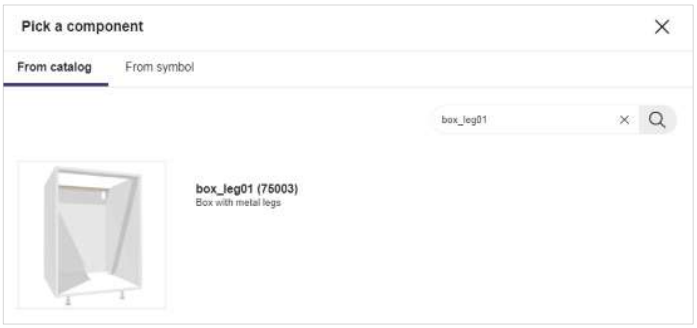

Whereby, you will be able to import it as a component in Assembly Editor to continue building the base cabinet.

## Import the Sub-Assemblies as Components

Now that both sub-assemblies are created and registered as products in 3DCloud, we can add them to a new assembly to be a part of the future top-assembly.

## Create Components

1. Click **Create new component** on the left menu, or click **Add a component** on the scene.
2. Browse the catalog by specifying the name of the "box and legs" sub-assembly in the search field, i.e. **box_leg01**. Press **Enter** to start the search.




3. Click the thumbnail of the product to add it to the list of components and display it on the scene.
4. Repeat the proceeding to add the "front and handle" sub-assembly. Now you have the following two components.
5. Rename the **box_leg01** component into **BoxWithLegs** and the **front_handle** into **FrontWithHandle**.

Important: Always capitalize the first letters of the words in a compound name to avoid confusion with parameter names.

## Component Parameters

Important: Use camel case convention when renaming the parameters.

You have to create one parameter per component, even if the component is a sub-assembly.

Actually, the only important parameter in our case is "handle" because this is the only component that we want the customer to control (when replacing the default handle with another one in the Kitchen Planner).

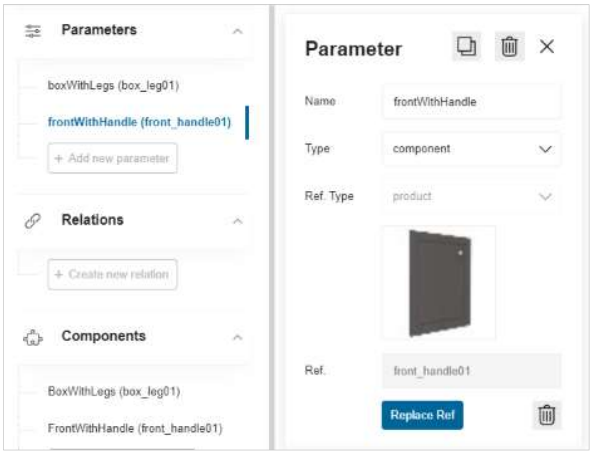
To create a parameter, proceed as follows:

1. Click **Add new parameter** in the **Parameters** area.
2. Click the parameter to display its properties.
3. Change the default name into **boxWithLegs**.
4. Select **component** in the **Type** drop-down list. The associated reference type is specified automatically.
5. Click **Choose Ref** to select a product reference to link with the parameter.

6. Search for the product using its 3DCloud name. Then, click the product thumbnail to add it as reference.

Changes are saved automatically. Repeat the proceeding to create a **frontWithHandle** parameter.

You now have the following parameters.




## Size Parameters

The next step consists in creating the size parameters for the sub-assemblies. Even if they have been specified at the creation, this is a new project and you need to specify the size parameters again.

### *Create the Size Parameters*

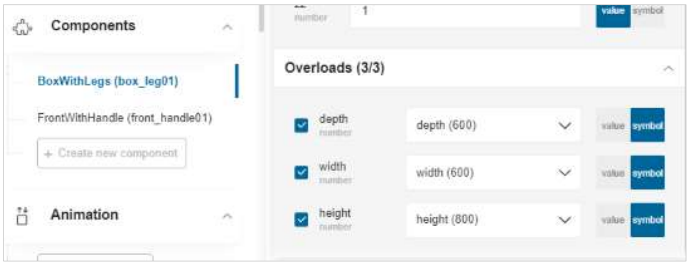
1. Click **Add new parameter** in the **Parameters** area.
2. Click the parameter default name to access its properties.
3. Rename the parameter into **width**.
4. Select **number** in the **Type** drop-down list.
5. Enter a value, i.e. 600 that is the width of the future base cabinet, determined by the width of the box.
6. Repeat the proceeding with the height (800) and the depth (600). The **Parameters** area now contains the following five parameters.

### *Link the Size Parameters to the Door Front*

This step consists in linking the parameters of the *product* to those of the *components*.

1. Click the **BoxWithLegs** component in the list to consult its properties.

2. Scroll down the panel to reach the **Overloads** field. The size parameters displayed in this area are recovered from the 3DCloud datasheet are displayed in a drop-down list.
3. Select the check box regarding the **depth** of the component to enable the modifications.
4. Move the cursor to **Symbol** to display the list of parameters declared in Assembly.
5. Select the **depth** parameter. The depth parameter of the “box and legs” product is now linked to the depth parameter of the “BoxWithLegs” component.
6. Repeat the proceeding with the **width** and **height** of the door front. The **Overloads** area now indicates that the **Overload Editor** contains three parameters.




Repeat the proceeding with the **FrontWithHandle** component to overload the width of the component with the width parameter. Leave the other sizes as such.

Both “box and legs” and “door front and handle” combinations must be linked dynamically to the width parameter to prepare the scaling of the base cabinet on the X-axis.

## Place the Shelf

The next step consists in placing the shelf inside the box, centered vertically. Take the height of the box into account (800 mm) and the position of the box from the floor (80 mm).

Start by creating a new component with the shelf of the catalog as resource, and rename it into **Shelf**.

The shelf is placed by default on the bottom, as usually.

### Define the Position of the Shelf

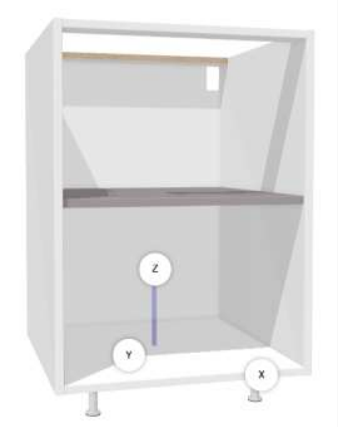
Define now the position of the shelf on the X, Y and Z axes.

1. Select the **Shelf** component in the **Components** list to access its properties.
2. Go to the **Position** fields.

3. Leave the **X** and **Y** position fields blank.
4. Enter **480** in the **Z** position field, which is the half of the height of the box *plus* the height from the floor.

The shelf is now placed in the box, as illustrated below:

Tip: To view the shelf in the box, click the eye icon of the FrontWithHandle component to hide it.



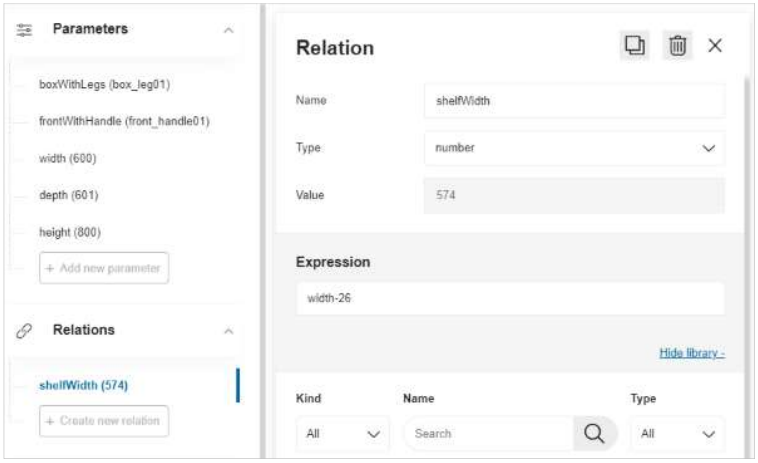

## Shelf Width Relation

To enable the scaling of the shelf with the whole cabinet, the width of the shelf must call a relation between its own size and the main width parameter.

Note: Do not create specific width parameters because this would dissociate the shelf from the box when scaling it.

Thus, you have to create a relation that will reduce the global width recovered from the width parameter to adapt it to the shell.

1. Click **Create new relation**.
2. Click the default name to display the properties of the relation.
3. Rename the relation into **shelfWidth**.
4. Select **Number** as type.
5. Enter **width–26** in the **Expression** field (without any space). This updates the **Value** field above: the width of the shelf is now **574** mm.



### Link the Width of the Shelf to the Relation

This step consists in overloading the default width parameter recovered from the datasheet with the value of the relation.

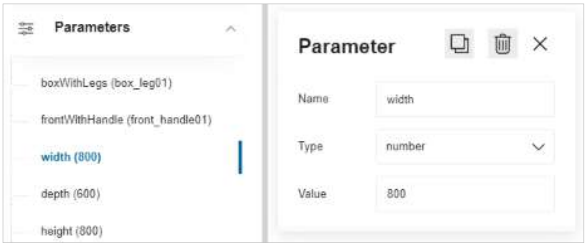
1. Click the **Shelf** component in the list to consult its properties.
2. Scroll down the panel to reach the **Overloads** field.
3. Select the check box regarding the width of the component to enable the modifications.
4. Move the cursor to **Symbol** to display the list of parameters declared in Assembly.
5. Select the width parameter and move the cursor to **Symbol**.
6. Select the **shelfWidth** *relation* in the drop-down list.
7. Leave the other sizes as such.

## Test the Scaling

We enabled a scaling in Geometry Editor to enlarge the base cabinet from 600 to 800 mm. In this perspective, we have also created relations at the sub-assembly levels to create dynamic dimensions and positions.

The base cabinet is thus “scaling ready” and you can test it now.

If you change the value of the width parameter, the base cabinet will increase proportionally.



## Create a Side Parameter

The possibility of choosing the opening side of the door front implies to define a side parameter at the top-assembly level too.

### Create the Parameter

1. Click **Add new parameter** and click the default name to access the properties.
2. Rename the parameter into **side**.
3. Select **number** (or **integer** to be consistent with the side parameter of the door front) in the **Type** drop-down list.
4. Enter **-1** as value because the door front is by default left opening.

### Overload the Parameter

The side value declared at the component level must be overloaded by the side parameter

1. Select the **FrontWithHandle** component and go to the **Overloads** area.
2. Check the box on the left of the **side** parameter.
3. Move the cursor to **Symbol**.
4. Select the **side** parameter in the drop-down list.

### Test the Opening

Test the opening side of the door front by changing the value of the side parameter to **1**.

## Handle Replacement

Creating the possibility for the customer to choose between various handles for the same door is a long process that started with a handle parameter in the ‘”front and handle” sub-assembly.

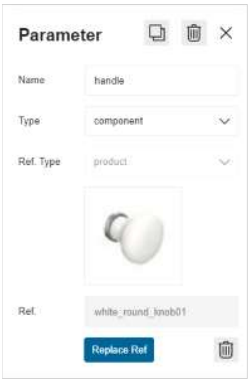
The action to perform at the top-assembly level is to create a handle parameter and to overload the component value with it.

Note: The final step will be the creation of a product replacement option in the top-assembly datasheet.

### Handle Parameter

Start by creating the handle parameter.

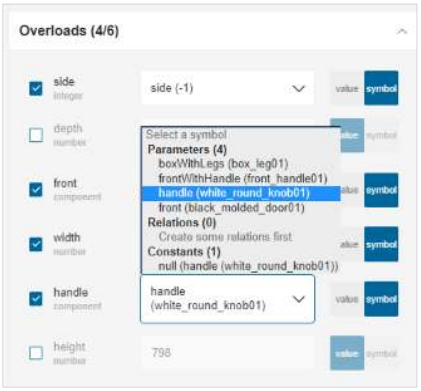
1. Click **Add new parameter**.
2. Click the parameter to access its properties and rename it into **handle**.
3. Select **component** in the **Type** drop-down list.
4. Click **Choose Ref** to search for the product using its 3DCloud name, i.e. **white_round_knob01** that is the default handle for this cabinet.
5. Click the product thumbnail to add it as reference.




## Handle Component

Continue by overloading the component.

1. Go to **Components** and select **FrontWithHandle** to access its properties.
2. Go to **Overloads**.
3. Select the check box of the **handle** component.
4. Move the cursor to **Symbol**.
5. Select the **handle** parameter in the drop-down list.




Test the overload by changing the product referenced in the handle parameter.

## Save

Click the **Save my work** icon on the up right corner to save the top-assembly as a .BMA file.

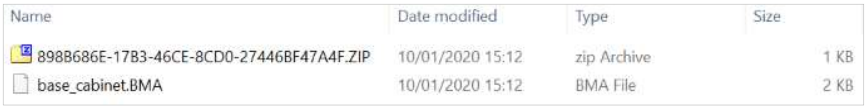
Assembly Editor produces a zip file containing a root.BMA file. Move the zip file to your project location and rename the file into base_cabinet.BMA (which will replace the previous one).

The base cabinet is now ready to be displayed in the Kitchen Planner.

## Save the Top-Assembly

Click the **Save my work** icon on the up right corner to save the top-assembly as a .BMA file.

Assembly Editor produces a zip file containing a root.BMA file. Move the zip file to your project location and rename the file into base_cabinet.BMA.



# STEP 6 – SAVE THE CABINET AS A PRODUCT

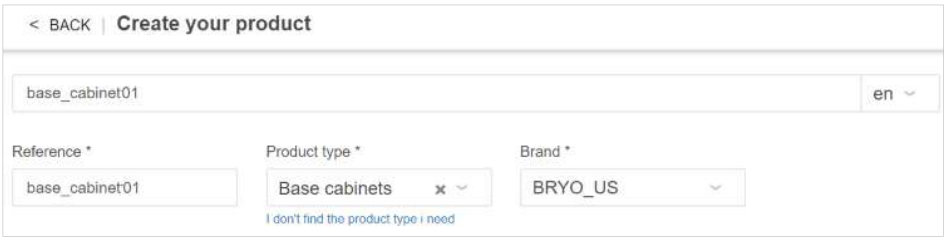
To make the cabinet usable in other assemblies or visible in the Kitchen Planner, you have to create a product datasheet in 3DCloud for the top-assembly.

Important: The 3D model in this datasheet is the base_cabinet.BMA file.

## Create a Base Cabinet Datasheet

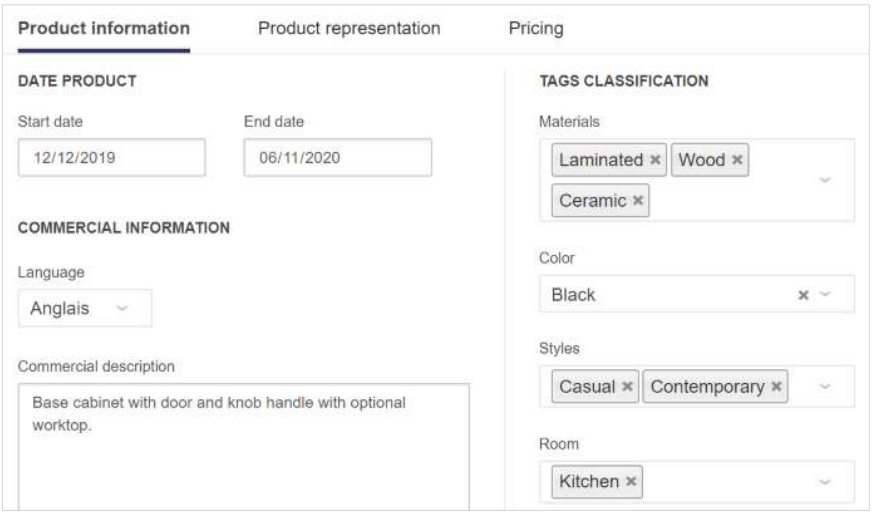
Click **New Product** in the **Products** list to create a datasheet.

General Information




1. Give a name to the product: select a language in the drop-down list and then, enter a name. Repeat this step for each language. This is an internal name; it will not be displayed to the customer.
2. Define an internal reference for the product that will be used in Assembly Editor. This reference is different from the product reference displayed in the Kitchen Planner.
3. Link the product to a product type, i.e. **Base cabinets**.
4. Link the product to your Brand.

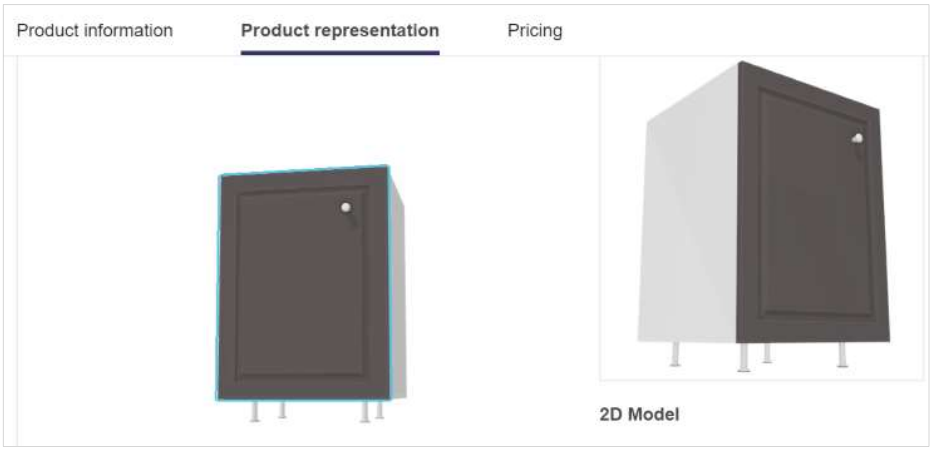
Product Information




1. Define a period of availability. These dates always overwrite the dates set in the catalog.

2. Choose a language to use in the Kitchen Planner.
3. Enter the commercial description and a short description of the cabinet.
4. Define tags to ease the classification of the base cabinet in the Kitchen Planner.

Product Representation




1. Click **Upload a 3D model** to search for the root.BMA file on your drive.
2. Add a bitmap picture of the base cabinet by clicking **Upload a picture**. It will be used as thumbnail in the product list in 3DCloud and in the Kitchen Planner.

### *Technical Information*

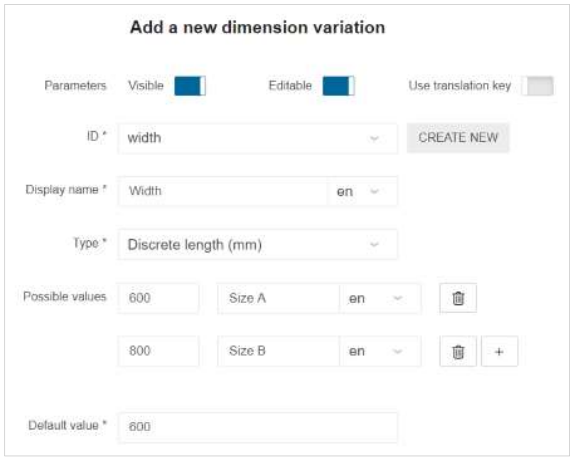
For our base cabinet, specify the following information. They will have an impact in the Kitchen Planner. Keep the default specifications for the other fields.

- **Target position**: OnFloor
- **Height from floor**: N/A
- **Manip mode**: Rotate, Translate.

The manipulation mode only applies to the whole product you will insert in the Kitchen Planner (i.e. a top-assembly). Select the first two modes for the base cabinet.

### *Variable Dimension*

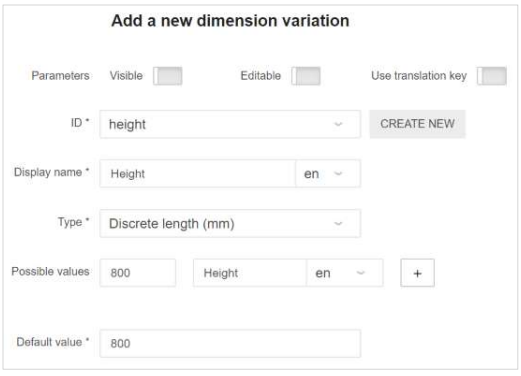
This time, we will enable the **Visible** and **Editable** parameters when specifying the width and the options of the base cabinet in order to make them visible and editable in the Kitchen Planner.

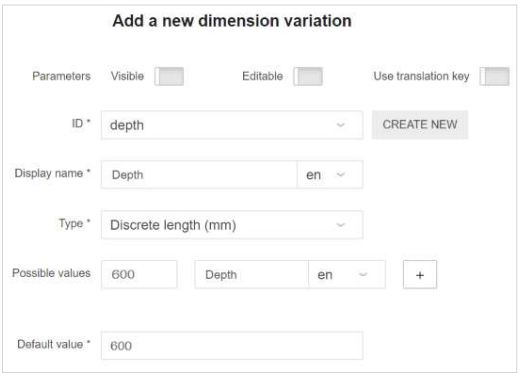



1. Check the **Visible** and **Editable** options to allow the customer to edit the cabinet in the Kitchen Planner and choose between two possible sizes.
2. Select **width** in the **ID** drop-down list (or create it if need be).
3. Enter a label for the ID in the **Display name** field. Capitalize the first letter to avoid confusion with the ID.
4. Select **Discrete length (mm)**) in the **Type** list to specify a choice of values.
5. Enter **600** and **Size A** for the first possible value.
6. Enter **800** and **Size B** as second possible value.
7. Click **Validate** to save the variation.

### *Fixed Dimensions*

Repeat the proceeding to specify the height and the depth of the base cabinet, but leave the first parameters disabled.

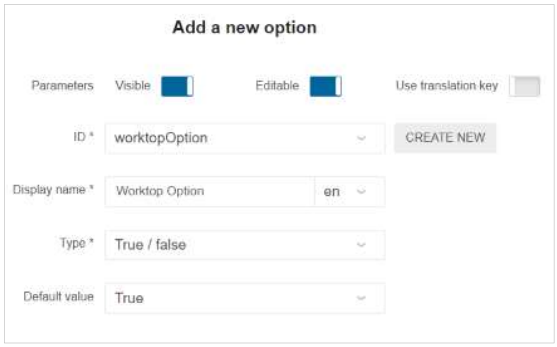



### *worktopOption*

⇨ See Appendix 2 for the list of possible parameters for a base cabinet.

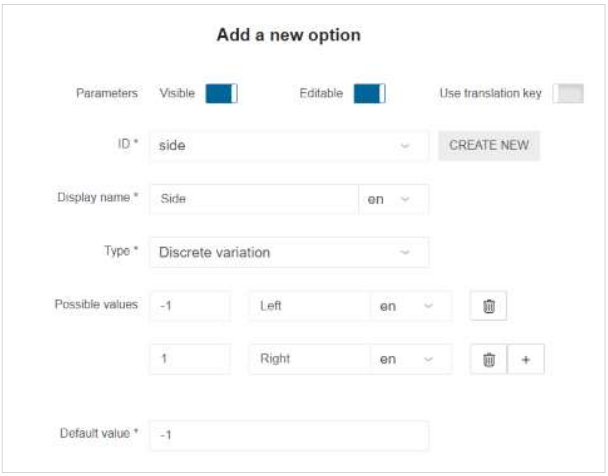
The worktopOption enables the customer to choose a worktop style that is generated by the Kitchen application automatically.




1. Check the **Visible** and **Editable** options to allow the customer to edit the worktop in the Kitchen Planner.
2. Select **worktopOption** in the **ID** drop-down list (or create it if need be).
3. Enter a label for the ID in the **Display name** field. Capitalize the first letter to avoid confusion with the ID.
4. Select **True / false** in the **Type** list because it is a boolean.
5. Enter **True** as default value.
6. Click **Validate** to add the option.

### *Opening Side*

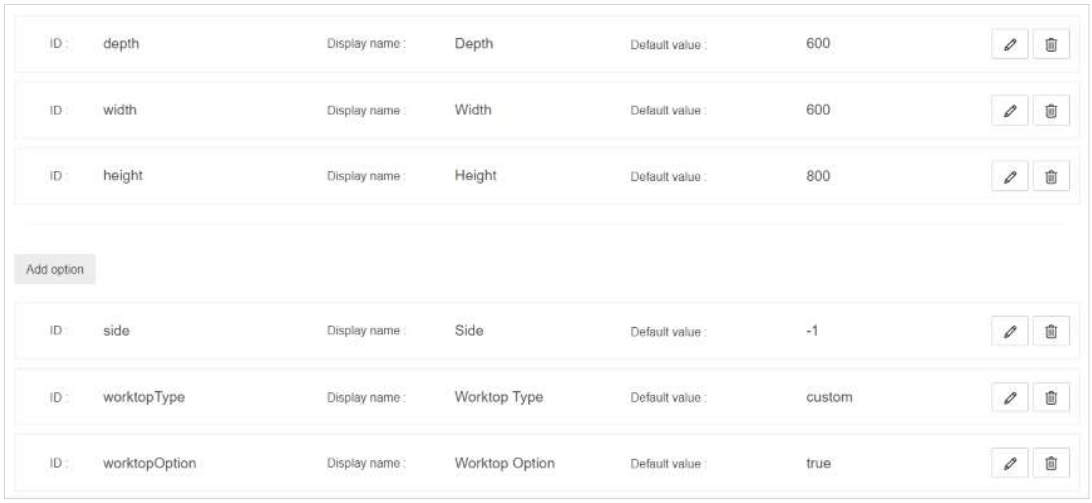
Declare the opening side of the door front in the datasheet of the top-assembly too, because a side parameter has been defined in Assembly Editor.




1. Click **Add an option**.
2. Click **Create new** and enter **side**. Click the cross to validate the new ID.
3. Enter **Side** in the **Display name** field.
4. Select **Discrete variation** in the **Type** drop-down list.
5. Enter **-1** and **Left** as first possible value.
6. Enter **1** and **Right** as second possible value.
7. Enter **-1** as default value.
8. Enable the **Visible** and **Editable** parameters on the top of the screen.

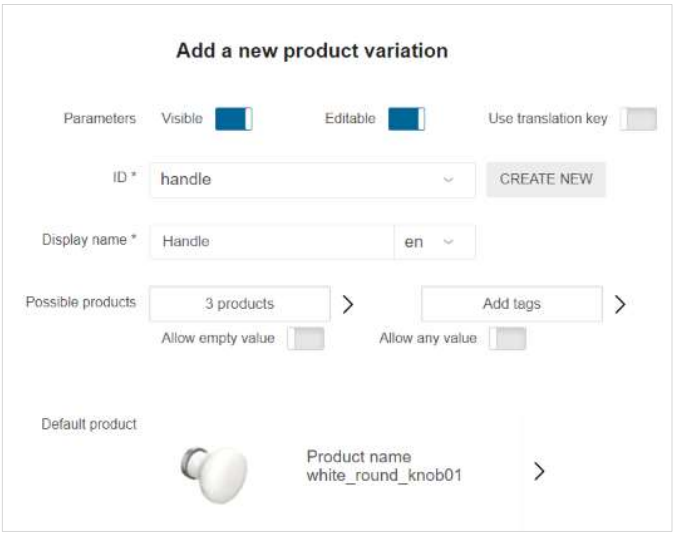
9. Click **Validate** to save the option.

Now, the parameters and options are the following:

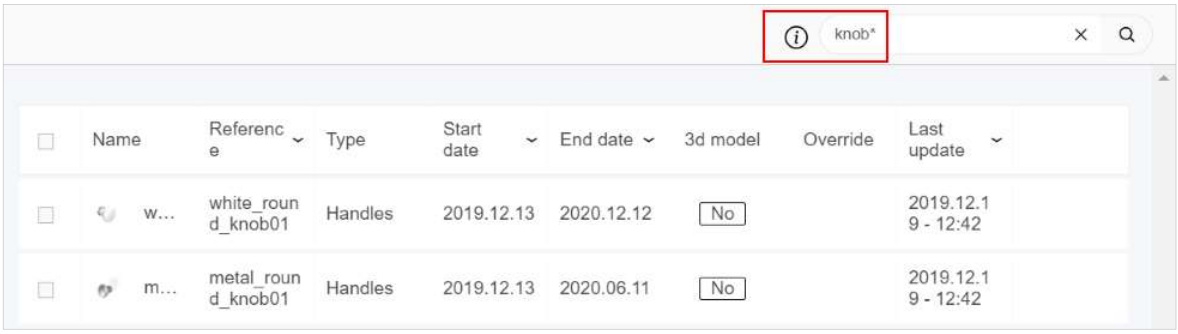



*Product Replacement*

Add a **Product replacement** option to create a choice of handles in the Kitchen Planner.



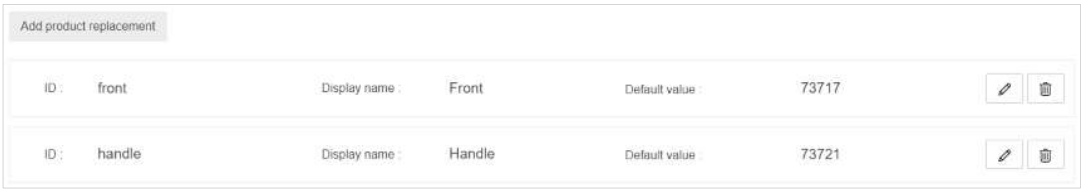

1. Enable the **Visible** and **Editable** option to allow the customer choose another handle in the Kitchen Planner.
2. Click **Create new** and enter **handle** in the text field.
3. Click the cross to add it to the **ID** field.
4. Enter **Handle** as display name.
5. Click **Add products** to open the list of all the products available in the database.
6. Enter the exact name of the first product in the search field, or use a generic word followed by an asterisk to reduce the list.




7. Select our three handles, i.e. **white_round_knob01**, **metal_round_knob01** and **metal_handle02**.
8. Click **Validate**.
9. Select **white_round_knob01** as default product.
10. Click **Validate**.

You can define a product replacement for the front too, if at least one other front datasheet already exists in 3DCloud or if you want to prepare this possibility. In this case, select the black molded door both as possible and as default product; you may add more products later.

The base cabinet now has the following replacement possibilities:




Save

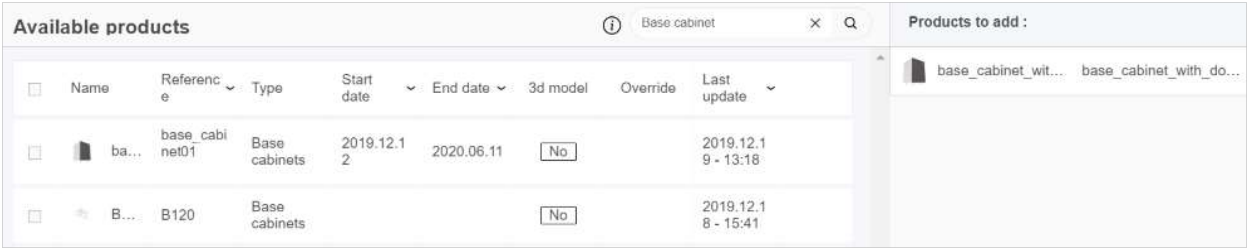
Click **Save** to upload the model and its datasheet in the 3DCloud storage. Click **Products** to see the new product in the list.

When the product is successfully added, the **3D model** column displays **Yes** in the product list.

## Add the Base Cabinet to a Catalog

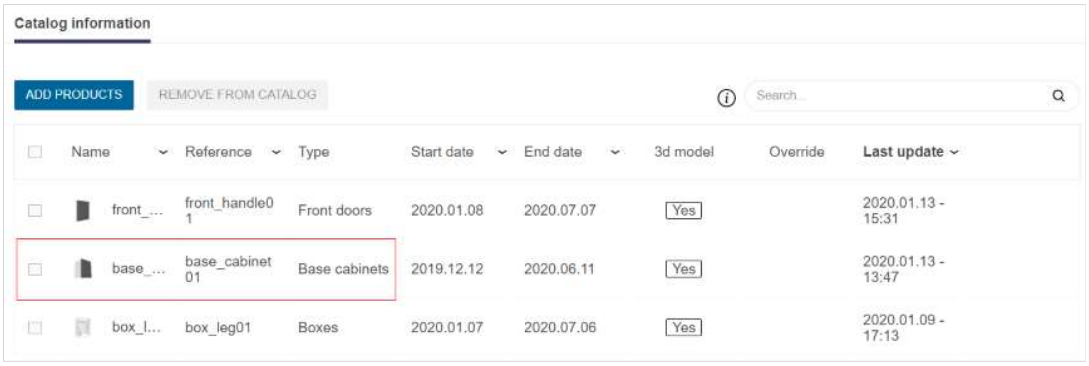
To make the base cabinet usable in Assembly Editor, you have to link it to a catalog.

1. Click the **Products catalogs** tab and search for your catalog.
2. Click **Add products** to open the list of all available products.
3. Select the base cabinet in the list.

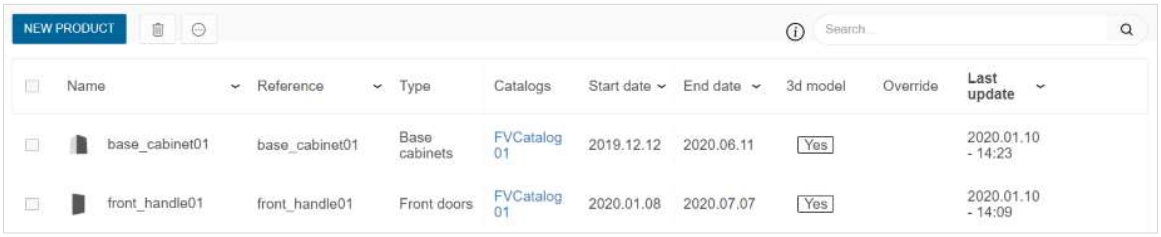



4. Scroll down and click **Validate** to add the product to the catalog.

The base cabinet is now listed in the catalog with the other products.




And it is listed with the other products, with the good catalog reference:



# STEP 7 – DISPLAY THE CABINET IN THE KITCHEN PLANNER

The final step is to display the base cabinet in the Kitchen Planner. This relies on a link between the catalog where the base cabinet is listed as product, and an application distribution.

There are three main types of kitchen application distributions:

- A type dedicated to Assembly Editor
- A type dedicated to the Template Editor
- A type dedicated to the Kitchen Planner

We will choose the third type to display our base cabinet.

## Process in 3DCloud

The process starts in 3DCloud.

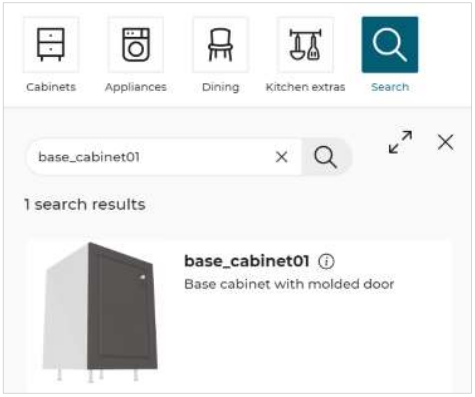
1. Open the catalog in which the base cabinet is listed.
2. Select a Kitchen type application distribution in the drop-down list.
3. Click **Save**.

The base cabinet is now linked to the appropriate application distribution.

## Insert in the Kitchen Planner

Connect to the Kitchen Planner and follow the process below:

1. Click the **Design your kitchen** tab.
2. Click the **Search** icon.
3. Enter the name of your base cabinet in the search field.
4. Press **Enter** to start the search.

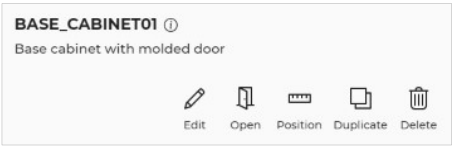



5. Click the thumbnail to add the base cabinet to the kitchen.

The cabinet is placed in a room and the worktop has been generated, as illustrated below:

## Test the Behavior in the Kitchen Planner

Click the base cabinet in the room to display the following **Edition** menu.

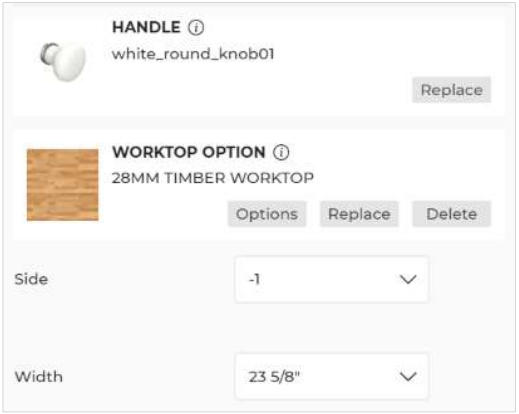



Note that this menu displays information recovered from the 3DCloud datasheet: the name and the short description.

It also contains an **Edit** icon to make some modifications on the cabinet, and an **Open** icon to play the door animation.

Via the **Edit** icon, you can test the following customizations that we have prepared on the base cabinet:

- The replacement of the handle
- The replacement of the worktop
- A choice between two door opening side
- A choice between two widths




Replace the Handle

Click the **Edit** icon to edit the replacement of the handle.

1. Click **Replace** to access the list of possible handle.
2. Click the thumbnail of the wished one, for example the metal handle.
3. Choose **Apply to the selected linear**. The base cabinet has a new handle.
4. Click **Back** to return to the change options.

## Replace the Worktop

Click the **Edit** icon to edit the replacement of the worktop.

5. Click **Replace** to access the list of possible worktops.
6. Click the thumbnail of the wished one, for example a grey worktop.
7. Choose **Apply to the selected linear**. The base cabinet has a new worktop.
8. Click **Back** to return to the change options.

## Change the Width

This option has been long prepared since Geometry Editor. It is now the time when you can test it.

Select **800** in the **Width** drop-down list to see the cabinet enlarge to 800 mm.

## Door Opening Side and Animation

Select **1** in the **Side** drop-down list to change the opening side to the right.

Click the **Open** icon to play the kinematic and see the door opening.

The base cabinet now looks different:

## CONCLUSION

Creating a base cabinet is long process through four applications: a 3D modelling software, Geometry Editor, 3DCloud and Assembly Editor, with many back and forth between 3DCloud and Assembly Editor; and finally you arrive to the Kitchen Planner.

This process should be done with the constant perspective of the result displayed to the customer in the Kitchen Planner. It recovers so much from 3DCloud and from Assembly Editor.

Each time you write a description or a display name, you write something that will be displayed to the customer.

Each time you opt for a continuous length or variation, you opt for a value field in the Kitchen Planner.

Each time you opt for a discrete length or variation, you opt for a list of choices in the Kitchen Planner.

When creating labels and parameters, remember that capitalization matters.

More difficult is to work with the perspective of the relations within the whole process. Remember that each parameter in Assembly Editor must have its duplicate in 3DCloud and vice versa.

Keep the data model in mind when creating the pyramid of components that will result in the top-assembly. Moreover, do not define parameters that are not required at this stage of the data model. Remember that the dimension parameters are the only parameters that are mandatory for every product in 3DCloud, whether it is a single product, a sub-assembly or a top-assembly. Other parameters have specific intentions, either depending on the level of assembly, or on the expected result in the Kitchen Planner.

We hope that you will use this sample cabinet to try different combinations and go further with our ByMe applications!

# APPENDIX 1 – CHECK-LIST

## Product Level

| Object | Application | Type | Name | Check |
|---|---|---|---|---|
| **BOX** | 3DCloud | Dimensions | width (600)<br>height (800)<br>depth (600) | |
| **LEG** | 3DCloud | Dimensions | width (40)<br>height (80)<br>depth (70) | |
| **SHELF** | 3DCloud | Dimensions | width (574)<br>height (19)<br>depth (575) | |
| **DOOR FRONT** | 3DCloud | Dimensions | width (598)<br>height (798)<br>depth (20) | |
| | | Option | type (1) | |
| **HANDLE**<br>**(handle / knob)** | 3DCloud | Dimensions | width (128 / 30)<br>height (20 / 30)<br>depth (30 / 30) | |
| | | Option | type (1) | |
| **WORKTOP** | 3DCloud | Dimensions | width (65)<br>height (40)<br>depth (65) | |
| | | Option | worktopType<br>(precut) | |

## Sub-Assembly Level - Box and Legs

| Application | Type | Name | Check |
|---|---|---|---|
| **3DCloud** | Dimensions | width<br>height<br>depth<br>xBackLegOffset<br>yBackLegOffset<br>xFrontLegOffset<br>yFrontLegOffset | |
| **Assembly Editor** | Components | Box<br>BackLeftLeg<br>BackRightLeg<br>FrontLeftLeg<br>FrontRightLeg | |
| | Component parameter | baseBox<br>leg | |
| | Size parameter | width (600)<br>height (800)<br>depth (600)<br>xBackLegOffset (100)<br>yBackLegOffset (100)<br>xFrontLegOffset (100)<br>yFrontLegOffset (100) | |
| | Relations | xLeftLegsRelation (-200)<br>xRightLegsRelation (200)<br>yBackLegsRelation (200)<br>yFrontLegsRelation (-200)<br>xPoint0NarrowPathPosition (300)<br>xPoint1NarrowPathPosition (-300) | |
| | Output sets | narrowTopPath (Polylines)<br>Point (300, -300, 880)<br>Point (-300, -300, 880)<br>outerTopPath (Polylines) | |

| | | | |
|---|---|---|---|
| | | Point (300, 300, 880)<br>Point (-300, 300, 880) | |

## Sub-Assembly Level – Front and Handle

| Application | Type | Name | Check |
|---|---|---|---|
| **3DCloud** | Dimensions | width<br>height<br>depth | |
| | Option | side (1 / -1) | |
| **Assembly Editor** | Components | DoorFront<br>WhiteKnob | |
| | Component parameter | doorFront<br>handle | |
| | Size parameter | width (600)<br>height (800)<br>depth (600) | |
| | Option parameter | side (1 / -1) | |
| | Relations | frontWidth (597)<br>frontDepth (20)<br>handleDepth (30)<br>yFrontPosition (-310)<br>xHandlePosition (140)<br>yHandlePosition (-335)<br>zHandleRelation (720)<br>xRotationAxisPosition (-300) | |
| | Animation | Rotation | |

## Top-Assembly Level - Base Cabinet

| Application | Type | Name | Check |
|---|---|---|---|
| **3DCloud** | Dimensions | width<br>height<br>depth | |
| | Option | side (1 / -1)<br>worktopOption<br>(true / false) | |
| | Product replacement | handle<br>front | |
| **Assembly Editor** | Components | BoxWithLegs<br>FrontWithHandle<br>Shelf | |
| | Component parameter | boxWithLegs<br>frontWithHandle<br>handle<br>front | |
| | Size parameter | width (600)<br>height (800)<br>depth (600) | |
| | Option parameter | side (1 / -1) | |
| | Relations | shelfWidth (574)<br>frontWidth (597)<br>frontDepth (600) | |

# APPENDIX 2 - BASE CABINET PARAMETERS

Important: Possible values in a range are formatted hereinafter as [0; ;∞]. However, never enter 0 as minimum value nor an infinite value as maximum value.

## General Parameters for any Product or Assembly

| Name | Type | Status | Default Value | Possible Values | Description |
|---|---|---|---|---|---|
| depth | real | mandatory | - | [0 ; ∞] | Defines the size of the model on the Y-axis. |
| width | real | mandatory | - | [0 ; ∞] | Defines the size of the model on the X-axis. |
| height | real | mandatory | - | [0 ; ∞] | Defines the size of the model on the Z-axis. |

## Specific Product Parameters

| Name | Type | Status | Default Value | Possible Values | Description |
|---|---|---|---|---|---|
| box | product | recommended | - | - | Dedicated parameter for client Range integration |
| damper | product | mandatory | - | - | Dedicated parameter for client Range integration |
| front | product | mandatory | - | - | Dedicated parameter for client Range integration |
| handle | product | recommended | - | - | Dedicated parameter for client Range integration |
| hinge | product | mandatory | - | - | Dedicated parameter for client Range integration |
| leg | product | recommended | - | - | Dedicated parameter for client Range integration |
| shelf | product | recommended | - | - | Dedicated parameter for client Range integration |

| | | | | | |
|---|---|---|---|---|---|
| type<br>(in case of doors) | integer | recommended | 1 | 1 or 2 | Specifies whether the door is a standard one or has an integrated handle. |
| type<br>(in case of handles) | integer | recommended | 1 | 1/2/3 | Specifies the position of the handle on the front. |
| worktopThick | real | recommended | - | [0 ;∞] | Enables the Range Manager to use the thickness of the worktop to calculate the position of other products in the cabinet assembly (e.g. sinks or cooktops). |
| worktopThickness | string | mandatory | - | - | Defines the size of the worktop. Also used to filter products in the Kitchen Planner. |
| worktopType | string | mandatory | - | Precut / custom | Defines the type of worktop, also used as filter in the Kitchen Planner. |

### Sub-Assembly Parameters

Box & Legs Sub-Assembly

| Name | Type | Status | Default Value | Possible Values | Description |
|---|---|---|---|---|---|
| boxAssembly | product | recommended | - | - | Dedicated parameter for client Range integration on assembly level. |
| legHeight | real | recommended | - | - | Dedicated parameter for client Range integration to change the height of the leg. This will change the vertical position of the box at the same time. |
| rightLegOption | integer | mandatory | 1 | 0/1/2 | Allows BR to enable, disable or move legs to ensure the right number of legs in a linear combination of cabinets. |
| leftLegOption | integer | mandatory | 1 | 1/2/3/4 | Allows BR to enable, disable or move legs to ensure the right number of legs in a linear combination of cabinets. |

Front & Handle Sub-Assembly

| Name | Type | Status | Default Value | Possible Values | Description |
|---|---|---|---|---|---|
| doorAssembly | product | recommended | - | - | Dedicated parameter for client Range integration at assembly level. |
| side | real integer | recommended | -1 | -1 (Left); 1(Right) | Enables the customer to change the door opening side of the cabinet, which will change the position of the handle at the same time. |
| handleLayout | integer | recommended | 1 | 0/1 | Enables the customer to force centered handle position instead of right and left position managed by side parameter. |
| handleOrientation | integer | recommended | 1 | 0/1 | Enables the customer to change the orientation of the handle (available only if handleLayout is set to 1).<br>0 corresponds to horizontal and 1 to vertical. |
| function | string | mandatory | - | - | Contains commercial functional description for filtering in the catalog browser (e.g. “door & drawer”). Requires using parameter display fields for translations. |

Drawer Sub-Assembly

| Name | Type | Status | Default Value | Possible Values | Description |
|---|---|---|---|---|---|
| drawer | product | recommended | - | - | Dedicated parameter for client Range integration on assembly level. |
| function | string | mandatory | - | - | Contains commercial functional description for filtering in the catalog browser (e.g. “door & drawer”). Requires using parameter display fields for translations. |

| | | | | | |
|---|---|---|---|---|---|
| drawerLight | product | recommended | - | - | Dedicated parameter for client Range integration. |
| drawerFront | product | recommended | - | - | Dedicated parameter for client Range integration. |
| internDrawer | product | recommended | - | - | Dedicated parameter for client Range integration on assembly level. |

### Top-Assembly Parameters

Dimension Parameters

| Name | Type | Status | Default Value | Possible Values | Description |
|---|---|---|---|---|---|
| commercialDimensions | string | mandatory | - | - | Description of the dimensions to display in the Kitchen Planner (e.g. "W400 x D600 x H880"). Use parameter display fields for translation. |
| commercialWidth | integer | mandatory | - | - | Specifies a commercial width used as a filter to browse the catalog in the Kitchen Planner (e.g." 40 cm"). Use parameter display fields for translation. |
| commercialDepth | integer | mandatory | - | - | Specifies a commercial depth used as a filter to browse the catalog in the Kitchen Planner (e.g." 60 cm"). Use parameter display fields for translation. |

Worktop Parameters

| Name | Type | Status | Default Value | Possible Values | Description |
|---|---|---|---|---|---|
| worktopOption | boolean | mandatory | TRUE | TRUE / FALSE | Enables the customer to choose a worktop that is automatically generated by the Kitchen application (requires paths defined in the BMA). |

Cover Panel Parameters

| Name | Type | Status | Default Value | Possible Values | Description |
|---|---|---|---|---|---|
| coverPanelDepth | real | mandatory | 13 | [0 ;∞] | Defines the thickness of the cover panels, for cover panel building and cabinet placing. |
| backCoverPanel | product | mandatory | - | - | Dedicated parameter for client Range integration, when a cabinet is not placed against the wall. |
| backCoverOption | boolean | mandatory | FALSE | TRUE / FALSE | Allows the BR to enable the backside panel of a cabinet, if backCoverPanel [product] is set. |
| rightCoverPanel | product | mandatory | - | - | Dedicated parameter for client Range integration, when a cabinet is not placed against the wall. |
| rightCoverOption* | boolean | mandatory | FALSE | TRUE / FALSE | Allows BR to enable the right side panel of a cabinet, if rightCoverPanel [product] is set. |
| zRightCoverPosition | real | recommended | - | - | Enables the Range Manager to define the position where the right cover panel starts from the floor (on the Z-axis), if rightCoverPanel [product] is set. |
| leftCoverPanel | product | mandatory | - | - | Dedicated parameter for client Range integration, when a cabinet is not placed against the wall. |
| leftCoverOption* | boolean | mandatory | FALSE | TRUE / FALSE | Allows the BR to enable the left side panel of a cabinet, if leftCoverPanel [product] is set. |

| | | | | | |
|---|---|---|---|---|---|
| zLeftCoverPosition | real | recommended | - | - | Enables the Range Manager to define the position where the left cover panel starts from the floor (on the Z-axis), if leftCoverPanel [product] is set. |

Wall Panel Parameters

| Name | Type | Status | Default Value | Possible Values | Description |
|---|---|---|---|---|---|
| wallPanelOption | boolean | mandatory | FALSE | TRUE / FALSE | Allows the customer to enable and choose a wall panel (based on 'WallPanel' type). |
| wallPanelType | boolean | mandatory | - | Precut / custom | Defines the type of wall panel, also used as filter in the Kitchen Planner. |
| allowWallPanelAutoCompletion | boolean | mandatory | TRUE | TRUE | Allows BR to automatically set wall panel option to a cabinet, if this cabinet is added to a linear where a wall panel is already set (requires wallPanelOption [boolean]). |

Other Linear Parameters

| Name | Type | Status | Default Value | Possible Values | Description |
|---|---|---|---|---|---|
| type<br>(in case of plinths) | string | recommended | | OnFront / OnFrame / OnLeg | Specifies the position of the plinth on the cabinet. |
| plinthOption | boolean | mandatory | TRUE | TRUE / FALSE | Allows the customer to add a plinth (based on 'Plinth' type). |
| wallEdgeStripOption | boolean | mandatory | FALSE | TRUE / FALSE | Allows the customer to add a wall edge strip (based on 'WallEdgeSrip' type). |

| | | | | | |
|---|---|---|---|---|---|
| railOption | product | recommended | - | - | Allows the customer to add and define a suspension rail product. |
| sinkCutOutEdge | product | optional | - | External dbID | Allows cut out edges on the sink base cabinet. |

# APPENDIX 3 – ADVANCED INFORMATION

## Select the Good Software / Format Combination

Below is the list of all possible 3D modelling software, from the most compatible to the lowest.

| Software | Export format | Support information |
|---|---|---|
| **Solidworks 2018** | Native | 3DCloud Solidworks plug-in |
| **3dsmax 2016 x64** | DAE OpenCollada | Qualified using http://www.opencollada.org v1.4.1 |
| | GLB / GLTF version 2.0 | Compatible using https://doc.babylonjs.com/resources/3dsmax_to_gltf |
| **Blender 2.79b** | DAE OpenCollada | Compatible v1.4.1 |
| | OBJ + MTL | Compatible Any version |
| | GLB / GLTF version 2.0 | Compatible using https://doc.babylonjs.com/resources/blender_to_gltf<br>As .glb or .gltf + .bin + textures. Self embedded GLTFs are not supported. |
| **Maya 2019** | GLB / GLTF version 2.0 | Compatible using https://doc.babylonjs.com/resources/maya<br>As .glb or .gltf + .bin + textures. Self embedded GLTFs are not supported. |
| **Sketchup** | | Not supported |
| **AutoCAD** | | Not supported |

## Analyze the Supported Features

OBJ Format

| Meshes & Primitives Features | Support | |
|---|---|---|
| 3 points faces | | Full |
| 4 points faces | | Partial<br>Caution: The conversion process might alter concave polygons. |

| | | |
|---|---|---|
| >4 points faces | ⭕ | None |
| Color/Dissolve interpolation | ⭕ | None |
| Free forms | ⭕ | None |
| Groups | 🟢 | Full |
| Level of details | — | Ignored |
| Lines | ⭕ | None |
| Normals | 🟢 | Full |
| Points | ⭕ | None |
| Shadow and Ray casting | — | Ignored<br>The Kitchen planner already computes shadows and reflections. They cannot be forced by the models data. |
| Smoothing groups | ⭕ | None |
| Texture coordinates | 🟢 | Full |
| Vertex | 🟢 | Full |

GLTF / GLB Formats

| **Meshes & Primitives Features** | **Support** | |
|---|---|---|
| Joints | ⭕ | None |
| Lines | ⭕ | None |
| Line loops | ⭕ | None |
| Line strips | ⭕ | None |
| Normals | 🟢 | Full |
| Points | ⭕ | None |
| Positions | 🟢 | Full |
| Tangents | ⭕ | None |
| Texture coordinates | 🟢 | Partial<br>Caution: Only TEXTCOORD_0 and TEXTCOORD_1 are taken into account. |

| | | |
|---|---|---|
| | | All material textures are applied to the texture coordinates of index 0.<br>The texture coordinates of index 1 can additionally reference the occlusion map. |
| Triangles | | Full |
| Triangle fans | | Full |
| Triangle strips | | Full |
| Vertex colors | | None |
| Weights | | None |

DAE (OpenCollada)

| **Geometries Features** | **Support** | |
|---|---|---|
| Basic meshes | | Full |
| Convex meshes | | None |
| Splines | | None |
| Basic vertices | | Full |
| Control vertices | | None |
| Material Binding | | Partial<br>Only the "technique_common" tag is taken into account. |

| **Primitives Features** | **Support** | |
|---|---|---|
| Lines | | None |
| Linestrips | | None |
| Polygons | | Partial<br>Caution: The conversion process might alter concave polygons. |
| Polylist | | Full |
| Triangles | | Full<br>Supported inputs are VERTEX, NORMAL and TEXTCOORD |
| Trifans | | None |

# 3DVIA Documentation

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