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DigitalTwin@Education Module 150-004
Creating a Static 3D Model Using the NX CAD System

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Table of contents

List of figures

List of tables

Creating a Static 3D Model Using the NX CAD System

1 Goal

Now that you have had the opportunity in the preceding modules to take a close look at the automation engineering aspects, the focus now turns to modeling and preparing your own 3D models.

In this module, you will use the NX CAD system from Siemens to independently create an initial static model of the sorting plant. This will give you the opportunity to become familiar with the basic operations and functionalities of NX.

2 Requirement

No prior knowledge is required for this module. However, to better understand the following description, it is recommended that you have prior experience with the sorting plant model. For more detailed descriptions of the layout and functionality of the sorting plant, please refer in particular to Module 1 of this workshop series.

3 Required hardware and software

The following components are required for this module:

1 **Engineering station**: Requirements include hardware and operating system (for additional information: see Readme on the TIA Portal Installation DVDs and in the NX software package)

2 **NX software with Mechatronics Concept Designer add-on** – V12.0 or higher

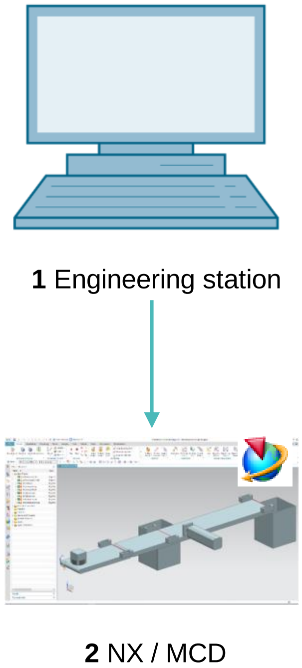


Figure 1: Overview of required software and hardware components in this module

Figure 1 clearly shows that the engineering station is the only hardware component of the system. The remaining components are based exclusively on software.

4 Theory

4.1 Static 3D model

To create a digital twin, it is essential to first have a suitable 3D model. This can arise from the design of a future plant or be derived from an existing plant that is to undergo future expansion. The model can comprise either a complete plant or only individual sections of a plant.

As already noted in Module 1 of the DigitalTwin@Education workshop series, the level of detail in the 3D model is extremely important for the quality of a digital twin. The more detailed the model, the more similar its behavior will be to a system in reality. But it is also true that the effort and computing capacity required increases significantly as the level of detail increases. Before creating the 3D model, it is therefore important to clearly specify which tasks and functions of the plant or components to be designed are to be covered. Only then is a realistic estimate of the effort possible.

A conventionally created CAD model is referred to as static 3D model. Here, static means that no dynamic properties have been incorporated in the model. Examples of dynamic properties of a body are gravitation and its reaction to forces. A simulation, as was carried out in the preceding modules, is therefore not possible with a purely static 3D model. However, a static model is always required as the starting point for dynamizing a digital model. For this reason, you need to also start by creating this static model.

4.2 Modeling in NX

Modeling of 3D models in NX is based on two different forms:

- Model
- Assembly

A model is always a self-contained individual component of a subsystem or overall system. When preparing a model, usually the first step is to digitally create a 2D sketch. This sketch must be associated with a datum plane. A datum plane designates the orientation in three-dimensional space. Simple and established planes are between the X and Y axes, the Y and Z axes and the X and Z axes. These are illustrated in Figure 2.

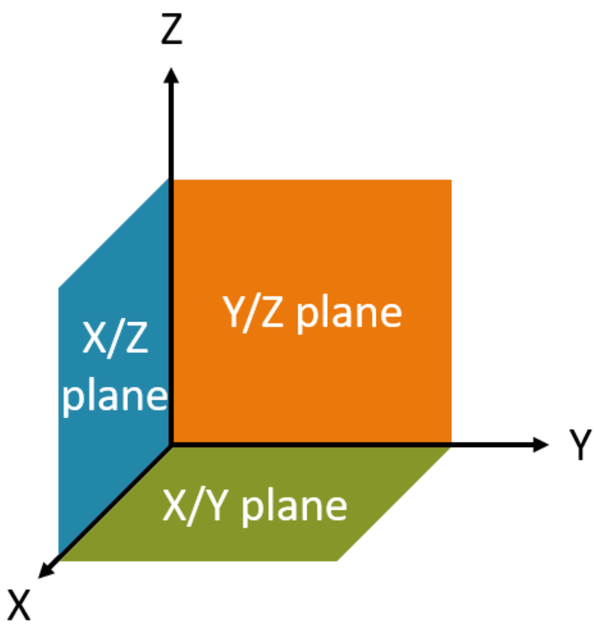


Figure 2: Standard reference planes in NX

If desired, it is also possible to define specific reference planes that follow a different orientation. After the two-dimensional sketches are complete, they are embedded into the corresponding plane and converted to a three-dimensional body by forming. NX has many forming options, such as extruding as well as forming by rotating. Only the functions relevant for the sorting plant will be used in this module.

Several models can be merged to form a subsystem – an assembly. The overall system is thus created by merging the assemblies, possibly along with other models. The orientation and placement in three-dimensional space is also important in this regard. The resulting 3D model can be used later for the dynamization.

The NX tool is more than a 3D CAD system. Rather, it allows the combined use of different applications in a single user interface. These include the "Modeling" and "Mechatronics Concept Designer" applications.

In NX, the complete modeling is carried out in the "Modeling" application, as shown in Figure 3.

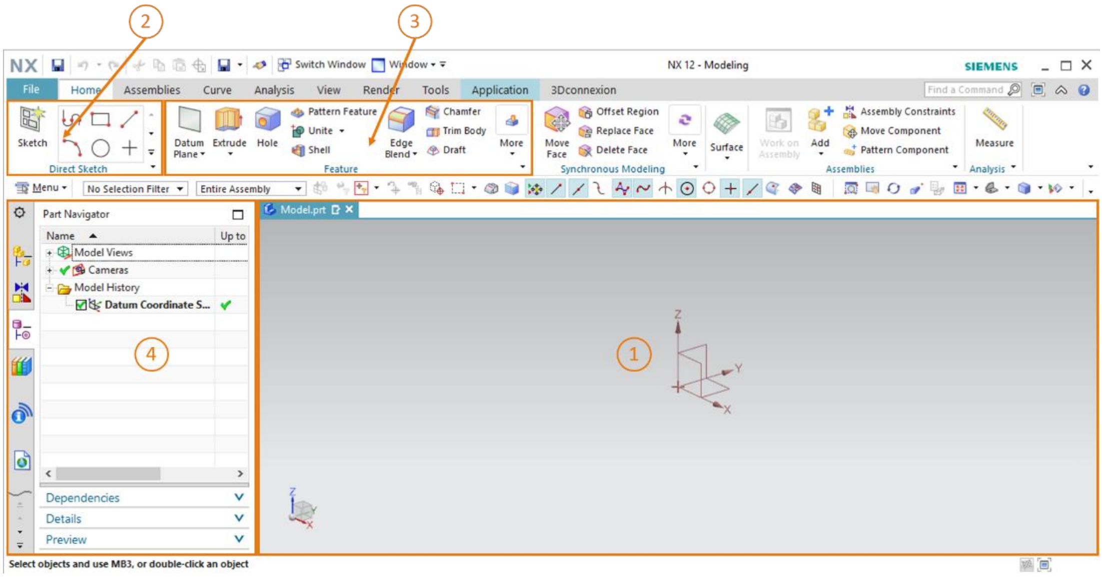
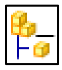


Figure 3: "Modeling" application in NX with markings for explanations of the areas in the text

In modeling mode, four windows are used for creating the sorting plant:

- The central screen (see Figure 3, area 1) contains the three-dimensional graphics window. Here, all the needed modeling steps are performed in both two-dimensional and three-dimensional space.
- The left part of the menu bar (see Figure 3, area 2) contains all the tools needed to prepare two-dimensional sketches.
- The center part of the menu bar (see Figure 3, area 3) lists all the form elements. For one thing, these allow you to create three-dimensional models from two-dimensional sketches. They also allow you to further model three-dimensional models, for example by rounding edges.
- The Resource bar (see Figure 3, area 4) can display the model history for the purpose of tracking the modeling steps performed. In the case of an assembly, the various individual components can also be listed here.

Both models and assemblies are saved as parts with file extension "**.prt**" in NX. So that you can easily distinguish the modeling form of an open file, you should settle on a clear naming convention when selecting names.

In case of uncertainty, you can open the "Assembly Navigator" menu  for an open file within the NX "Modeling" application. You can see the following distinction here:

- Models are always provided with the symbol . They can only consist of an individual part (see Figure 4, left side).
- Assemblies have the symbol . They can be composed of multiple models and assemblies (see Figure 4, right side).

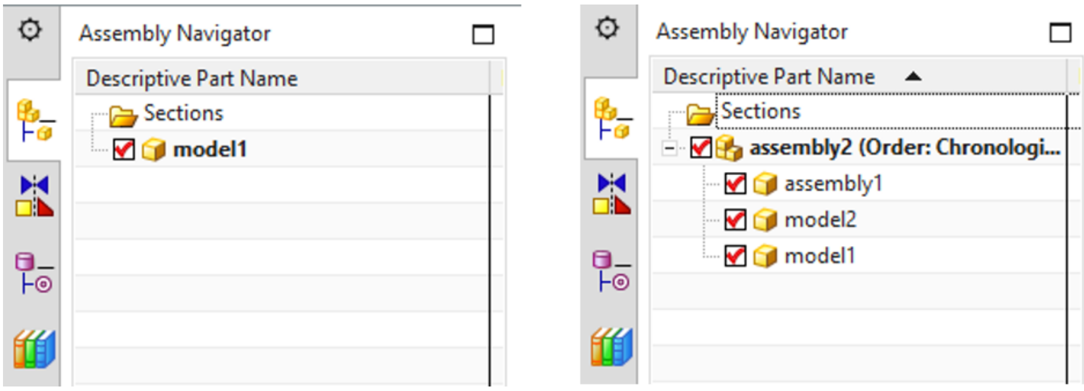


Figure 4: Differentiation of models (left) and assemblies (right) in the Assembly Navigator

5 Task

In this module, you are to independently create the static 3D model of the sorting plant that you have used previously in the preceding modules.

This involves, first of all, the modeling of various individual components of the sorting plant in NX through use of various basic functionalities of the "Modeling" application. The individual components you create, together with provided models, are then to be merged and correctly positioned to form an assembly.

6 Planning

The static 3D model requires at least Version **V12.0** of the **NX** CAD system.

For proper understanding of the individual components to be created, you should be familiar with the sorting plant from the first three modules of this workshop series. In case of uncertainty, you should refer in particular to the theory section Chapter 4.2 from **Module 1**.

For the naming conventions of the individual models, you can continue using the "**Guide to Standardization**" from Siemens. You will find this in Chapter 9 by selecting the link [1].

Programming of the PLC, visualization and creation of a virtual PLC for simulation purposes is not necessary in this module.

7 Structured step-by-step instructions

The project "**150-004_DigitalTwinAtEducation_NX_statModel**" is made available with this module. You will find two folders in this project:

- "**ComponentsToImport**" contains important components for Chapter 7.1.8.
- "**fullStatModel**" contains the solution for this module in case you need help completing a step.

Perhaps the most important functionality, which you should make frequent use of, is the Command Finder. This is located in the upper right part of the NX user interface screen, as shown in Figure 5.

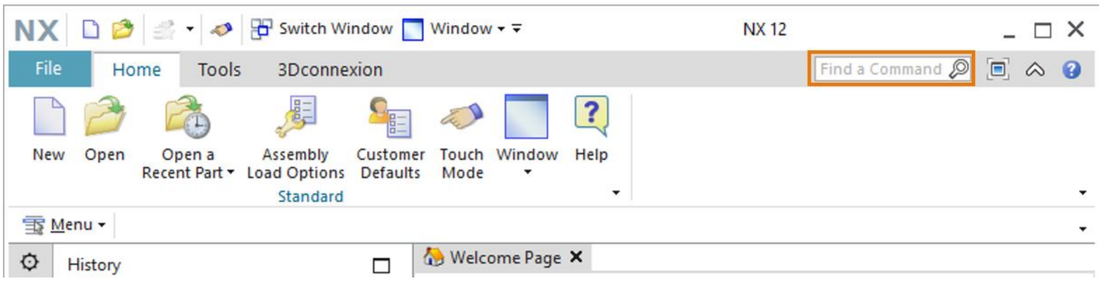


Figure 5: Command Finder in the NX menu, highlighted in orange

You can use this to search the complete command library of NX and all the associated add-ons and applications. You can choose the appropriate command from the search results. NX also shows you where, the command is located so that you can also select it directly from the menu in the future.

IMPORTANT: The user interface and the arrangement of various commands in the menus change with new versions of NX. In addition, users can define their own user interfaces. While the following descriptions depict the standard user interface of NX12.0, the interface in your version may be different. **For this reason, if you are unable to find a command in the window at the positions described, use the Command Finder.**

Also bear in mind that this description represents only a suggested solution. There, are countless ways of designing 3D models in NX. The aim here was to describe a procedure that is easy to follow. Of course, you can also try out various other ways here.

Note that certain passages in this module are labeled as "Sections". Because frequent reference is made to these passages in the course of the description, this labeling is intended as an orientation guide.

7.1 Modeling of all the individual components for the sorting plant

In this chapter, the individual components of the sorting plant are to be modeled as self-contained models in NX.

To create a model, you need to perform the following three steps:

1. First, ensure that the "**NX V12.0**" software is installed and open. If that is not the case, look for the application in the Start menu or on your desktop. After starting the software, you should arrive at the Welcome Page of NX, as illustrated in Figure 6.

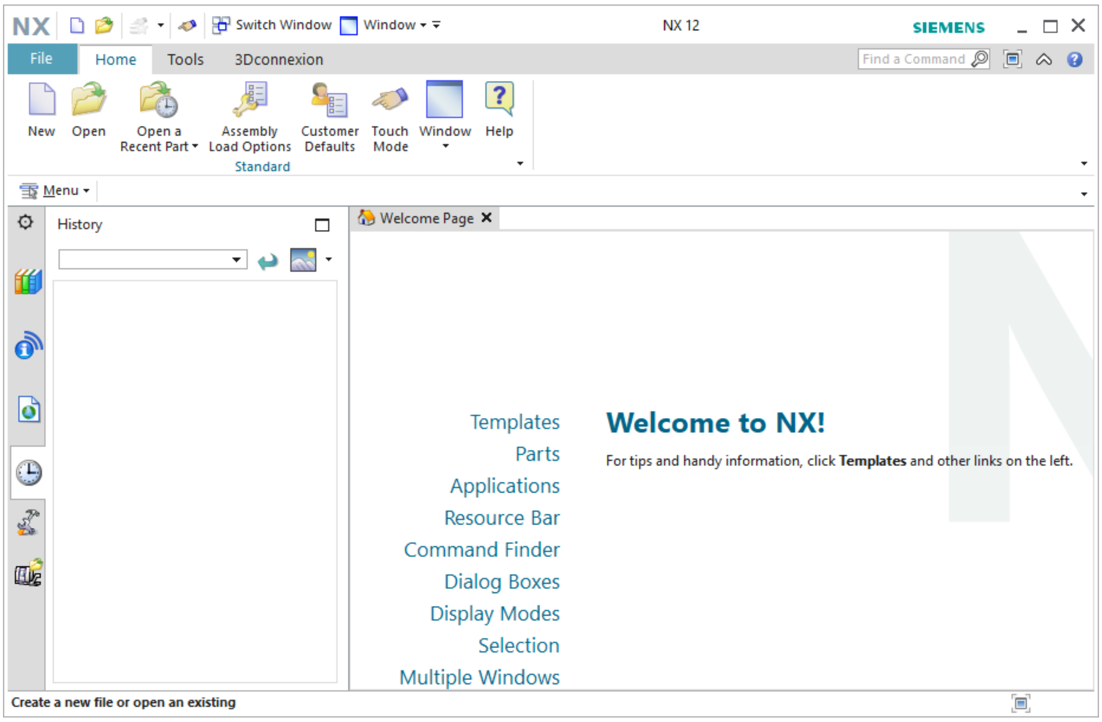


Figure 6: Welcome Page of the NX software

NOTE

If the user interface language is not your preferred language, the only way to change it for NX12.0 is in the **environment variables** of your operating system. Under Windows 10 you will find these in Control Panel → System → Advanced system settings → "Advanced" tab → Environment Variables…

There, you will find the value "**UGII_Lang**" in the system variables. Specify your preferred language using English language (e.g. **German** or **English**).

Section: Creating a model

2. Next, you create a new model. To do so, first press the "**New**" button (see Figure 7, step 1). The window for creating new modeling data now opens. Select the "**Model**" tab in this window (see Figure 7, step 2). You will now find various kinds of models you can create. Select a simple **model** (see Figure 7, step 3). Because this is a "Modeling"-type model, the NX application "Modeling" will open automatically after the model is created. Next, assign an appropriate file name for the model, always with file extension ".prt", and choose a working directory (see Figure 7, step 4). Click the "**OK**" button to create the new model (see Figure 7, step 5).

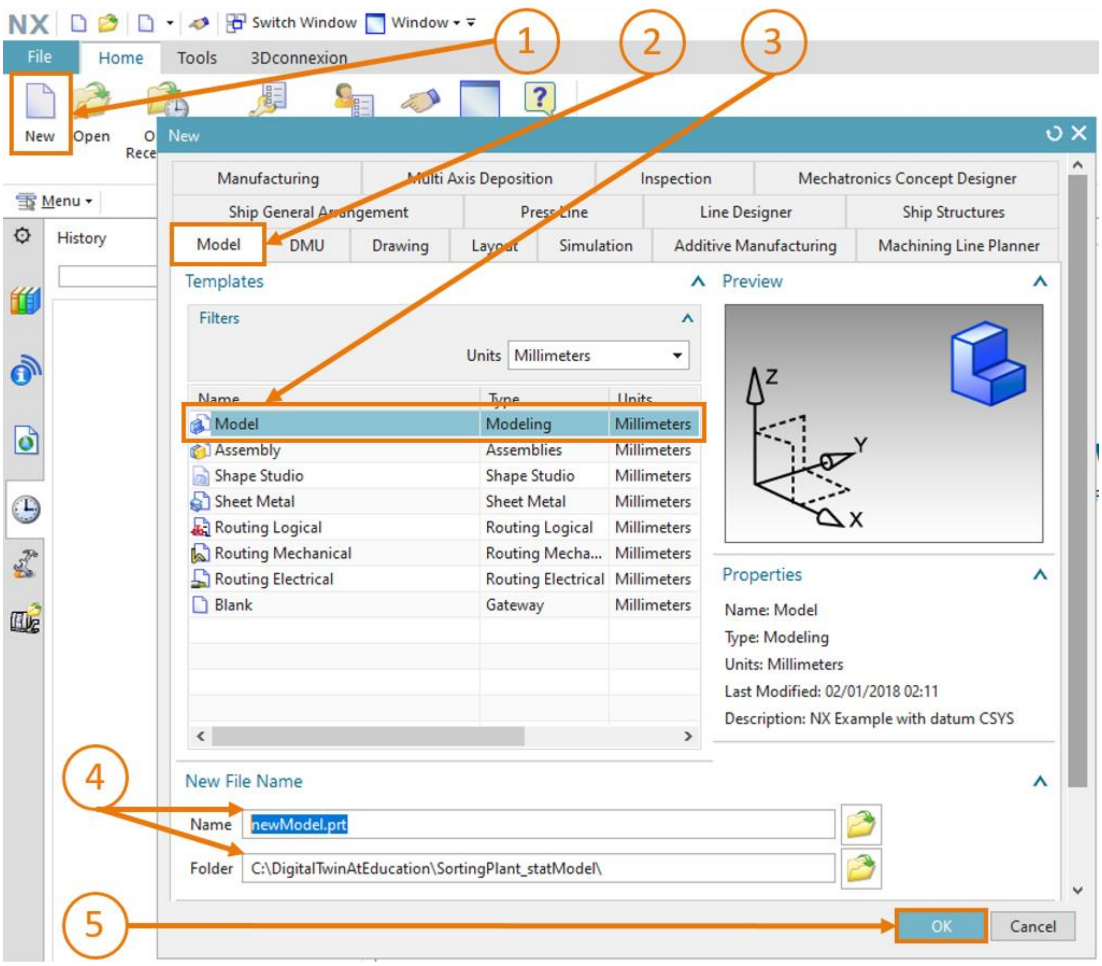


Figure 7: Creating a new model in NX

3. You are now in the "**Modeling**" application of NX, as you can see from the program heading (see Figure 8, highlighted in orange). Before you begin the modeling, make sure that you are in the trimetric view. The view orientation specifies the perspective from which objects are represented in NX (for example, from above, from the side, trimetric, etc.) The best way to change the view is to use the Command Finder, as described at the beginning of the chapter. Select the "**Trimetric**" view in this way, (see Figure 8).

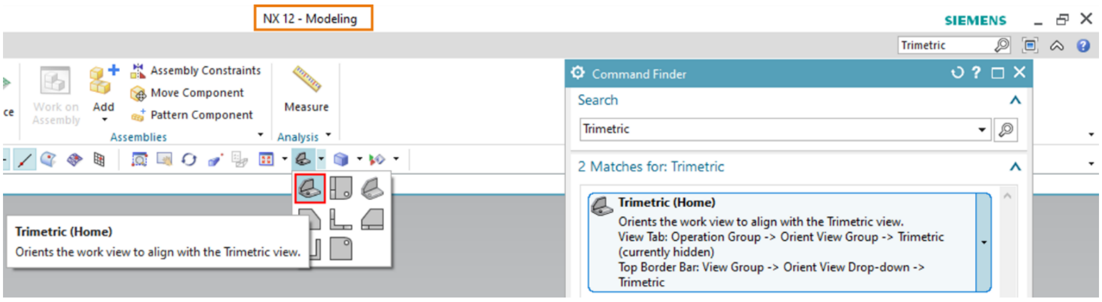


Figure 8: Selecting the "Trimetric" view in NX

NOTE

You can also switch between various NX applications using the Command Finder.

For example, you could perform a search for "Modeling" and start this application via the selection list.

7.1.1 Modeling the cube workpiece

Two workpieces are to be sorted with the sorting plant. One of these workpieces is in the shape of a cube. The cube is to have the following characteristics:

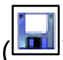
- The length of each side is to be 25 mm.
- Because it is a cube, all surfaces of the workpiece are square and equal in size.
- The cube is to use the X/Y plane as the datum coordinate system.

To create this model in NX, follow these steps:

Section: Creating a sketch

Creating a sketch in NX:

→ Create a new model following the description in Chapter 7.1, "**Section: Creating a model**".

Name the model "**workpieceCube**". Save the file by clicking the Save symbol () or selecting the corresponding menu command "File " → Save.

→ You begin creating a new sketch in two-dimensional space. To do this, click the "Sketch" button, as shown in Figure 9, step 1. The "Create Sketch" window opens. Here, you should have set the following parameters:

- Sketch Type = On Plane
- Sketch CSYS:
  - Plane Method = Inferred
  - Reference = Horizontal
  - Origin Method = Specify Point

→ Then select "**Specify CSYS**" in the window (see Figure 9, step 2). Open the associated selection list (see Figure 9, step 3) and select the "**Inferred**" method (see Figure 9, step 4).

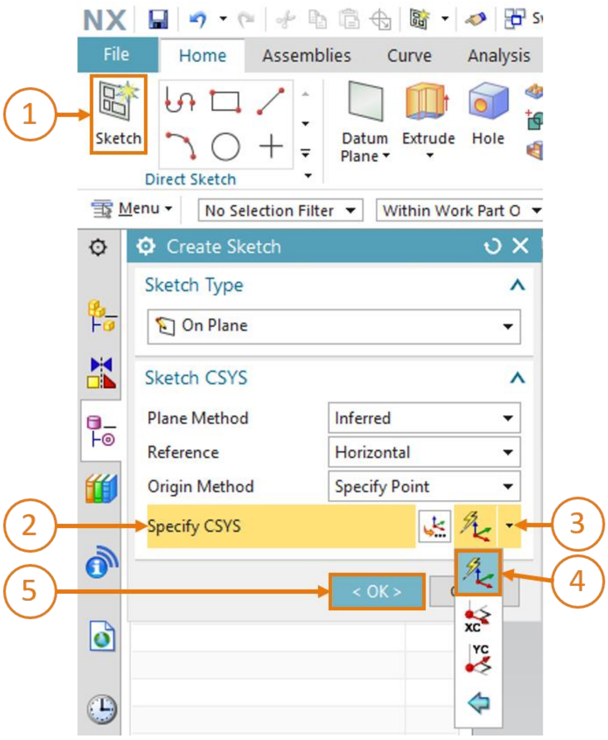


Figure 9: Creating a sketch in NX – Part 1

NOTE

Whenever you need help on the command window and parameters associated with a command, select this and then press the "**F1**" key to be taken to the corresponding help window where you will find information.

Note that you need to have an Internet connection to use the help function.

→ Select the X/Y plane in the graphics window, as indicated in Figure 10, step 1. This plane should now change from blue to orange. An orange orientation arrow in the positive direction along the Z axis also appears in this case. The plane for drawing the sketch is thereby selected, and all you have to do is confirm by clicking the "**OK**" button in the "Create Sketch" window (see Figure 9, step 5).

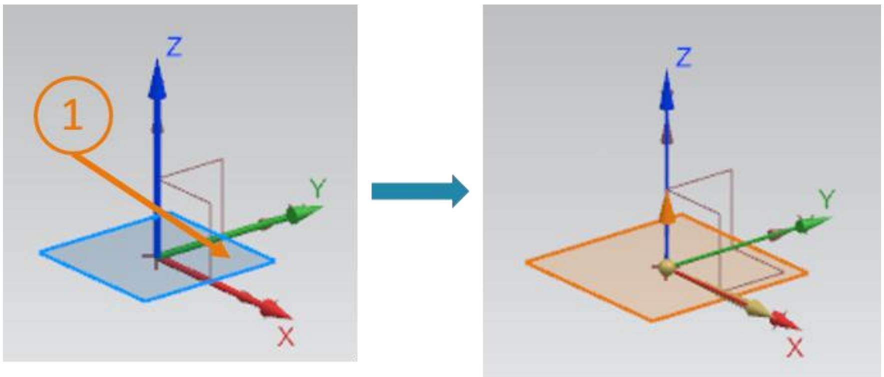


Figure 10: Creating a sketch in NX – Part 2

You are now taken automatically to the X/Y view where, you can begin your sketch.

→ You have access to various sketching functions within a sketch. A portion of these are shown in Figure 11. You can select a variety of forms for creating or correcting sketches in the selection menu in the center of the window. On the left are functions for creating another sketch within this sketch, as described in the preceding steps, or finishing the sketch.

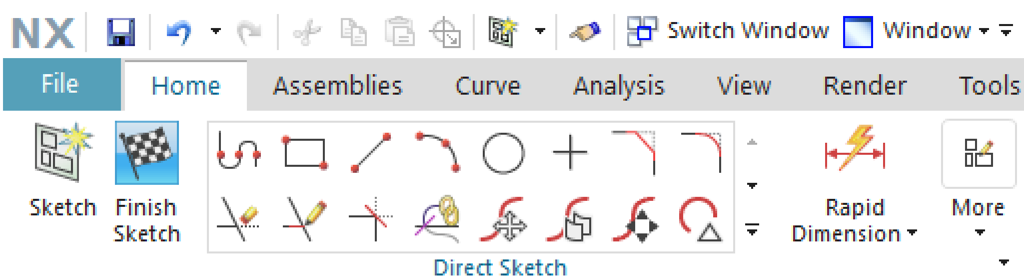
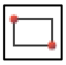


Figure 11: Sketching functions in NX

Sketching a square:

→ In order to create a cube, you need to first sketch a square. For this purpose, you select the **Rectangle** sketching function with the  button.

→ The "Rectangle" window opens on the screen. You can see in this window that more than one method is available for sketching a rectangle in NX. In this case, you are to sketch a "**Rectangle from 2 points**" (see Figure 12, step 1) with input of the "**X/Y coordinates**" (see Figure 12, step 2). Enter **XC = 0** and **YC = 0** as the coordinate values, as shown in Figure 12, step 3. Note that you need to confirm individual coordinate inputs with the Enter key.

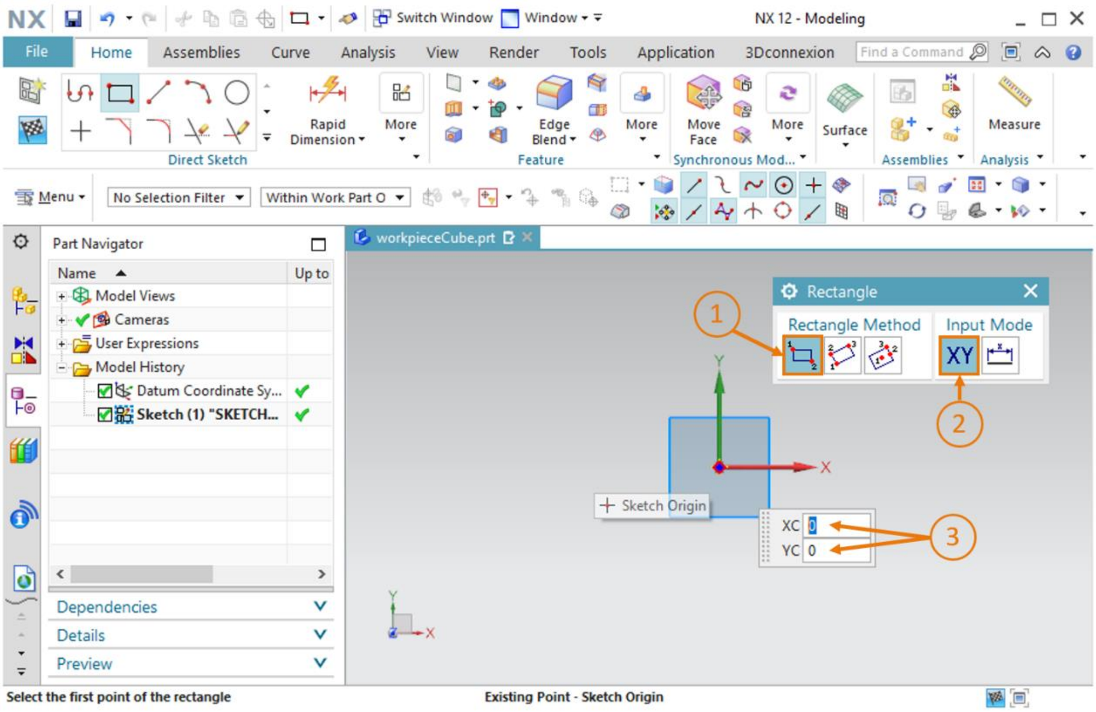


Figure 12: Preparing the sketch for the cube – Part 1

→ Once the first point of the square has been created, the second point must also be added. Enter **XC = 25** and **YC = 25** as the coordinate values for this (see Figure 13, step 1), and confirm the inputs once again with the Enter key. Be sure to select the "X/Y coordinates" input mode in this case as well. You should now see the square with side lengths of 25 mm (see Figure 13, right side). This completes the sketch for the cube.

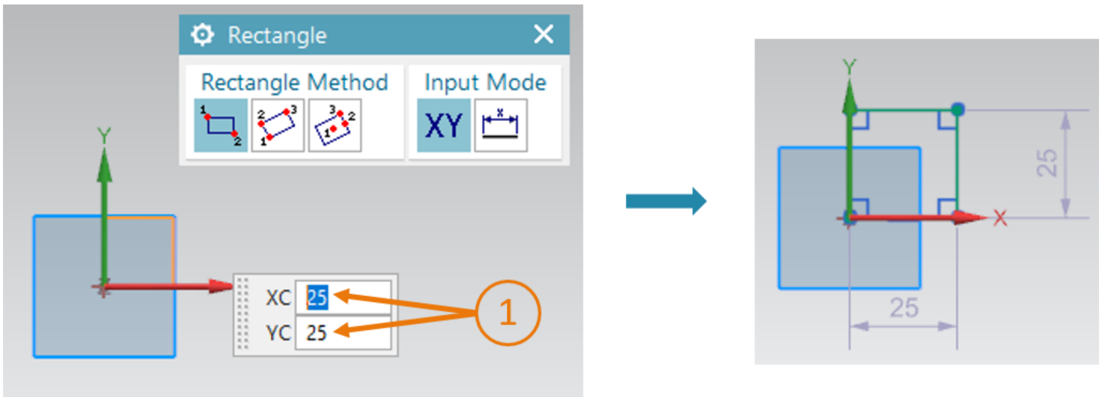
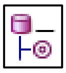


Figure 13: Preparing the sketch for the cube – Part 2

To finalize the sketch, click the "**Finish Sketch**"  button in the sketching functions of NX (see Figure 11). The sketch editor then closes, and you are taken back to the three-dimensional view.

Extruding a square into a cube:

→ To create a cube from the two-dimensional square, the sketched square must be converted to three-dimensional space using the "**Extrude**" function. For this purpose, click the "Extrude" function in the form elements menu bar (see Figure 14, step 1). The "Extrude" window opens, where, you specify the parameters for this operation. In this window, click on "**Select Curve**" under "Section" (see Figure 14, step 2).

In the **Part Navigator** , select the **sketch** you previously created in the **Model History** (see Figure 14, step 3). You should now see the future 3D model in the graphics window. Adjust the limit parameters, as highlighted in Figure 14, step 4, as follows:

- **Start = Value** with a distance of **0 mm**
- **End = Value** with a distance of **25 mm**

Confirm the operation with the "**OK**" button (see Figure 14, step 5).

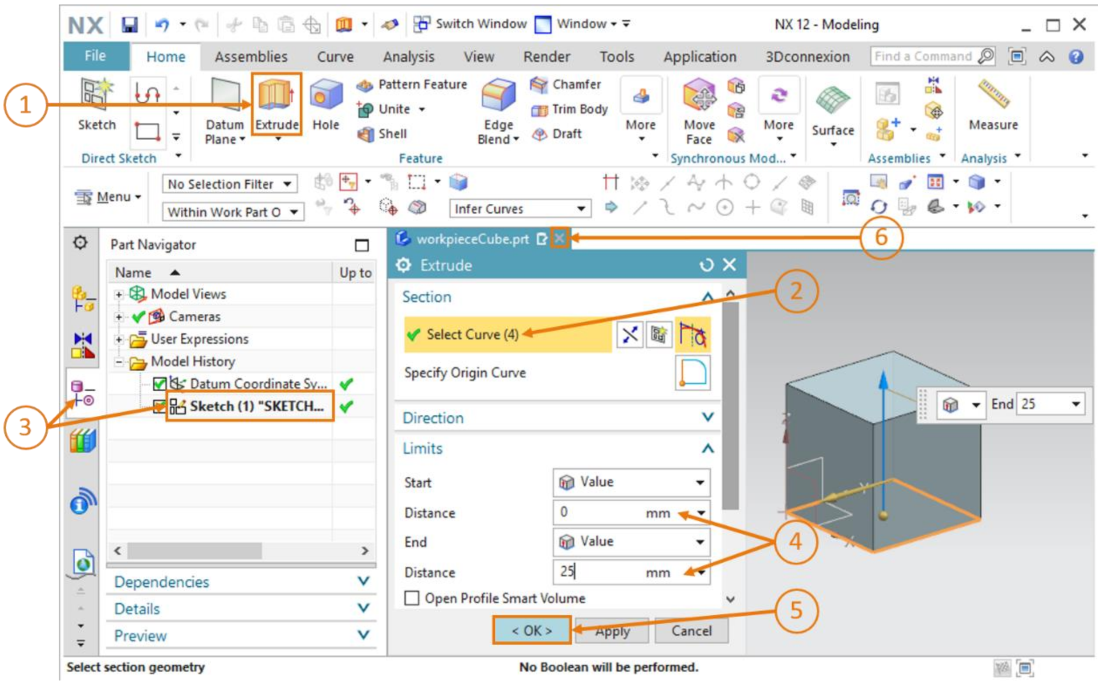


Figure 14: Extruding a square into a cube

You have now designed your first 3D model completely independently. Finally, switch back to the trimetric view, as described in Chapter 7.1, step 3, and save your model. Close the model by clicking on the "X" button in the model selection bar (see Figure 14, step 6).

7.1.2 Modeling the cylinder workpiece

The second workpiece of the sorting plant is be a cylinder with the following features:

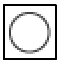
- The cylinder is to have a diameter of 30 mm and a height of 10 mm.
- The center of the circle of the surface is positioned at the origin of the coordinate system.
- The X/Y plane is to be selected as the datum coordinate system.

The following steps can be used to design this 3D model:

Creating a sketch in NX:

→ For this, you follow the same steps as described previously in Chapter 7.1.1, "**Section: Creating a sketch**". However, this time you save the model under the name "**workpieceCylinder**".

Sketching a circle:

→ The base of a cylinder is a circle. You must therefore insert circle in your sketch. Select the **Circle** button  in the sketching function selection menu (see Figure 15, step 1). A new "Circle" window appears. This offers several options for sketching a circle. Select "**Center point and diameter**" as the circle method (see Figure 15, step 2), activate the **coordinate mode** for entering the center of the circle (see Figure 15, step 3) and specify the coordinates **XC = 0** and **YC = 0** (see Figure 15, step 4). You must confirm these inputs by pressing the Enter key.

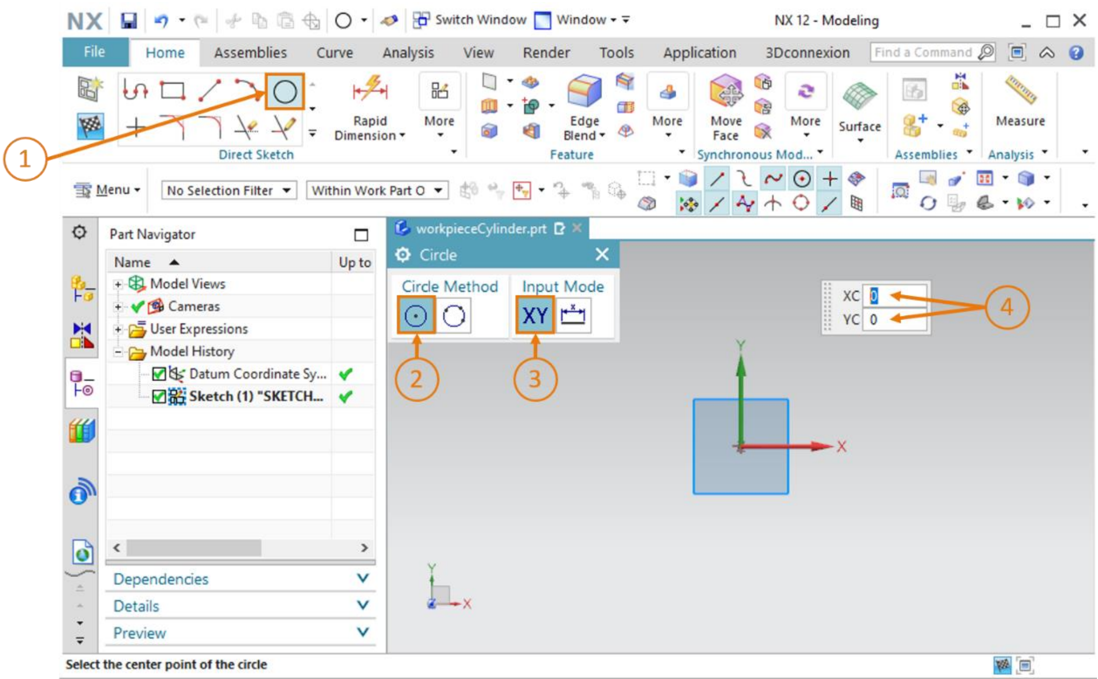


Figure 15: Preparing the sketch for the cylinder – Part 1

→ Switch to the **parameter mode** for selecting the diameter and specify a value of **30 mm** (see Figure 16, steps 1+2). Once again, confirm your entry by pressing the Enter key. You should then see the sketch of your circle in the graphics window. You can recognize this from the indication of the diameter, as shown on the right side in Figure 16.

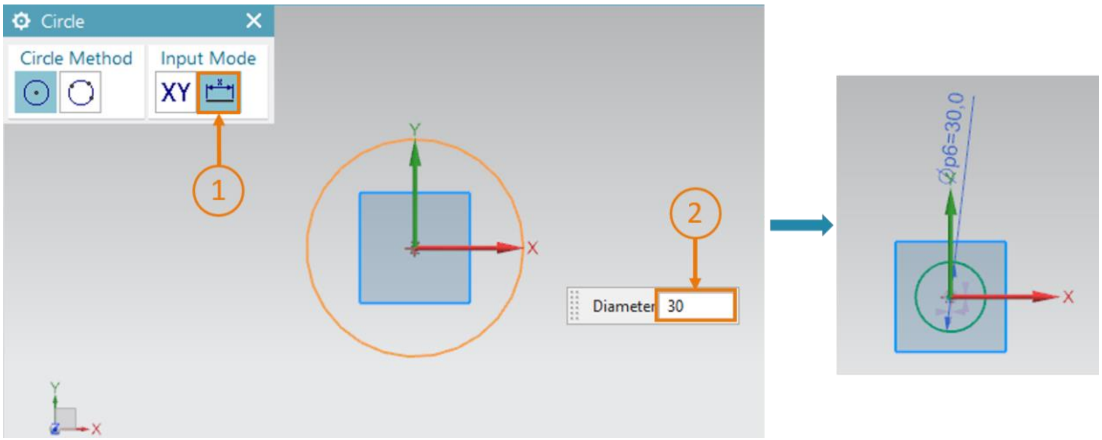


Figure 16: Preparing the sketch for the cylinder – Part 2

→ Finalize the sketch by clicking the "**Finish Sketch**" button  in the Sketching functions of NX (see Figure 11). This closes the sketch editor and returns you to the three-dimensional view.

Extruding a circle into a cylinder:

→ For generating a cylinder from a circle, you will also use the "**Extrude**" function. The workflow used for generating the cube in Chapter 7.1.1 (see Figure 17, steps 1-5) is also used here, except that a value of **10 mm** is specified here for the end distance (see Figure 17, step 4).

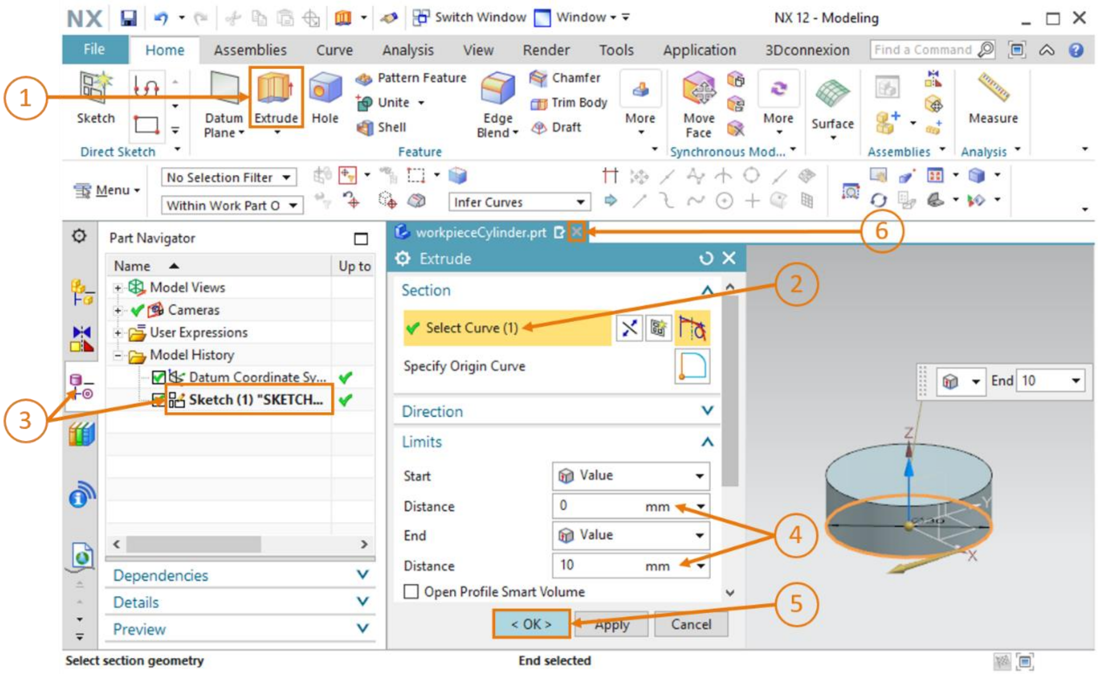


Figure 17: Extruding a circle into a cylinder

→ This completes the modeling of the 3D model for the cylinder workpiece. Switch back to the trimetric view, as described in Chapter 7.1, step 3, save your model and close the model via the model selection bar (see Figure 17, step 6).

7.1.3 Modeling the "ConveyorShort" conveyor belt

Conveyor belts are needed for transporting the workpieces during the sorting process. In this chapter, you are to create the first, shorter conveyor belt "ConveyorShort". This will transport the workpieces you have modeled in Chapters 7.1.1 and 7.1.2 to the sorting process. The "ConveyorShort" conveyor is a body with the following properties:

- The conveyor belt has a length of 150 mm, a width of 65 mm and a height of 10 mm.
- The edges at both ends of the transport surface are rounded with a radius of 5 mm.

A procedure for creating the transport surface is described in the following:

Creating a sketch in NX:

→ The basic procedure for creating this sketch is very similar to the procedure described in Chapter 7.1.1, "**Section: Creating a sketch**". However, this sketch is to be prepared in the **X/Z plane** because the vertical side of the conveyor belt is to be sketched first. To select a coordinate system, move your mouse inside the graphics window to the X/Z plane of the datum coordinate system (see Figure 19, step 1). You can see that a new coordinate system with a different orientation is displayed in addition to the datum coordinate system of the model. A plane always lies on the X and Y axis of the associated coordinated system, and the Z axis runs perpendicular to this. Figure 18 shows the new coordinate system of the X/Z axis on the right side. Click on this area in order to select this plane for the sketch.

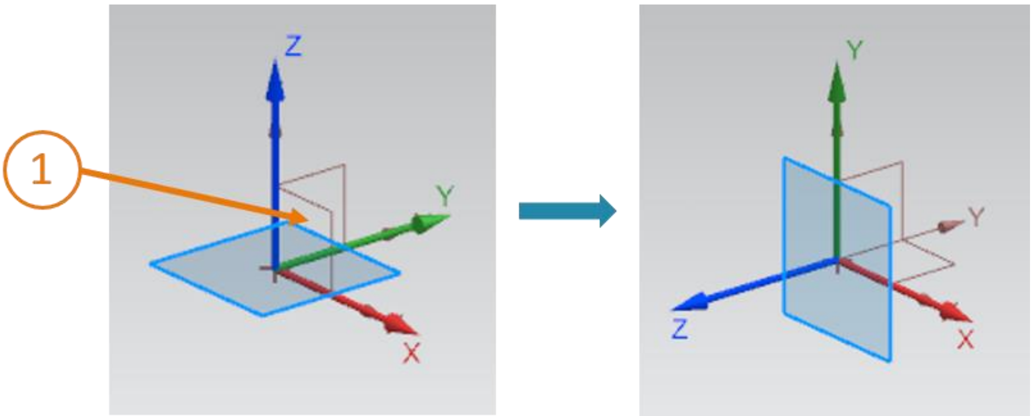


Figure 18: Selection of the X/Z plane for the conveyor belt

→ You should notice a color change of the plane from blue to orange. You should also see that the **orientation of the sketch** (represented by the orange arrow along the blue Z axis), determined on the basis of the specified datum coordinate system, runs in the negative Y direction. In order to orient the sketch in the positive Y direction, you need to double-click on the orange arrow in the graphics window (see Figure 19, step 1). You then see the changed course of the new Z axis in the positive Y direction. The sketch is now correctly configured, and you can click the "OK" button to create the sketch.

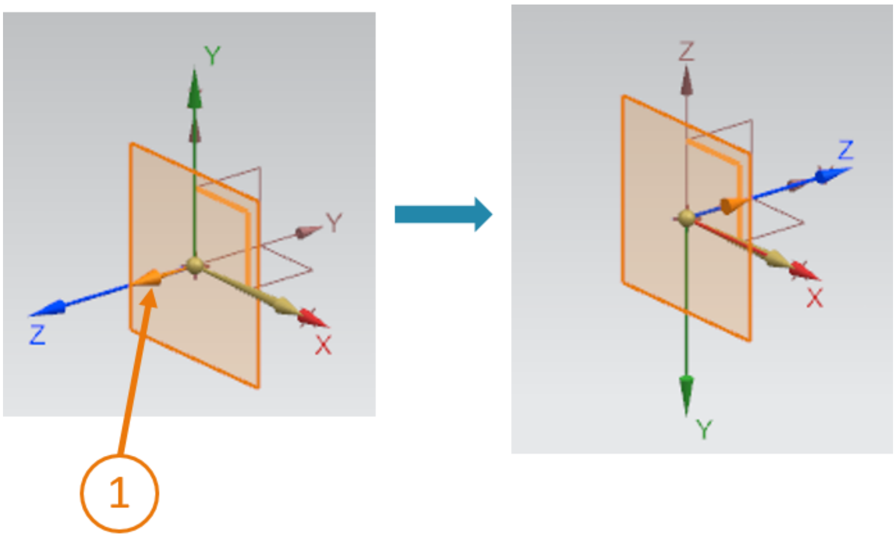


Figure 19: Adjusting the orientation of the coordinate system of the sketch

Sketching a rectangle:

→ Follow the same procedure here as described in Chapter 7.1.1. However, since you are to sketch the side of the conveyor belt, a rectangle with dimensions **65x10 mm** must be created in this case based on the conveyor properties. You can see from Figure 20 that the YC value is "-10". The negative sign is necessary because of the inverted orientation of the sketch when the sketch was created in the previous step. Close the sketch when you have completed it.

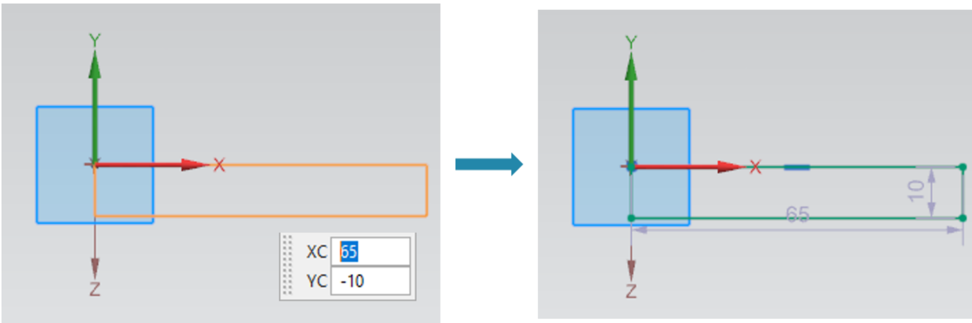


Figure 20: Preparing the sketch for a conveyor belt

Extruding a rectangle into a cuboid:

→ The conveyor belt is to have a length of **150 mm**. You can now use the "**Extrude**" function, as explained previously in Chapter 7.1.1, to create a cuboid from the rectangle (see Figure 21).

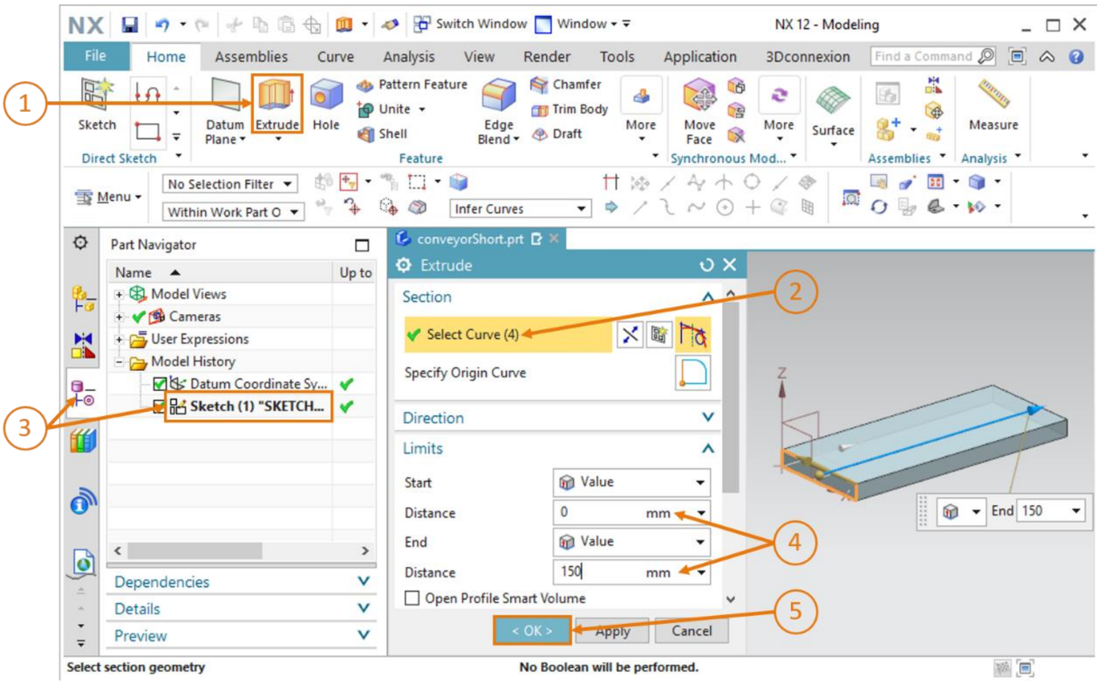
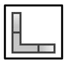


Figure 21: Extruding a rectangle into a cuboid

Rounding the edges of the cuboid:

→ To adapt the image of the body to a real conveyor belt, you are to round the front and rear edges of the cuboid. For this, you switch from the trimetric view to the "**Front**" view , as shown in Figure 22, step 1. If you cannot find this menu, use the Command Finder from Chapter 7.1. Click on the "**Blend Edge**" form element (see Figure 22, step 2) to open the corresponding parameter window. Leave the continuity set to "**G1 (Tangent)**", click on the "**Select Edge**" command and select, one after the other, the front edges of the body, top and bottom (see Figure 22, step 3).

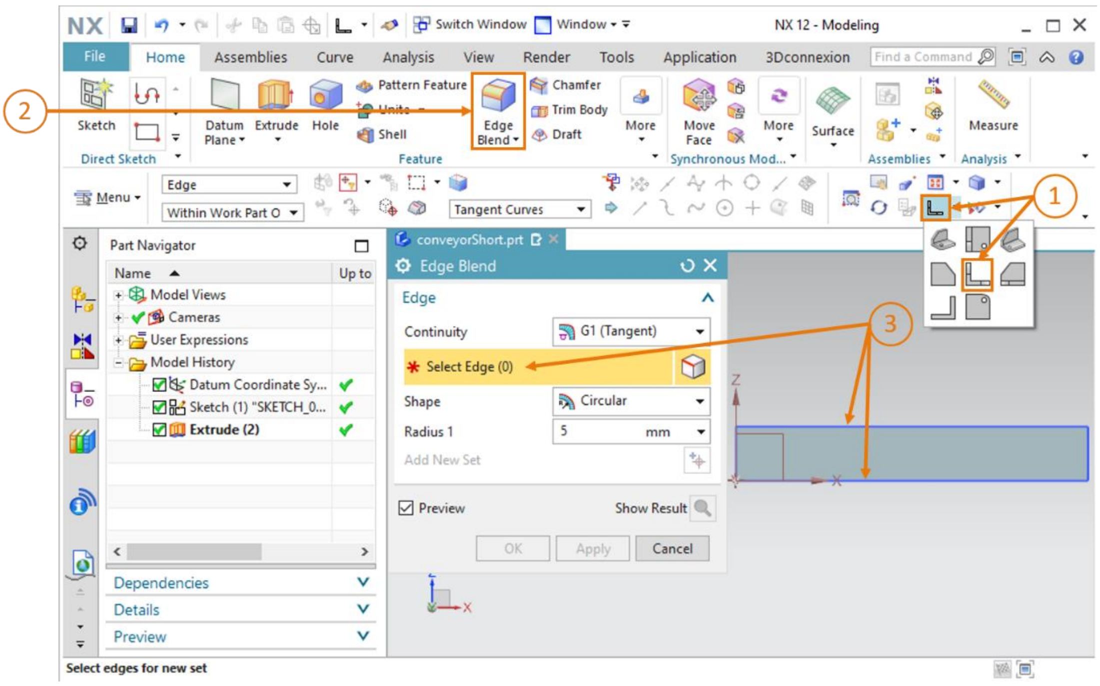


Figure 22: Edge rounding of the cuboid – Part 1

→ You should now see the start and end points of the rounding on both sides of the body (see Figure 23). You can also recognize this from the parenthesized information in the "Select Edge" line, which in this case is "**(2)**". This stands for both edges. Select "**Circular**" for the shape and **5 mm** for the radius, as indicated in Figure 23, step 1. Confirm your entry with "**OK**" (see Figure 23, step 2).

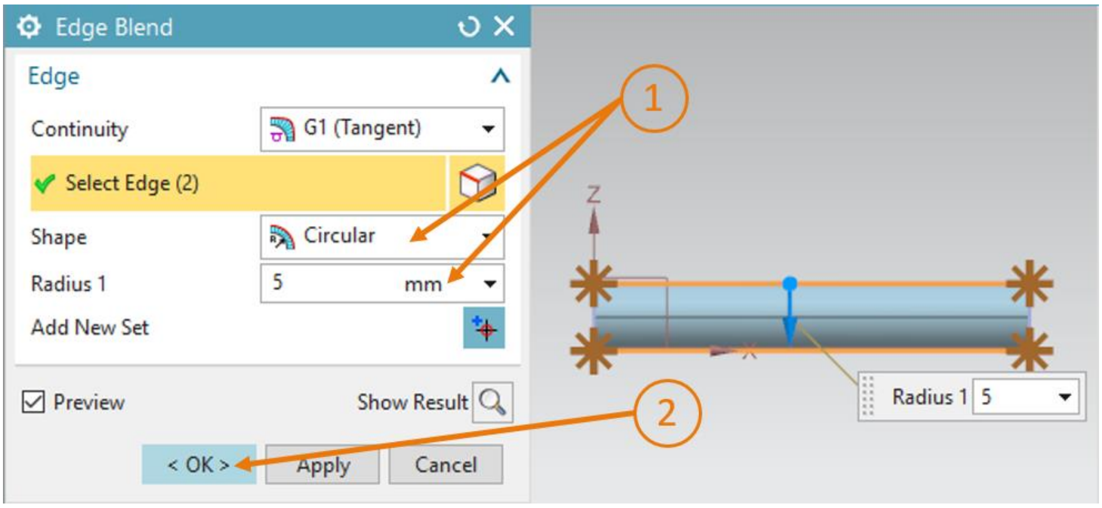
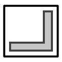


Figure 23: Edge rounding of the cuboid – Part 2

→ Follow the same procedure for the two rear edges. However, you need to first change the view to the "**Back**" view . You will find this function in the same submenu in which you previously selected the "Front" view. Of course, you can also search for it in the Command Finder.

→ The modeling of the transport surface is now complete. Switch back to the trimetric view to save and close the model (see Figure 24).

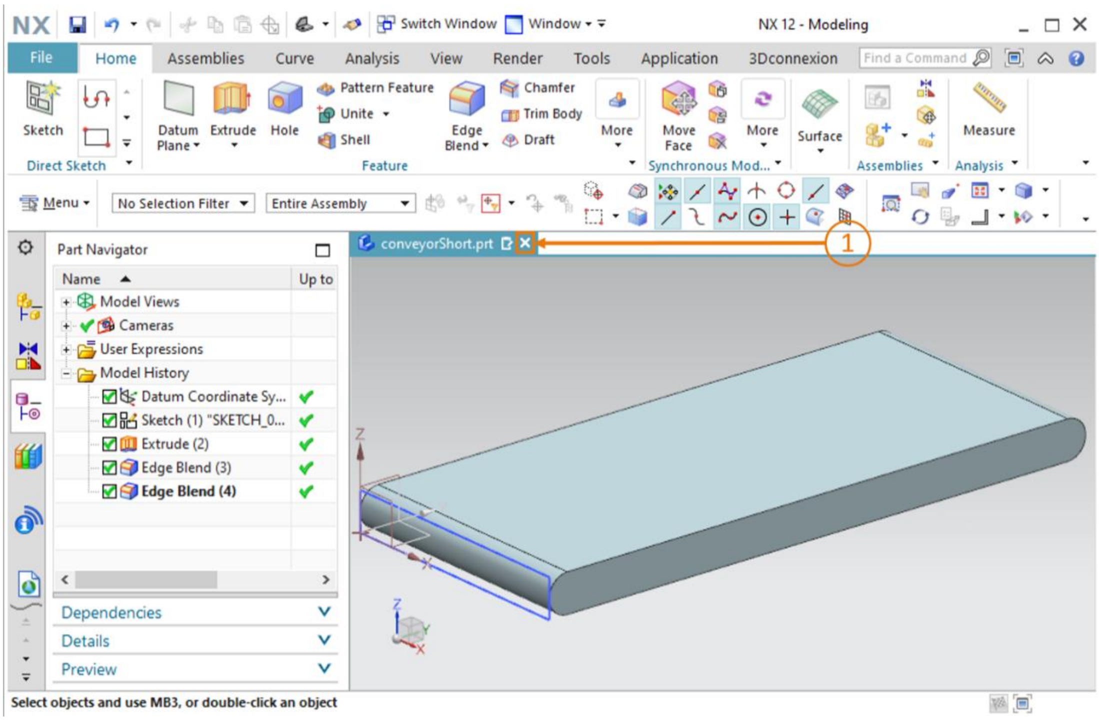


Figure 24: Closing the finished 3D model of the conveyor belt

7.1.4 Modeling the "ConveyorLong" conveyor belt

The second conveyor belt "ConveyorLong" is needed to forward the workpieces into the sorting process. "ConveyorLong" has the following parameters:

- The conveyor belt has a length of 390 mm, a width of 65 mm and a height of 10 mm.
- The edges at both ends of the transport surface are rounded with a radius of 5 mm.

You can see that the data largely corresponds to that of the "ConveyorShort" conveyor belt. Only the length of the conveyor belt must be adjusted.

Follow the same modeling procedure here as explained in Chapter 7.1.3. However, specify the file name "**conveyorLong**" when creating the model. You must now take the new length of **390 mm** into account when extruding.

7.1.5 Modeling a container

For sorting of the workpieces, containers are needed for receiving the parts. This is implemented in the 3D model using containers with the following parameters:

- The base is a square with a side length of 65 mm.
- The container has a height of 80 mm.
- The wall thickness of the container is 1.5 mm.

You can model this container by creating two cuboids, whereby one is the outer form and the other is the inner form. These two cuboids must then be subtracted from each other to form the container. This works as follows:

Creating a sketch for the first square:

→ Create a sketch in accordance with Chapter 7.1.1, "**Section: Creating a sketch**". Save the model under the file name "**container**".

Sketching the first square for the container:

Two squares are needed for modeling the container.

→ The first square forms the outer form of the container. For this purpose, you create a **square** with dimensions of **65 mm**, starting from the origin. The procedure is the same as explained in Chapter 7.1.1 except that different dimensions are used here. The result of the sketch is shown in Figure 25. Then close the sketch.

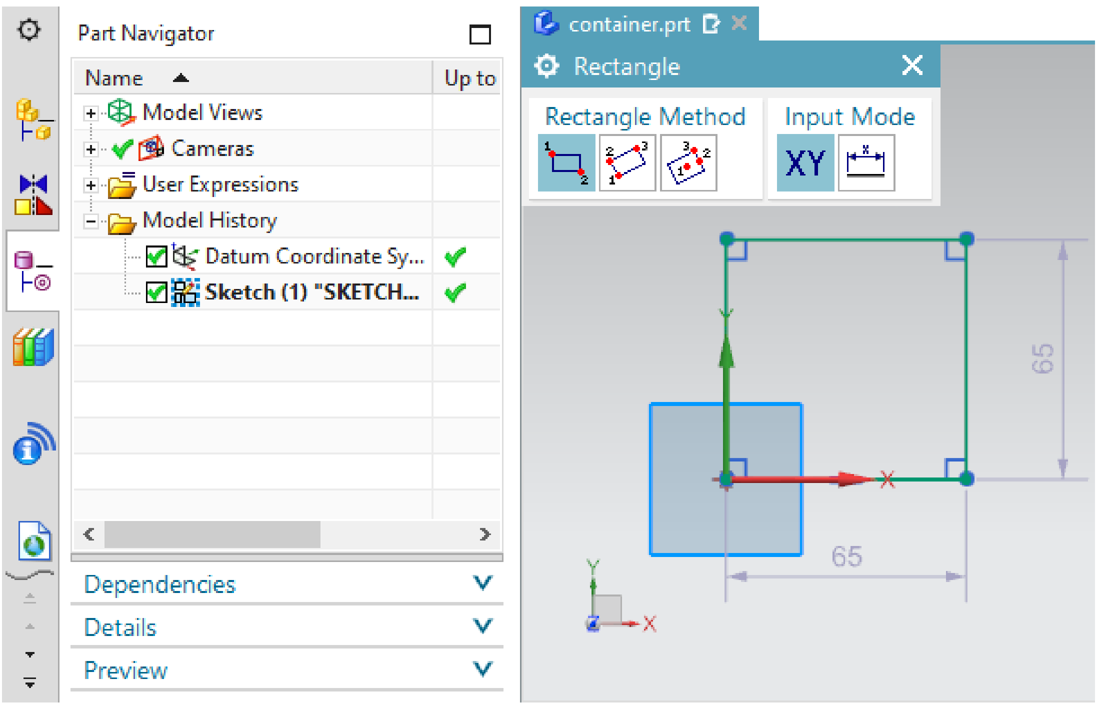


Figure 25: Sketch of the outer form of the container

Creating a sketch for the second square:

→ Create another sketch **in the same model** in accordance with Chapter 7.1.1, "**Section: Creating a sketch**".

Sketching the second square for the container:

→ The second square is to define the interior of the container. The positioning of the square will determine the thickness of the walls of the container. For this, you create a **square** with side length of **62 mm**. Use the method you applied for the first square of the container, but change the coordinates of the points (see Figure 26):

- Point 1 is to be located not at the origin but at coordinates **XC = 1.5** and **YC = 1.5**.
- Point 2 is to be fixed at coordinates **XC = 63.5** and **YC = 63.5**.

Now, close this sketch as well.

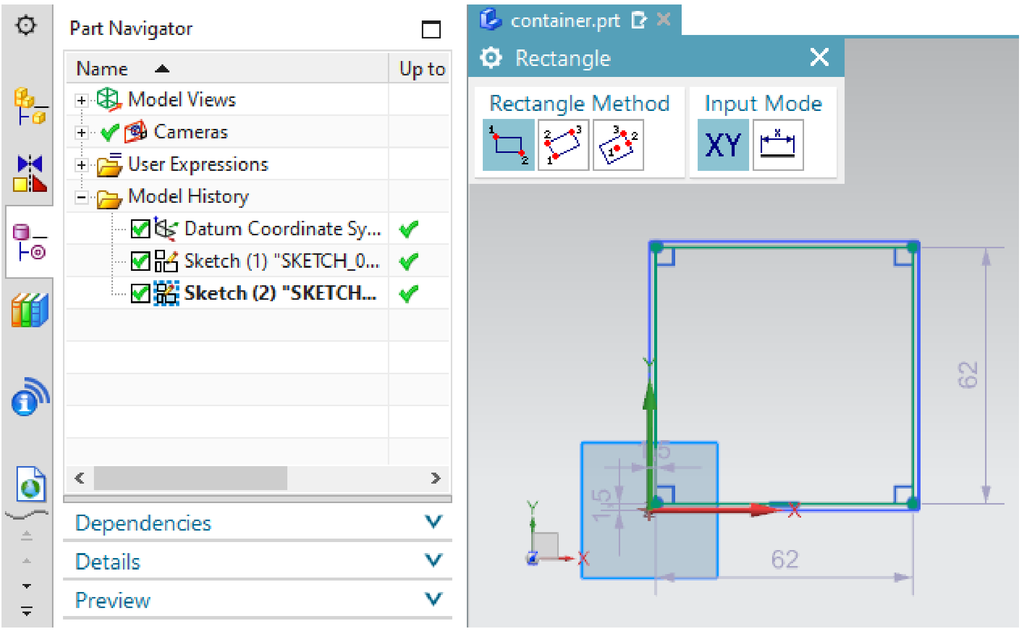


Figure 26: Sketch of the interior of the container

NOTE

NX uses an international number system. For this reason, decimal numbers use a period as the decimal separator (e.g. 1.5 mm) and not a comma (e.g. 1,5 mm).

At present, this can only be changed by changing the system configuration of the operating system, but this is not recommended.

Extruding the outer square:

→ You should now see both sketched squares in the graphics window. Now, switch back to the trimetric view. Begin converting the first square into a three-dimensional cuboid. This is to represent the outer form of the container. Use the "**Extrude**" command again to implement this (see Figure 27, step 1). In the "Extrude" window, click on "Select Curve" (see Figure 27, step 2). Switch to the Part Navigator in the Resource bar, and select "**Sketch (1) "SKETCH_000"** " (see Figure 27, step 3). In the Limits submenu, select a distance of **0 mm** for the start value and a distance of **80 mm** for the end value (see Figure 27, step 4). Confirm your entries with the "**OK**" button (see Figure 27, step 5). You have now generated the outer form of the container.

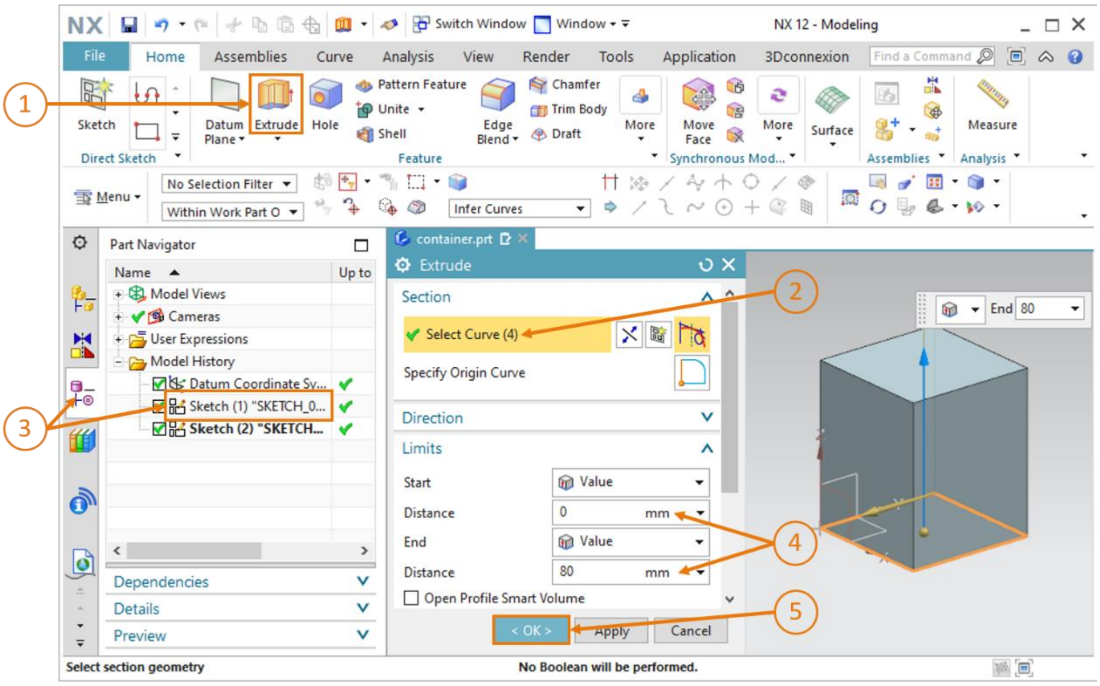


Figure 27: Modeling the outer form of the container with "Extrude"

Extruding the inner square and subtracting it from the outer form

→ The interior of the container must now be created. Within a CAD system it is not only possible to model individual forms but also to combine them with one another. In this case, an inner cuboid is to be created and subtracted from the outer form of the container. First, select the "**Extrude**" command again (see Figure 28, step 1). As indicated in Figure 28, step 2, click on "Select Curve" and select the sketch with the inner square "**Sketch (2) "SKETCH_001"** " in the Part Navigator (see Figure 28, step 3). In the "Limits" tab, select a distance of **1.5 mm** for the start value so that the wall thickness of the base will be 1.5 mm and a distance of **80 mm** for the end value (see Figure 28, step 4).

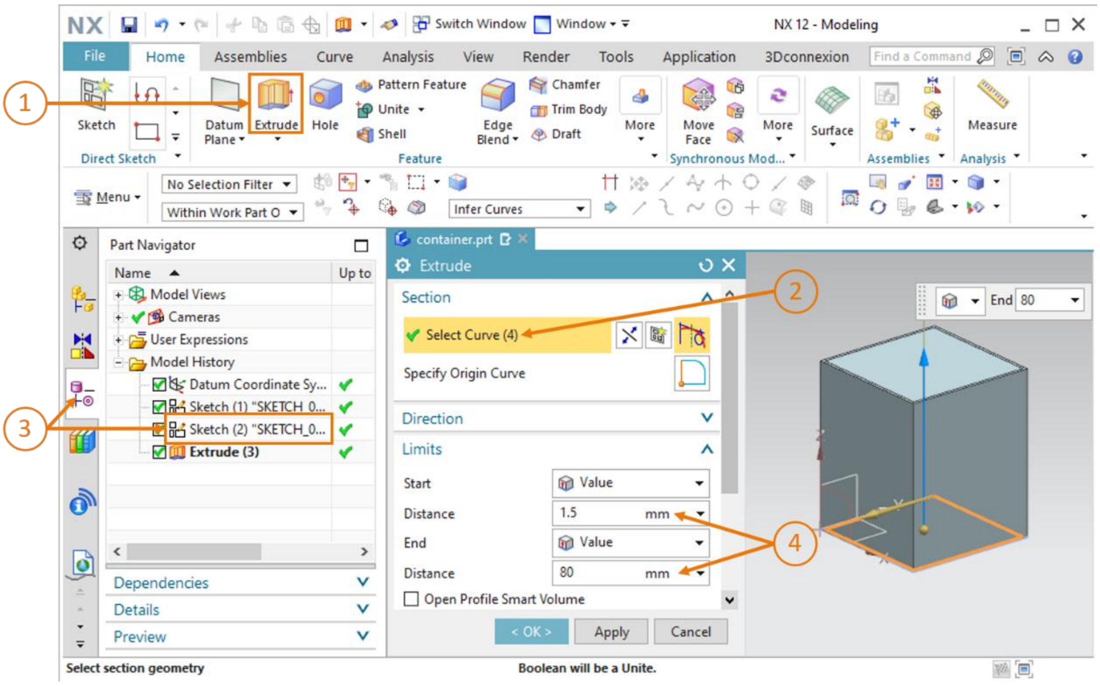


Figure 28: Extruding the inner form of the container

→ To subtract one body from another, navigate down to the "**Boolean**" section of the "Extrude" window. First, select the "**Subtract**" option for the Boolean operation (see Figure 29, step 1). Then click on "Select Body" (see Figure 29, step 2) and select the extruded outer form of the container from the Part Navigator. In this model, you will find this under the name "**Extrude (3)**", as shown in Figure 29, step 3. You can now see from the graphics window how the subtraction acts on the overall body. Confirm your entries by clicking "**OK**" (see Figure 29, step 4).

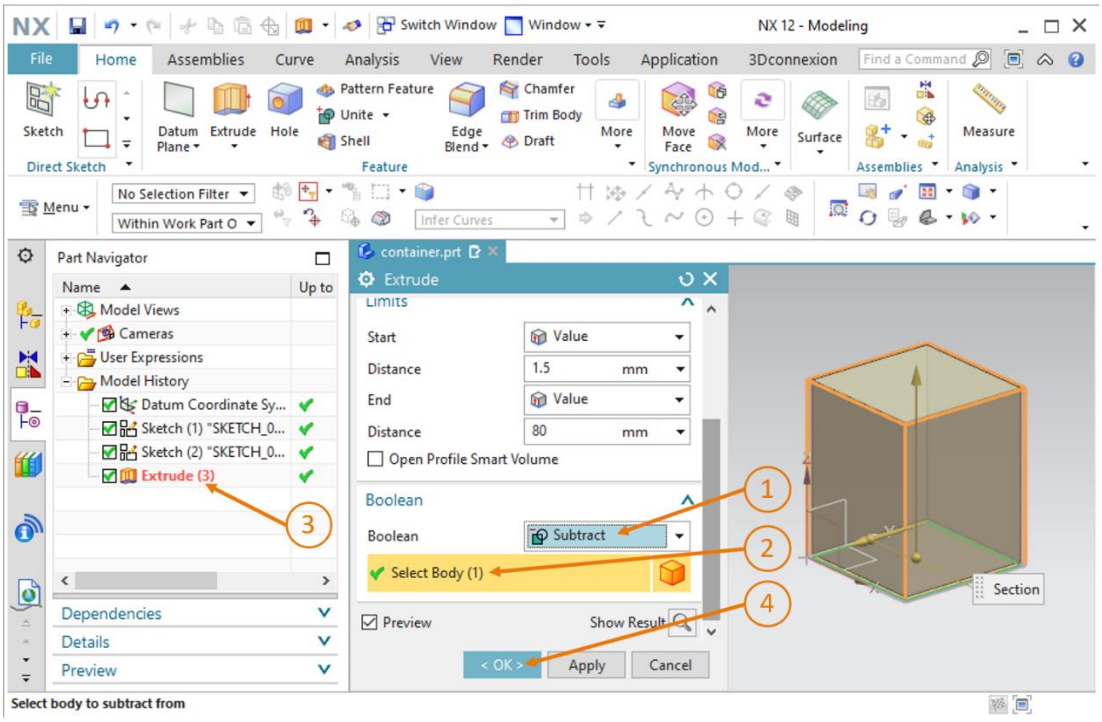


Figure 29: Subtracting the interior from the outer form of the container

NOTE

Most of the forming commands in NX have an "Apply" button in addition to the "<OK>" button.

- If you click on "<OK>", the last settings are applied and the corresponding command window is then closed.
- If you click on "Apply", the last settings are applied, but the window remains open.

You have now completed the modeling of the container. It should have the same form as shown in Figure 30. As done in the previous models, switch back to the trimetric view. Save and close the model.

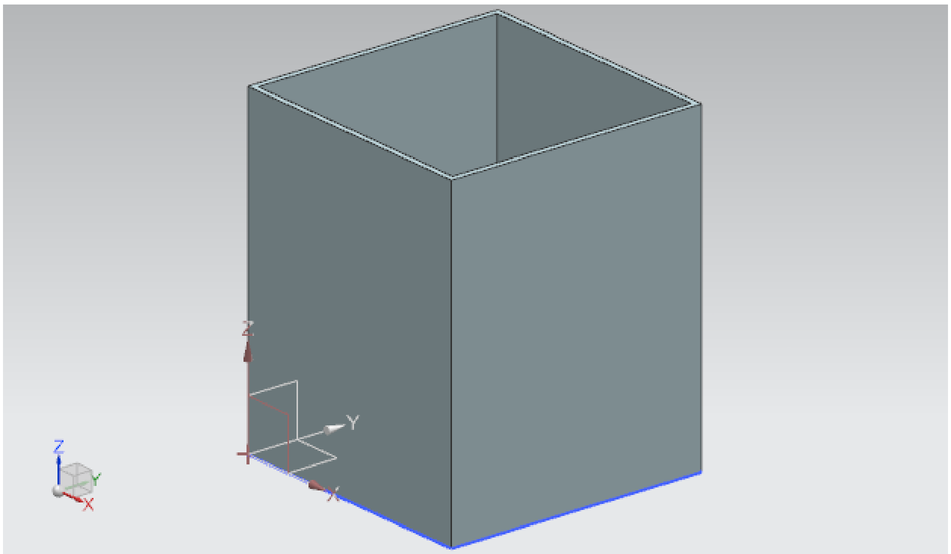


Figure 30: Finished container model

7.1.6 Modeling the base of the thrust cylinder

The thrust cylinder consists of two components: the fixed base and the head for ejecting workpieces. In this chapter, you are to model the base of the thrust cylinder. In so doing, the following parameters are to be adhered to:

- The base has a square base surface with a side length of 25 mm. The total height of the base is 90 mm.
- A hole with a diameter of 10 mm is inserted in the center of the body and extends 80 mm into the body. The head of the thrust cylinder, which will be modeled in Chapter 7.1.7, will be passed through this hole.
- The outer edges of the base are to be rounded.

The following is a description of how you can prepare this model.

Creating a sketch for the base surface of the thrust cylinder in NX:

→ Create a new model as previously described in Chapter 7.1.1, "**Section: Creating a sketch**". However, save the model under the name "**cylinderLiner**".

Sketching a square for the base surface:

→ The procedure for sketching the square base surface is similar to the one explained in Chapter 7.1.1. However, the points are to be positioned differently.

- Point 1 is to be assigned the value "**-12.5 mm**" for both XC and YC.
- Point 2 is to be assigned the value "**+12.5 mm**" for both XC and YC.

This should produce a sketch like the one shown in Figure 31. Then close the sketch.

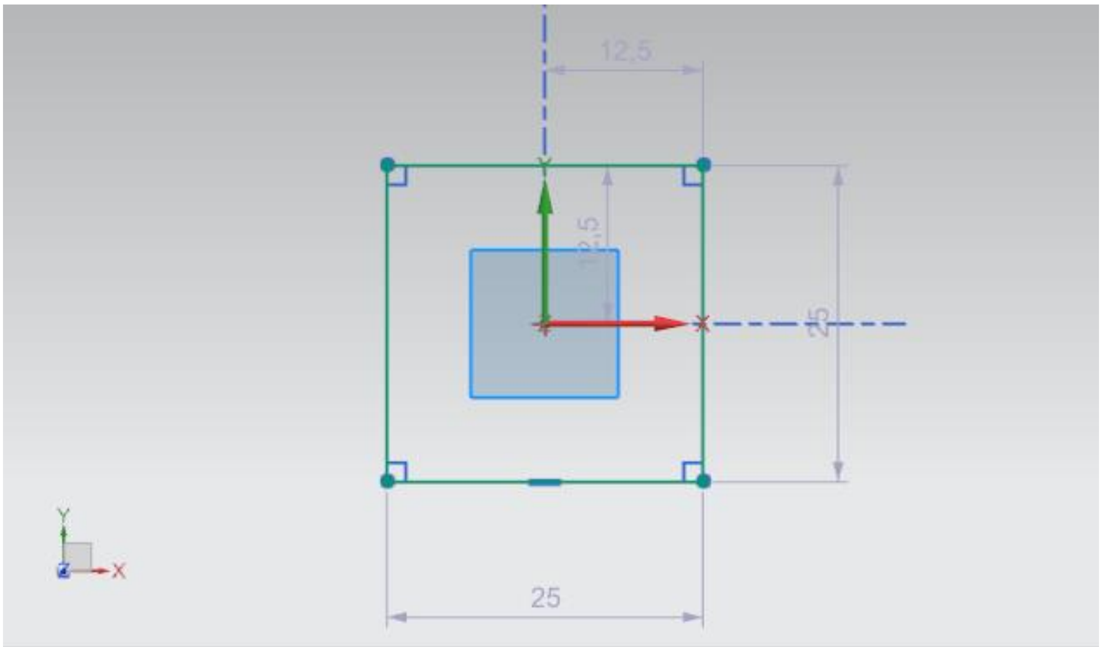


Figure 31: Sketch of the square for the base

Creating a sketch for the hole in the base:

→ Now create a new sketch in the same model, as was done previously in Chapter 7.1.5.

Creating a circle for the hole in the base:

→ Because the hole corresponds to the surface of a cylinder, follow the procedure indicated previously in Chapter 7.1.2. The circle is to have a diameter of **10 mm** in this case. Your sketch should now look like Figure 32. Then close the sketch.

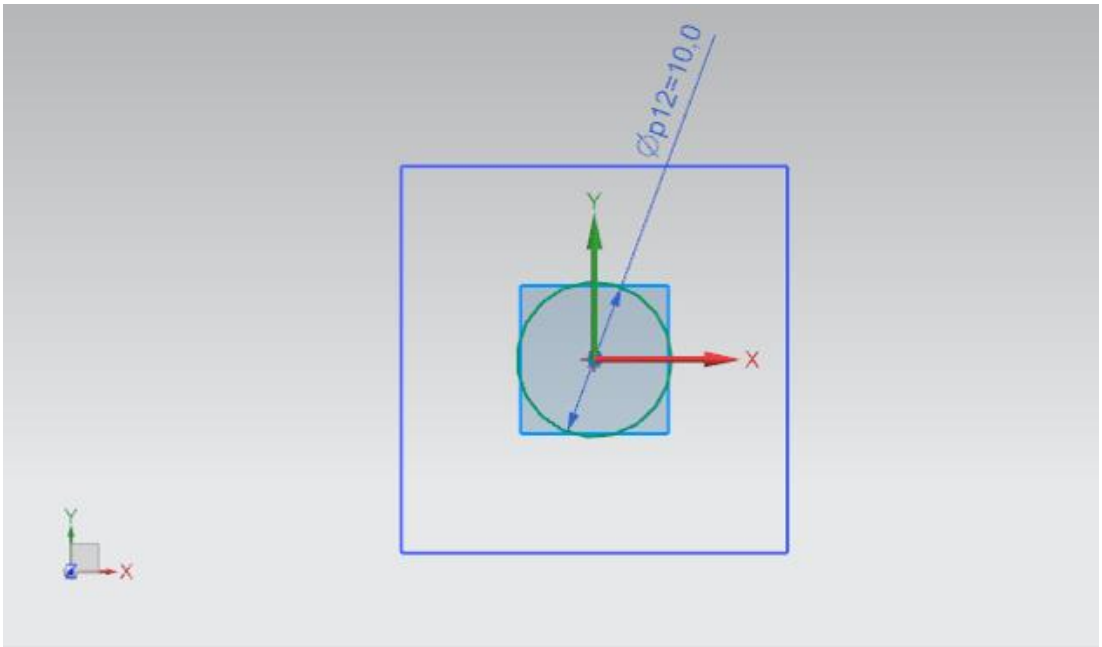


Figure 32: Sketch of the circle as the hole for the base

Extruding the base for the thrust cylinder:

→ Now, generate a 3D body for the base of the thrust cylinder from your preceding sketch. Use the "**Extrude**" command for this (see Figure 33, step 1). For definition of the sketch, select the first sketch named "**Sketch (1) "SKETCH_000"** " in the Part Navigator (see Figure 33, steps 2+3). The start value is to have a distance of **0 mm** and the end value a distance of **90 mm** (see Figure 33, step 4). Confirm your entries by clicking the "**OK**" button as indicated in Figure 33, step 5.

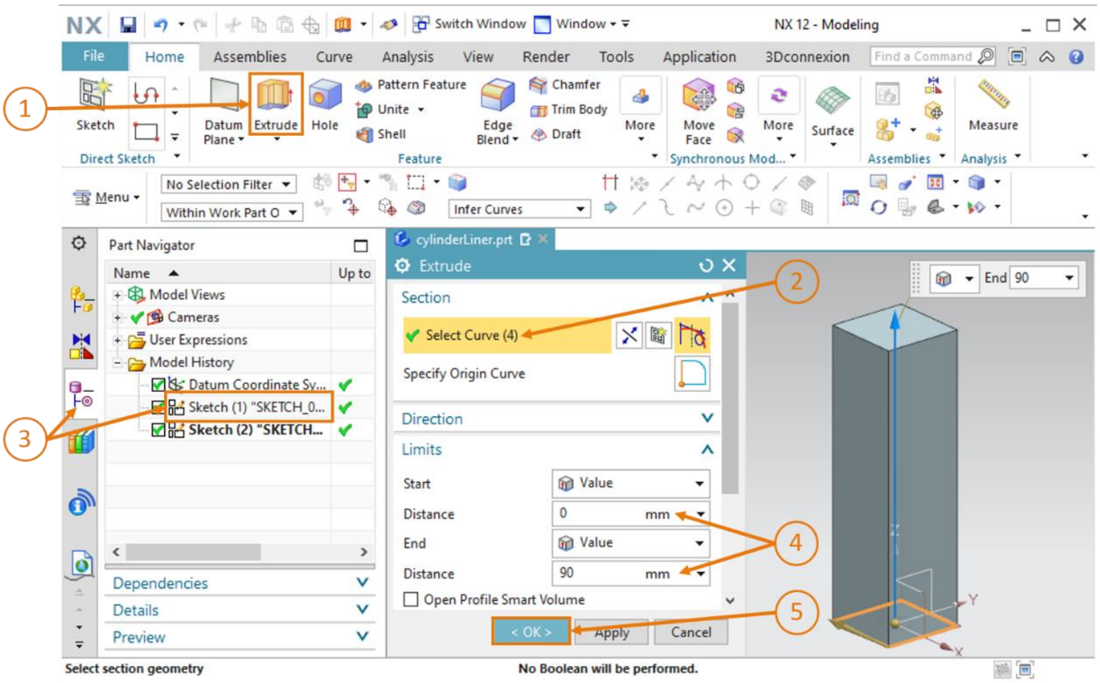


Figure 33: Extruding a square into the base body of the thrust cylinder

Inserting a hole in the base of the thrust cylinder by extruding and subtracting:

→ Following a very similar principle to Chapter 7.1.5, a hole is also to be introduced into the base body of the thrust cylinder. Execute the "**Extrude**" command (see Figure 34, step 1) and select the circle as the sketch. You will find this in the sketch named "**Sketch (2) "SKETCH_001"** " (see Figure 34, steps 2+3). Because the parameters dictate that the hole is to extend only 80 mm, a start value with a distance of **10 mm** and an end value with a distance of **90 mm** must be specified (see Figure 34, step 4).

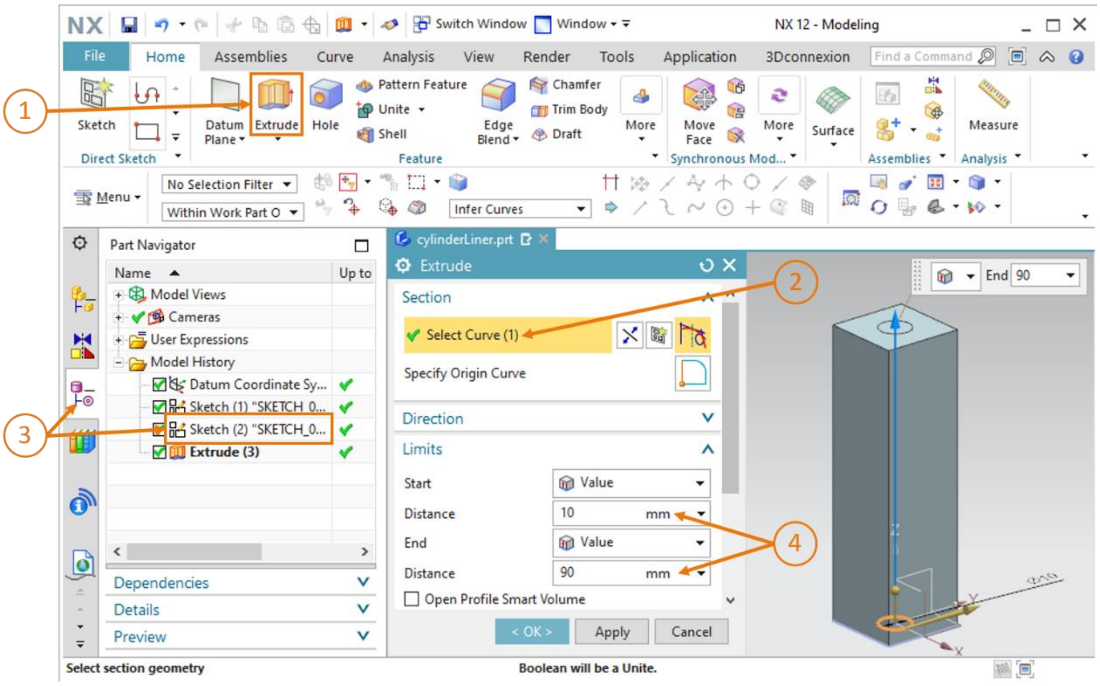


Figure 34: Extruding a hole in the base body

→ Go to the "**Boolean**" section in the "Extrude" command window. Select the Boolean operation "**Subtract**" in order to subtract the cylinder body from the base (see Figure 35, step 1). Select the extruded base body as the body. The name of the body in the model history is once again "**Extrude (3)**" (see Figure 35, steps 2+3). Confirm the operations by clicking the "**OK**" button (see Figure 35, step 4).

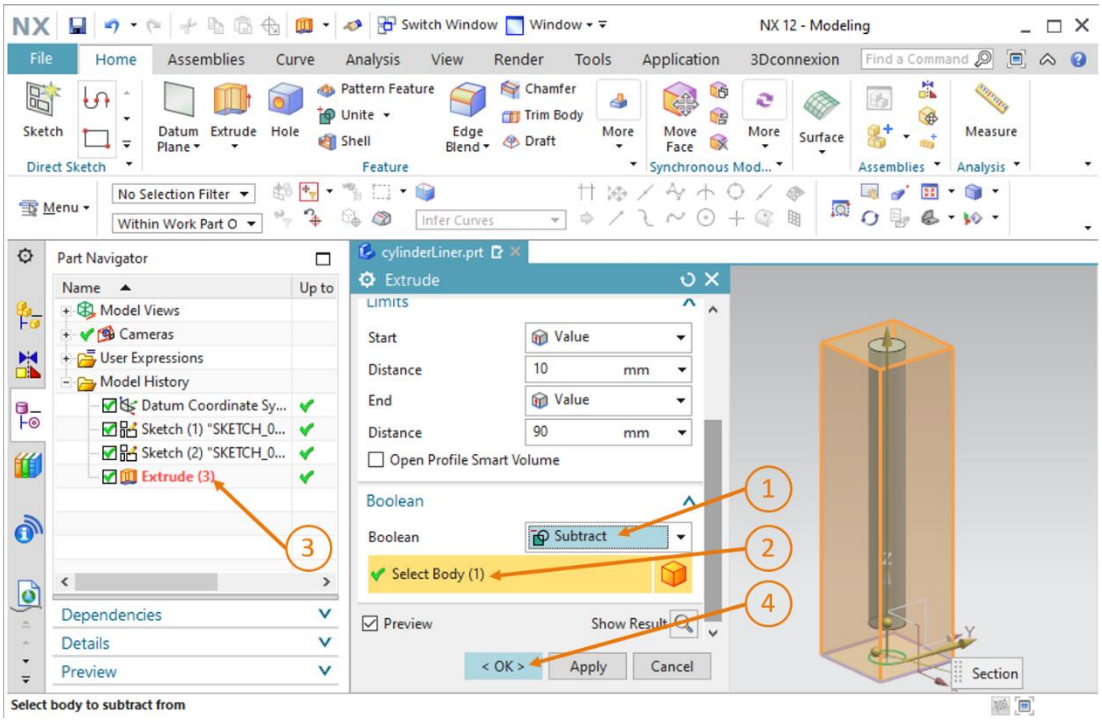
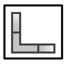


Figure 35: Inserting a hole in the base by "subtracting"

Rounding the long edges of the base of the thrust cylinder:

→ The long edges of the base are to be rounded similarly to the two conveyor belts. To do this, first switch the view to the "**Front**" view  as previously described in Chapter 7.1.3. You can also use the Command Finder for this. Open the "**Edge Blend**" command (see Figure 36, step 1) and select the **two long edges**, which can be seen in the graphics window (see Figure 36, step 2). Leave the shape set to "**Circular**" and specify a radius of **5 mm** (see Figure 36, step 3). Then confirm your entries with "**OK**" (see Figure 36, step 4).

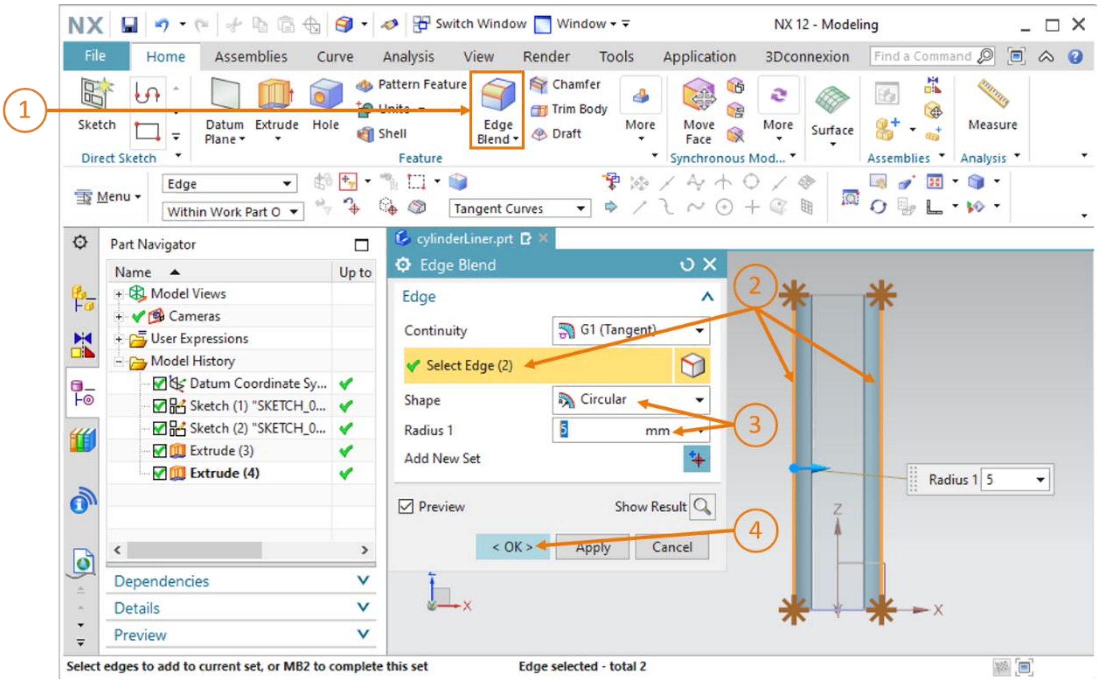
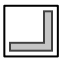


Figure 36: Rounding the long edges on the base

→ Repeat this step for the rear side of the base. Change from the "Front" view to the "**Back**" view  for this and follow the instructions given above.

The modeling of the base for the thrust cylinder is now complete. You can see the finished model in Figure 37. Conclude by activating the trimetric view to save and close this model.

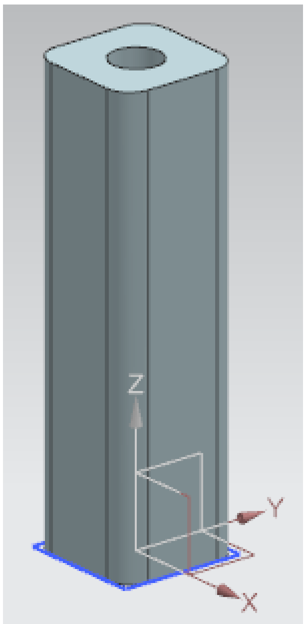


Figure 37: Finished 3D model of the base for the thrust cylinder

7.1.7 Modeling of the head of the thrust cylinder

The head or arm is needed as the second component of the thrust cylinder. This will eject the cylinder workpieces. The following data must be heeded when modeling the 3D model:

- The guide cylinder has a length of 92 mm and a diameter of 10 mm so that it will fit exactly into the hole of the thrust cylinder base.
- The head of the thrust cylinder has a square surface with a side length of 25 mm. The thickness of the head is 5 mm.
- The head or arm is attached orthogonally to the tip of the guide cylinder.

To create this 3D model, you can proceed as follows:

Creating a sketch for the guide cylinder in NX:

→ Create a new model in accordance with Chapter 7.1.1, "**Section: Creating a sketch**", and save it under the name "**cylinderHead**".

Sketching a circle for the guide cylinder:

→ When creating the circle sketch, follow the procedure described previously in Chapter 7.1.2. Note that, in this case, the circle is to have a diameter of **10 mm** because it is to fit into the base of the thrust cylinder. Then close the sketch.

Your sketch should now look like the one shown in Figure 38.

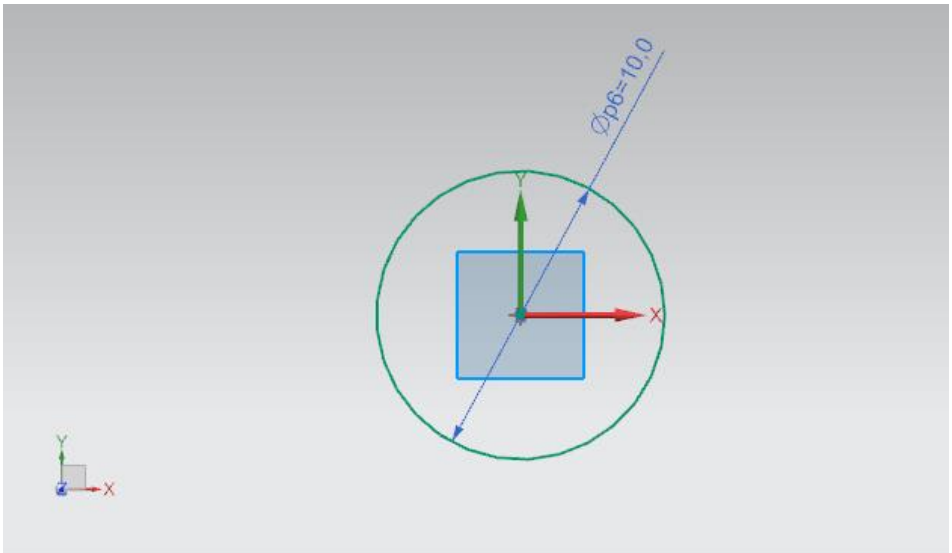


Figure 38: Sketch of the circle for the guide cylinder

Creating a sketch for the arm in NX:

→ Create a second sketch in the same model. The process is the same as described in Chapter 7.1.5.

Sketching a square for the arm:

→ You can use Chapter 7.1.6 as a guide for drawing the square in this sketch. The points are to be set exactly the same as for the base.

- Point 1 is to have the value **-12.5** for both XC and YC.
- Point 2 is to obtain the value **+12.5 mm** for both XC and YC.

This yields a square with a side length of 25 mm whose center point is in the center of the circle created in the previous step (see Figure 39).

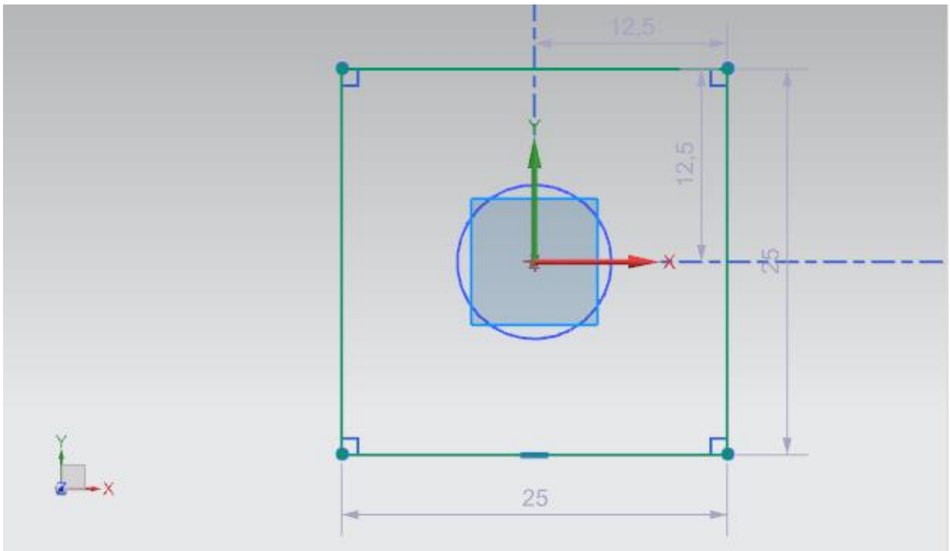


Figure 39: Sketch of the square for the arm

Creating the guide cylinder by extruding:

→ As the first 3D body of this model, create the guide cylinder for the head of the thrust cylinder. To do so, open the "**Extrude**" command window (see Figure 40, step 1), and for definition of the sketch, as described previously in Chapter 7.1.5, select the first sketch with the circle in the Part Navigator. Here, this is named "**Sketch (1) "SKETCH_000"** " (see Figure 40, steps 2+3). Specify a distance of **0 mm** for the start value and a distance of **92 mm** for the end value (see Figure 40, step 4). Confirm your entries by clicking the "**OK**" button (see Figure 40, step 5).

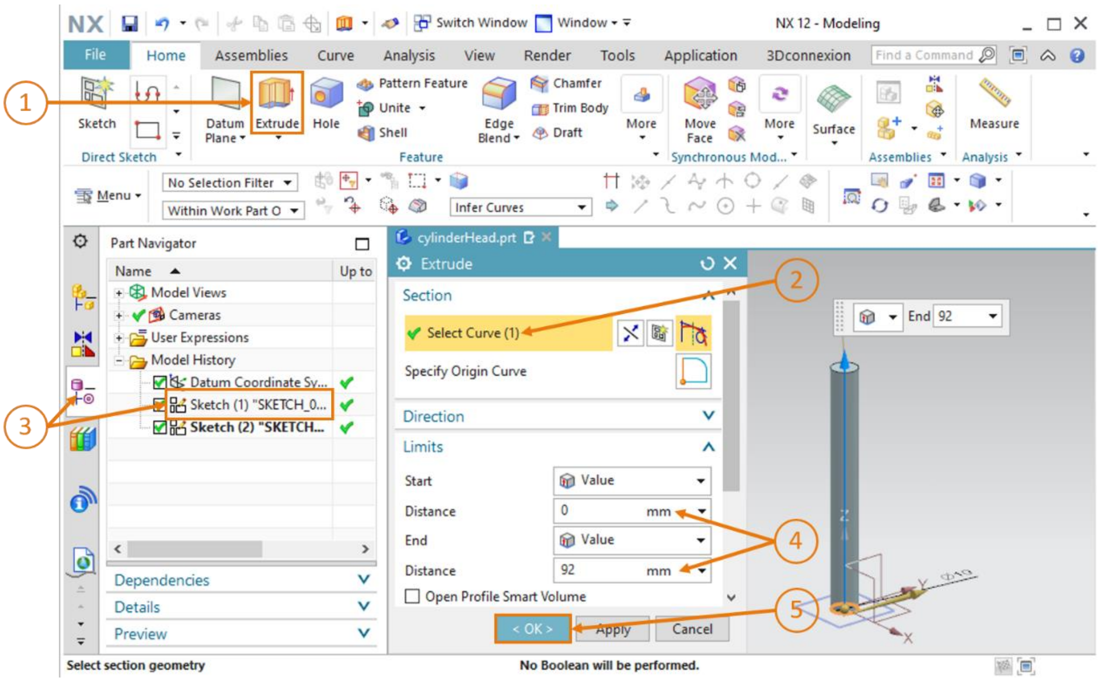


Figure 40: Creation of the guide cylinder by extruding

Modeling the arm and uniting the two bodies:

→ For modeling the arm, you also need the "**Extrude**" function (see Figure 41, step 1). However, in this case you need to select your second sketch named "**Sketch (2) "SKETCH_001"** " as illustrated in steps 2 and 3 in Figure 41. Because the arm is to be attached at the top end of the guide cylinder, specify a distance of **92 mm** for the start value. For a thickness of 5 mm, you need to define a distance of **97 mm** for the end value. This is shown in step 4 in Figure 41.

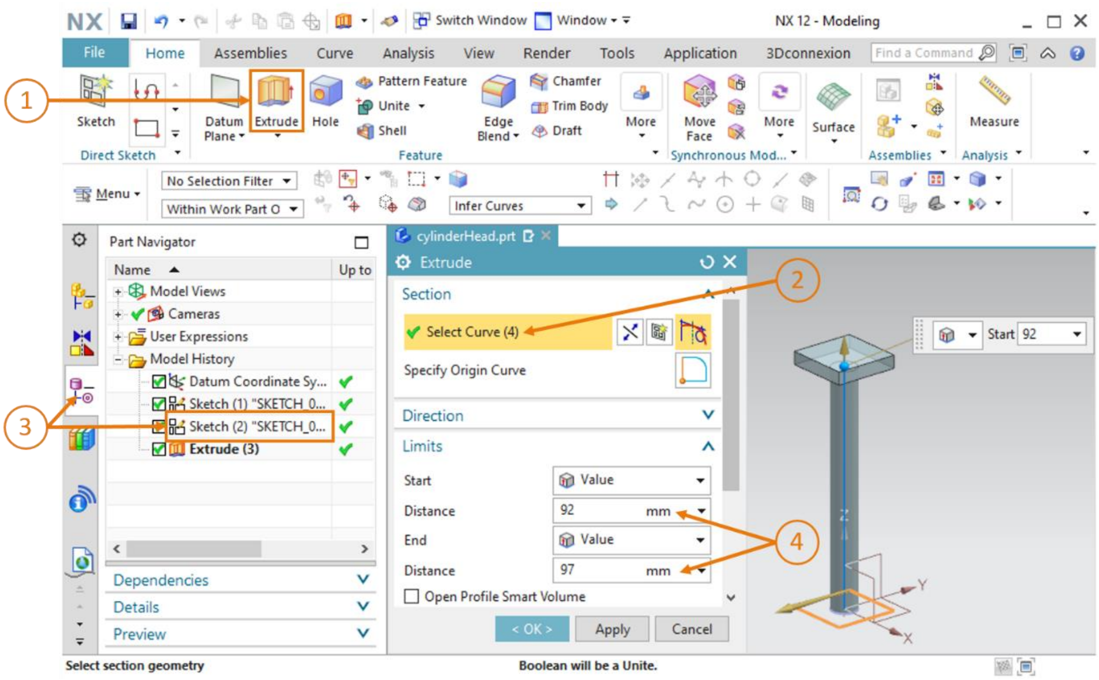


Figure 41: Modeling of the arm

→ To create one overall component from the two bodies, go to the "**Boolean**" section of the "Extrude" window, as described previously in Chapter 7.1.5. Instead of subtraction, this time select the "**Unite**" option (see Figure 42, step 1) and select the previously extruded guide cylinder as the body (see Figure 42, steps 2+3). Confirm your selection with "**OK**" (see Figure 42, step 4).

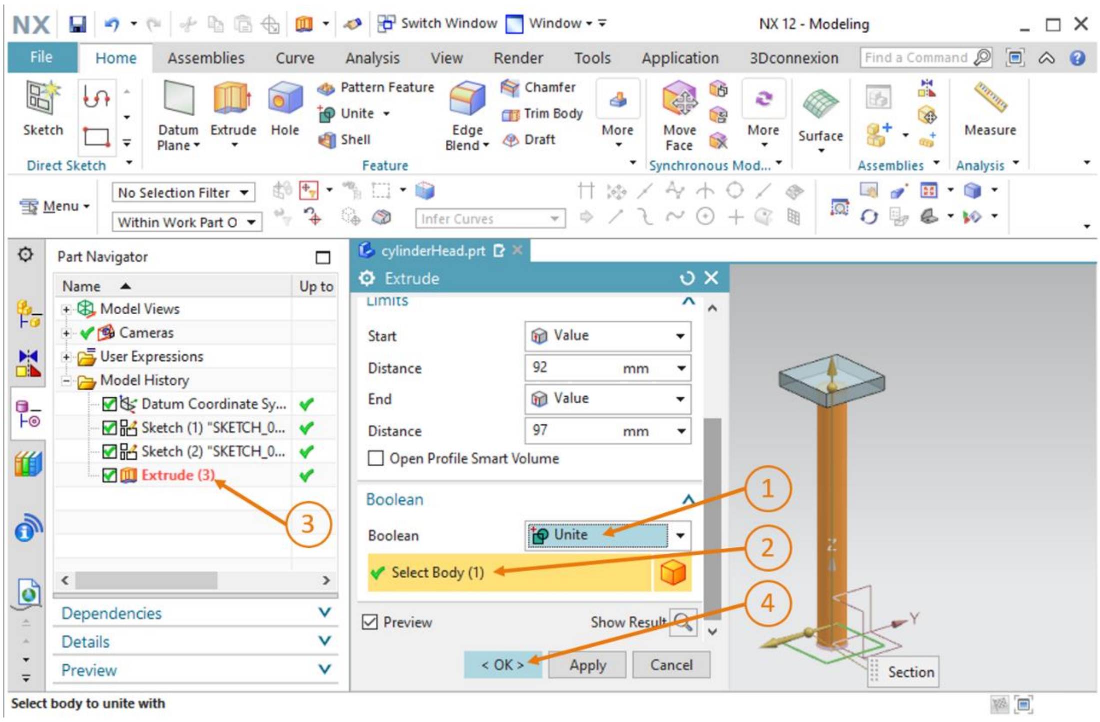


Figure 42: Uniting the arm with the guide cylinder to form one component

The modeling of the thrust cylinder head is now complete and should result in one overall body as in Figure 43. Switch back to the trimetric view to save and close the model.

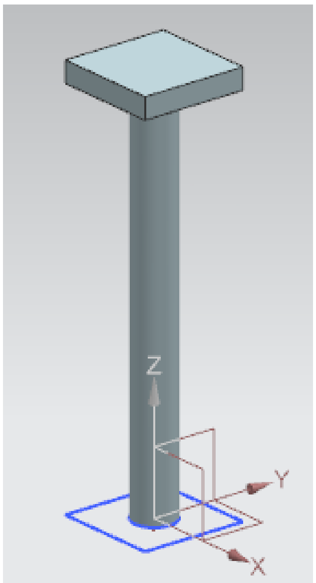


Figure 43: Finished 3D model of the head component of the thrust cylinder

7.1.8 Modeling a position sensor with photoelectric sensor

Photoelectric sensors are use in the sorting plant to detect the different workpieces. Two different 3D models are used for this:

- A photoelectric sensor (see Figure 44).

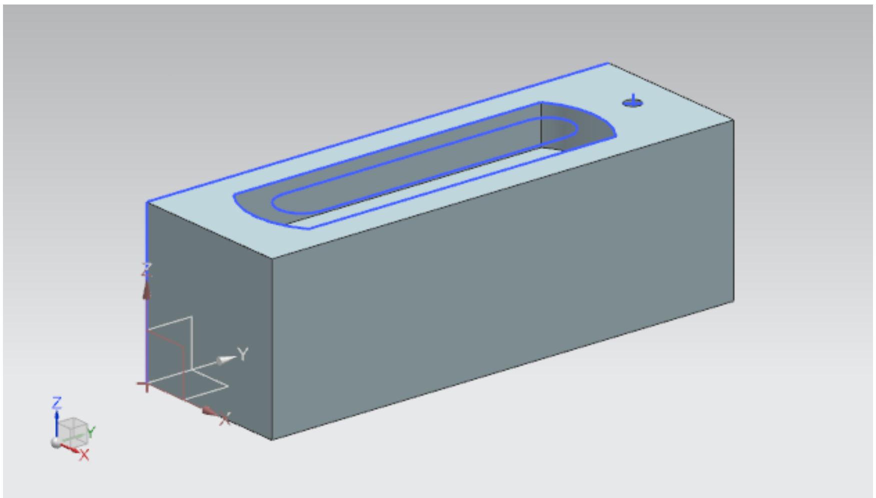


Figure 44: 3D model of the photoelectric sensor

- A light beam that is to be inserted between the two photoelectric sensors (see Figure 45).

Figure 45: 3D model of the light beam for the photoelectric sensors

These two models are provided to you so that you do not have to model them personally. Following the introductory instructions in Chapter 7, copy the "**lightSensor.prt**" and "**lightRay.prt**" files from the "**Components ToImport**" folder to your own working directory containing the models you created previously. If necessary, you can trace the underlying modeling process via the Model History in the Part Navigator of the model.

All the 3D models for the sorting plant are now available and you can proceed to create an assembly.

7.1.9 Modeling the limit switch for the thrust cylinder

Two limit switches are to be inserted in the base for detecting the position of the thrust cylinder. They signal if the head of the thrust cylinder is fully retracted or fully extended. The light beam from Chapter 7.1.8 is to serve as the basic model. However, different dimensions must be set for the light beam.

Proceed as follows:

→ Open the light beam in NX. To do this, click the "**Open**" button in the "Home" menu bar (see Figure 46, step 1). Navigate to your working directory and select the "**lightRay.prt**" file containing your model of the light beam (see Figure 46, step 2). Be sure to select the "**Partially Load**" option so that only the model with the relevant drawings is opened (see Figure 46, step 3). Then confirm your selection with "**OK**" (see Figure 46, step 4).

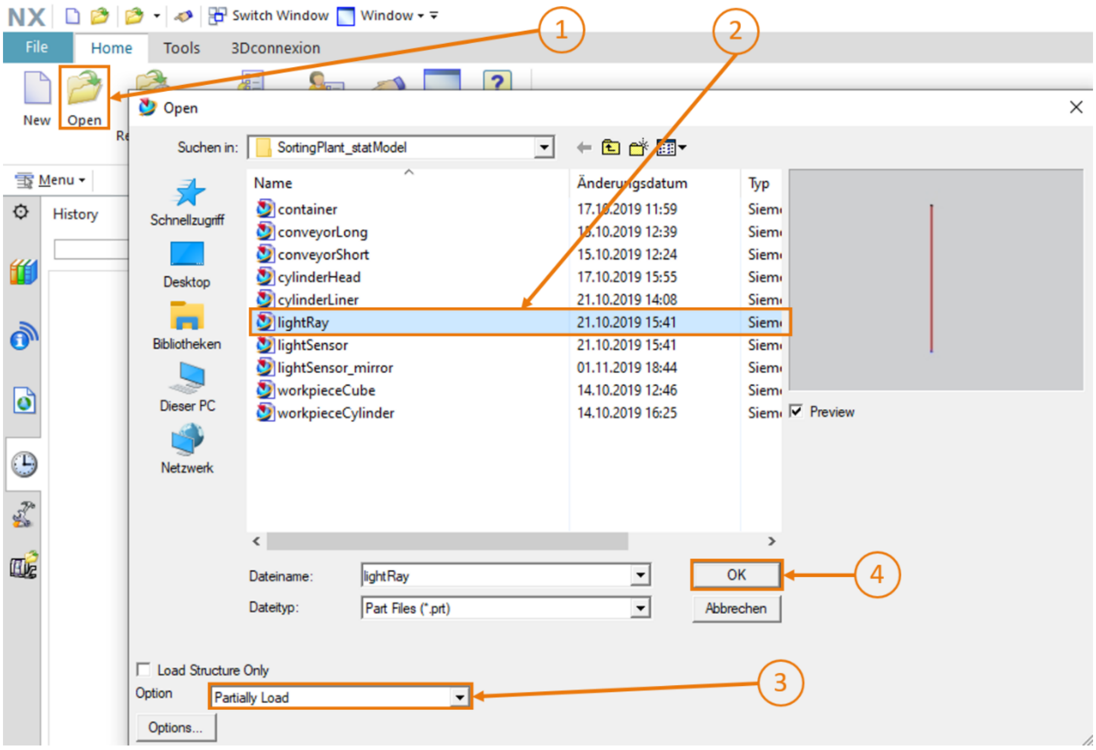


Figure 46: Opening the "lightRay" model in NX

→ Now save the model as a copy for the limit switches. For this, you go to the "File" menu and click on the "**Save As**" button in the "Save" submenu (see Figure 47, step 1). Navigate to your working directory and save the copy under the name "**limitSwitchSensor**" (see Figure 47, step 2). Confirm your settings by clicking the "**OK**" button, as shown in Figure 47, step 3.

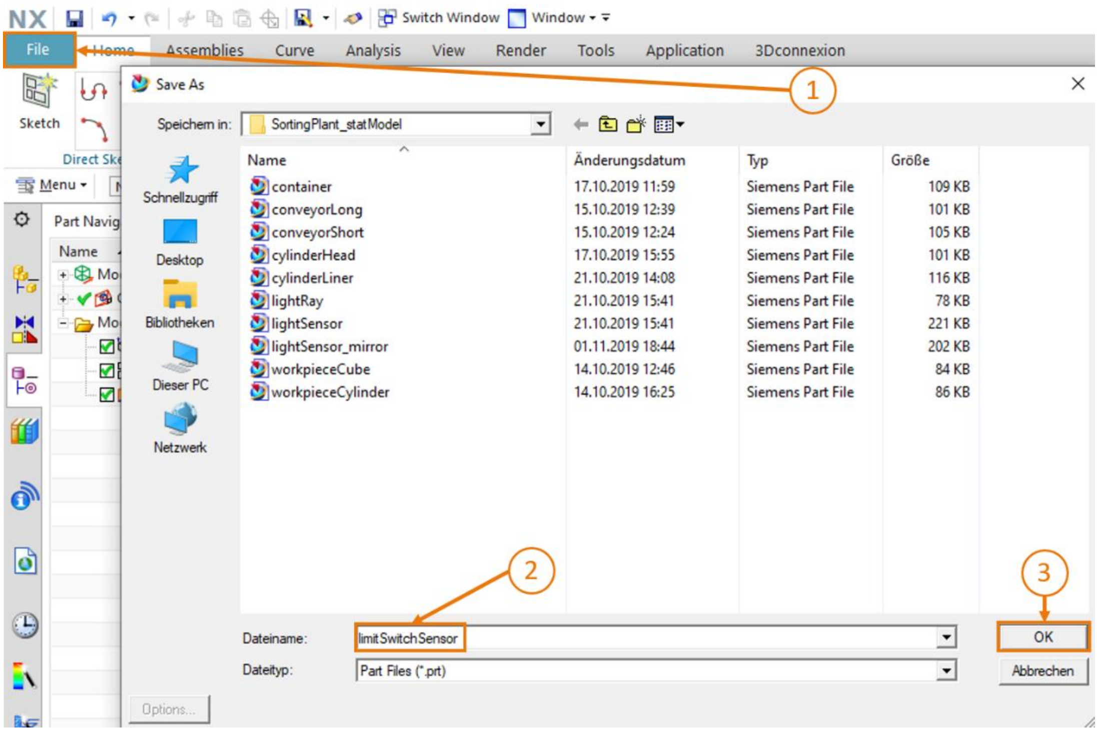


Figure 47: Saving a copy for the limit switches

→ Next, the copy of the model is to be adjusted by shortening the height of the light beam. For this, you right-click on the "**Extrude**" modeling step in the **Part Navigator** (see Figure 48, step 1). Then, left-click on "**Edit Parameters**" in the shortcut menu (see Figure 48, step 2).

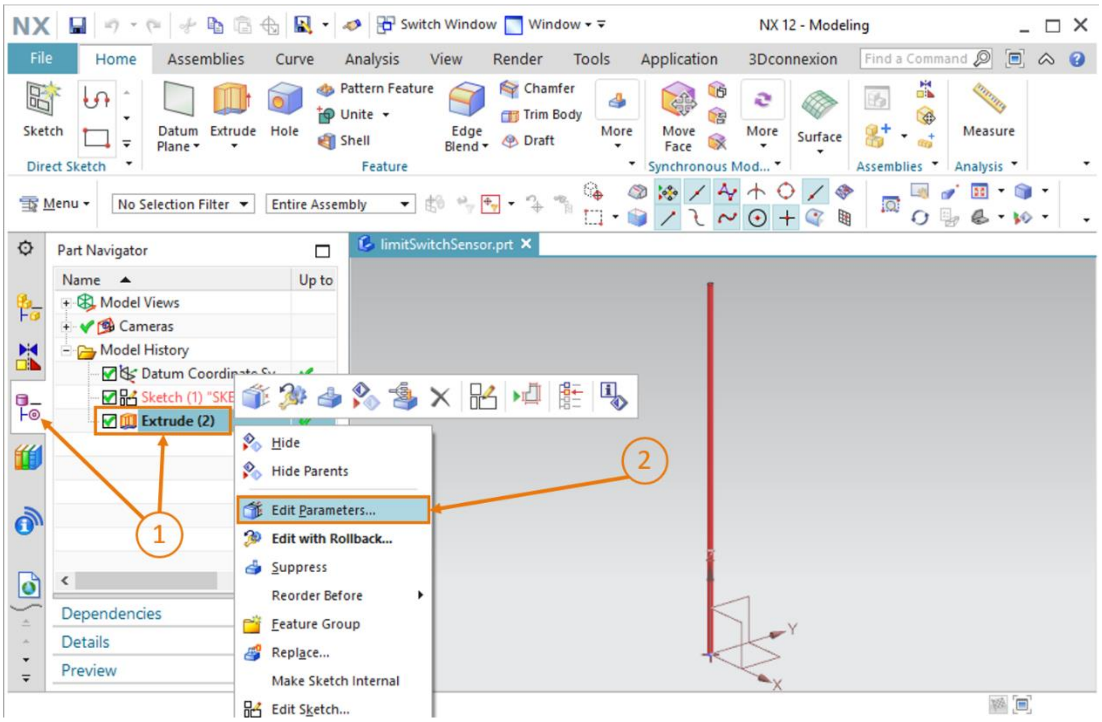


Figure 48: Editing with the Extrude option for the limit switches

→ Now, adjust the height to **8 mm** using the "Distance" parameter, as shown in Figure 49, step 1. Confirm this by clicking the "**OK**" button (see Figure 49, step 2).

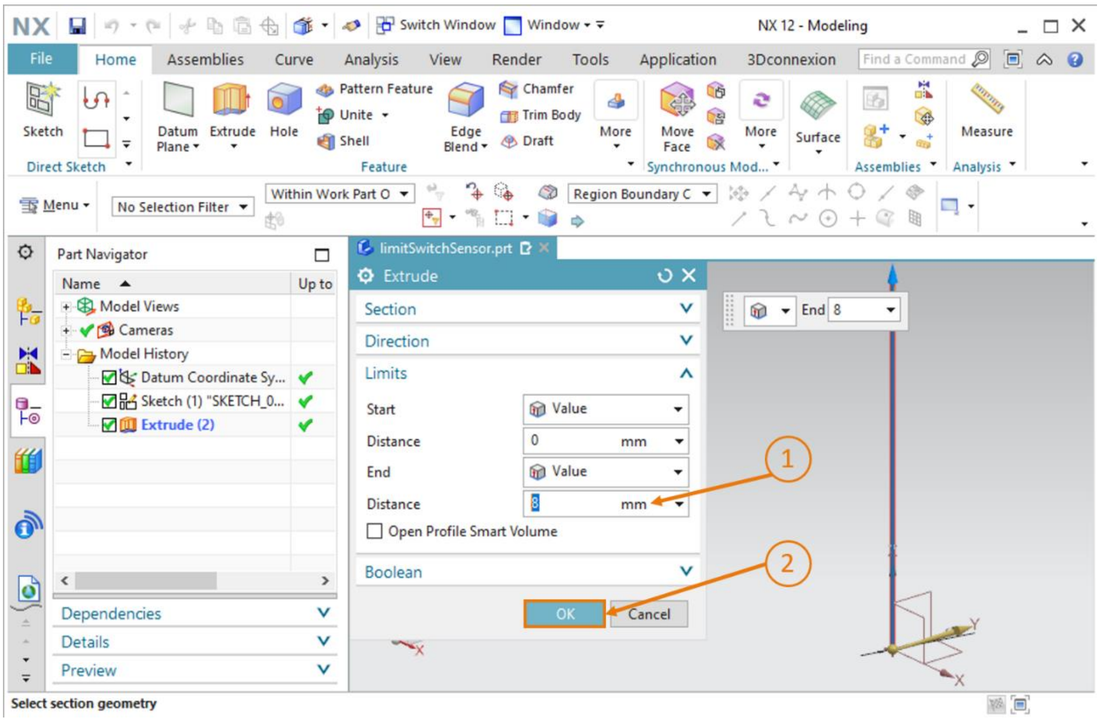


Figure 49: Adjusting the height of the limit switch

The modeling of the limit switch for the thrust cylinder, as represented in Figure 50, is now complete. Switch back to the trimetric view to save and close the model.

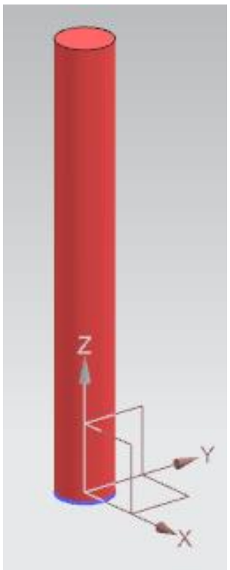


Figure 50: 3D model of the limit switch for the thrust cylinder

All the 3D models for the sorting plant are now available and you can proceed to create an assembly.

7.2 Merging all the models into an assembly

You have finished creating the individual static models in Chapter 7.1. Now, all the individual models must be converted into an overall model. "Assemblies" are used in NX for this. Components can be inserted and positioned in an assembly. In this chapter, you are to create the sorting plant from your previously completed models.

Note that, for simplification purposes, the coordinates for positioning the models are specified here, so that the complete sorting plant will result from the components in the end. In assemblies you yourself create, you will have to determine the orientation and position of your components and arrange them accordingly.

7.2.1 Creating an assembly

The first step is to create an assembly. Proceed as follows:

→ If you have not already done so, open the "**NX V12.0**" software and wait until the Welcome Screen appears, as shown in Figure 6. Click the "**New**" button (see Figure 51, step 1) and navigate in the new window to the "**Model**" tab (see Figure 51, step 2). This time, an "**Assembly**" instead of a model, as shown in Figure 51, step 3. Use a meaningful name for the assembly. Specify the name "**assSortingPlant**" for the sorting plant and select the same directory in which the individual models were saved (see Figure 51, step 4). Confirm with the "**OK**" button to create the new assembly (see Figure 51, step 5).

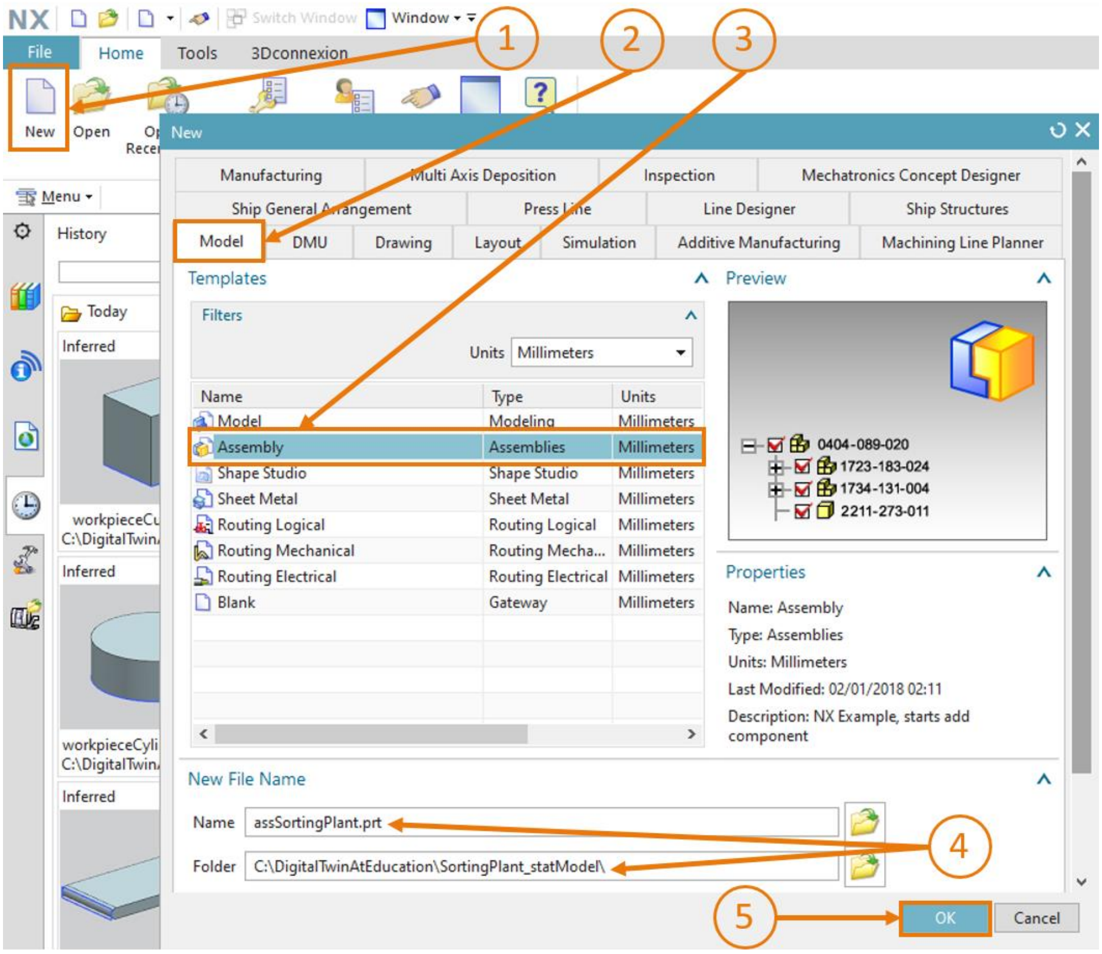


Figure 51: Creating an assembly

Afterwards, similar to the case with models, the NX "Modeling" application opens. This application is also used for preparation of assemblies. Next, switch to the trimetric view and save the empty assembly.

You can now insert the individual models gradually into this assembly.

NOTE

When saving files, you should follow a clear naming convention so that you can distinguish between a model and an assembly. In the existing modules, the models have normal names in "camelCase" notation. To distinguish them, the names of assemblies begin with the prefix "**ass**".

You can make use of several assembly functions, some of which are shown in Figure 52. The main function you will use within this task is "Add".

Figure 52: Excerpt of common assembly functions in NX

Section: Inserting and positioning a model

7.2.2 Inserting and positioning the "ConveyorShort" conveyor belt

The first component to be inserted is "ConveyorShort". This is to be positioned at the origin of the world coordinate system of the assembly. You can insert the short conveyor belt into the assembly with the following steps:

→ Ensure that you are in the "**Home**" tab of the menu bar (see Figure 53, step 1). Select the "**Add**" assembly function  as shown in Figure 53, step 2. The "**Add Component**" command window containing four submenus opens. Expand the "**Part To Place**" submenu (see Figure 53, step 3) and click on the "**Open**" button (see Figure 53, step 4). A new window appears in which you can select the appropriate model. Navigate to your working directory containing the models you created in Chapter 7.1. Select your "**conveyorShort**" model in this window and confirm your selection by clicking "**OK**". If you are unable to see your models, make sure you have selected the "**Part files (*.prt)**" option as the file type, which is NX's modeling format.

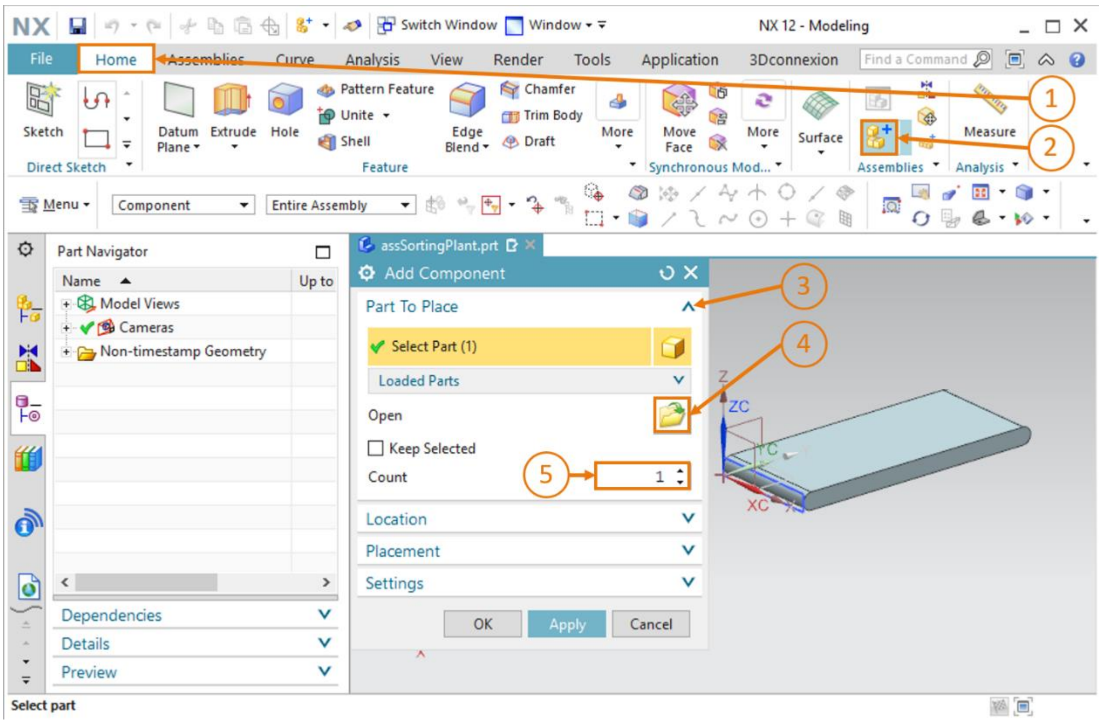


Figure 53: Adding the "conveyorShort" model to the assembly – Part selection

→ The result should be that a part has been selected. You can recognize this from the parenthesized information next to "**Select Part**", which is "**(1)**", as shown in Figure 53. Finally, also select the number "**1**" for the count, because only a single short conveyor belt is to be placed in the assembly (see Figure 53, step 5).

→ Close the "Part To Place" submenu and open the "**Location**" and "**Placement**" submenus (see Figure 54, step 1). In the "Location" submenu, select "**Absolute**" for the component anchor and "**WCS**" for the assembly location (see Figure 54, step 2) in order to align the orientation with the world coordinate system. Then select the "**Move**" method in the "Placement" submenu (see Figure 54, step 3). Click on Specify Orientation and then on the "**Manipulator**" button , as shown in Figure 54, step 4. You now see the image of the model with the orientation coordinates in space (X, Y and Z coordinates) in the three-dimensional graphics window. Enter the following coordinates for the short conveyor belt (see Figure 54, step 5):

- X value = **32.5 mm**
- Y value = **75.0 mm**
- Z value = **5.0 mm**

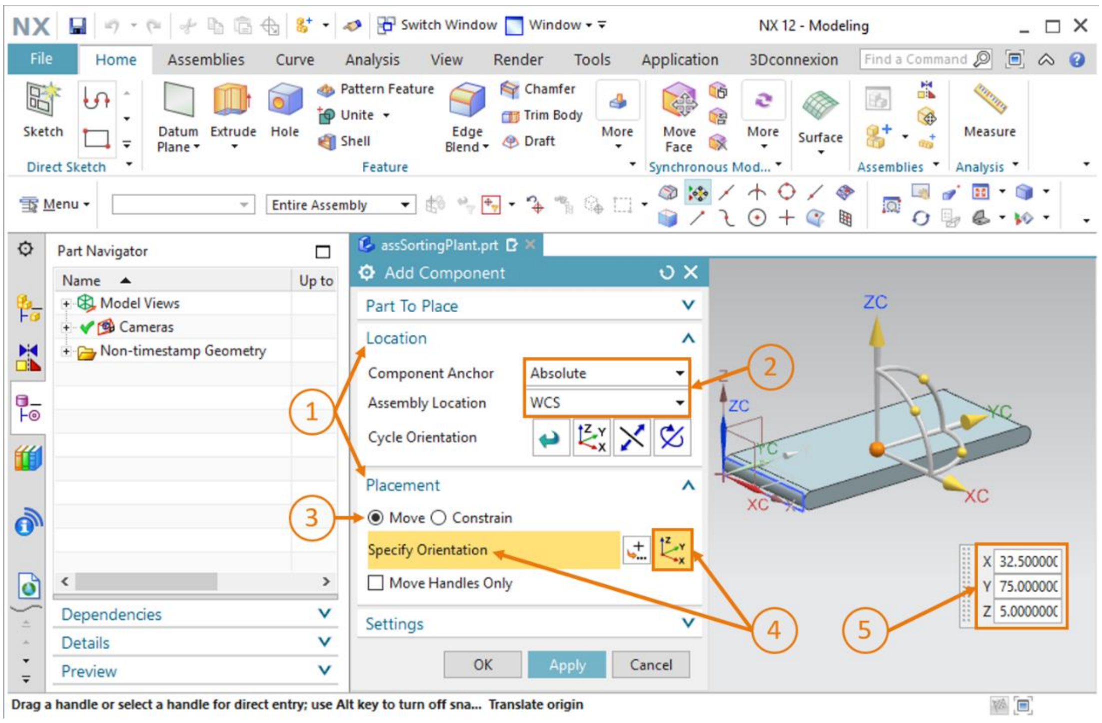


Figure 54: Adding the "conveyorShort" model to the assembly – Location and placement

→ Close the "Location" and "Placement" submenus. Next, open the "**Settings**" submenu (see Figure 55, step 1). Here, you should leave the **component name in uppercase letters**. **Ensure that only "Model" ("MODEL") is displayed as the reference set.** Only the three-dimensional model and not the two-dimensional sketches will then be inserted into the assembly. Leave the layer option set to **Original** and finish the creation process by clicking the "**OK**" button (see Figure 55, step 2). If you receive a message offering to automatically generate a so-called "fixed constraint", click the "No" button. Constraints will be covered in Module 5 of this workshop series.

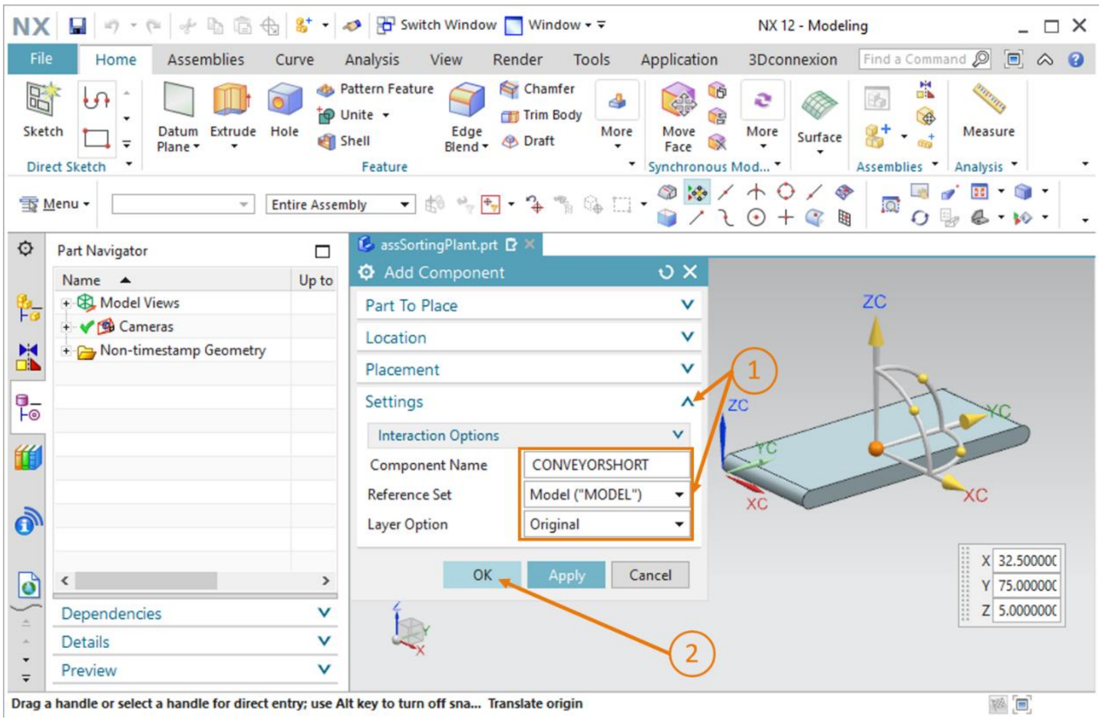


Figure 55: Adding the "conveyorShort" model to the assembly – Settings

→ You have now inserted and positioned your first model in the assembly Save the assembly.

NOTE

The "Settings" submenu may not show up in your command window by default. You can change this by adapting your Options preferences for the user interface. You will find these under Menu → Preferences → User interface (see Figure 56, step 1). The "**User Interface Preferences**" window opens. In the Options (see Figure 56, step 2), you can change the "**Default Presentation of Dialog Contact**" under "**Dialog Boxes**". Select the "**More**" option here, (see Figure 56, step 3) and then confirm with "**OK**" in order to apply the changes. You should now also see the additional settings by default.

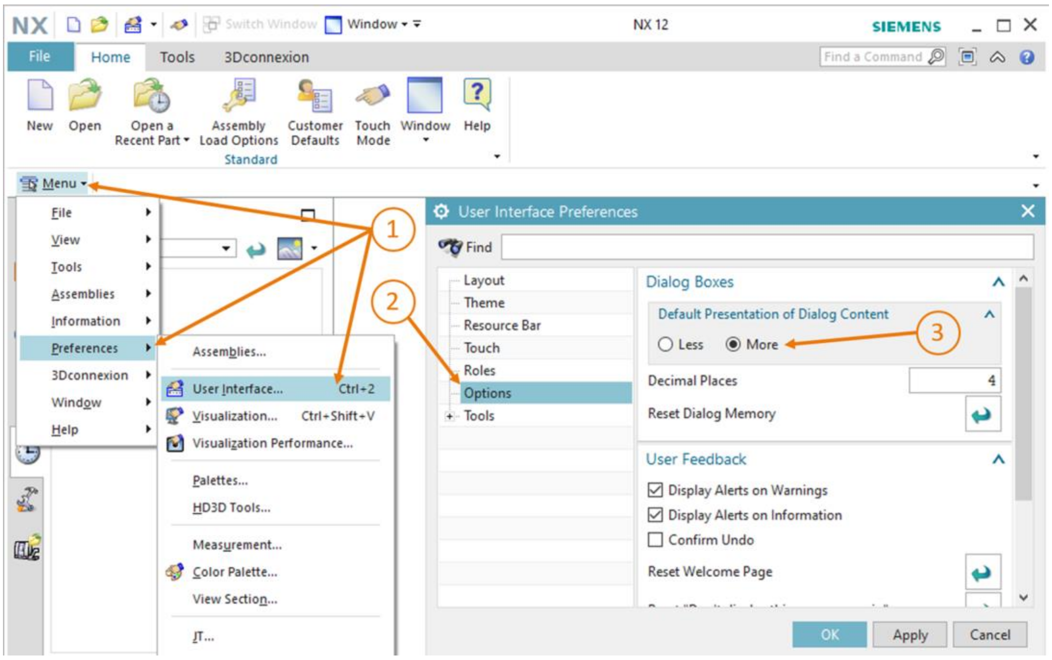


Figure 56: Advanced options for default presentation of dialog content

7.2.3 Inserting and positioning the "ConveyorLong" conveyor belt

In this chapter, you are to insert the "ConveyorLong" conveyor belt. This is to be positioned after the short conveyor belt so that the workpieces from "ConveyorShort" are then further transported with "ConveyorLong".

The procedure for this is identical to that described in Chapter 7.2.2, "**Section: Inserting and positioning a model**" except for the following:

→ When you select the "Part To Place" from the menu, select your "**conveyorLong**" model.

→ For the "Placement" of the long conveyor belt, specify the following coordinates, as shown in Figure 57, step 1:

- X value = **32.5 mm**
- Y value = **350.0 mm**
- Z value = **5.0 mm**

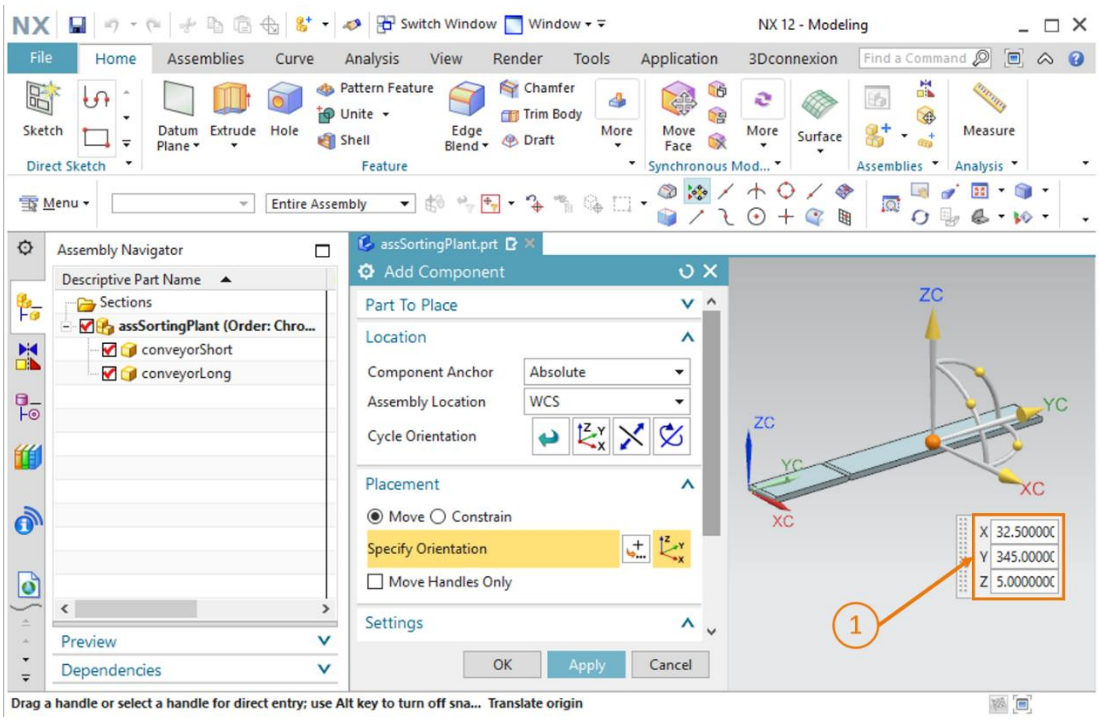


Figure 57: Positioning the "conveyorLong" model in the assembly

All the needed transport surfaces are now implemented as static models. You should save the assembly to conclude this chapter.

7.2.4 Inserting and positioning the cube workpiece

Next you are to place the workpieces at the starting position on "ConveyorShort". In this chapter, you are to place the cube workpiece onto the short conveyor belt.

The steps needed for this are as described in Chapter 7.2.2, "**Section: Inserting and positioning a model**" You must make the following adjustments, however:

→ For the "Part to Place", select the "**workpieceCube**" model from your working directory.

→ Position the component using the following space coordinates, as shown in Figure 58, step 1:

- X value = **32.5 mm**
- Y value = **25.0 mm**
- Z value = **22.5 mm**

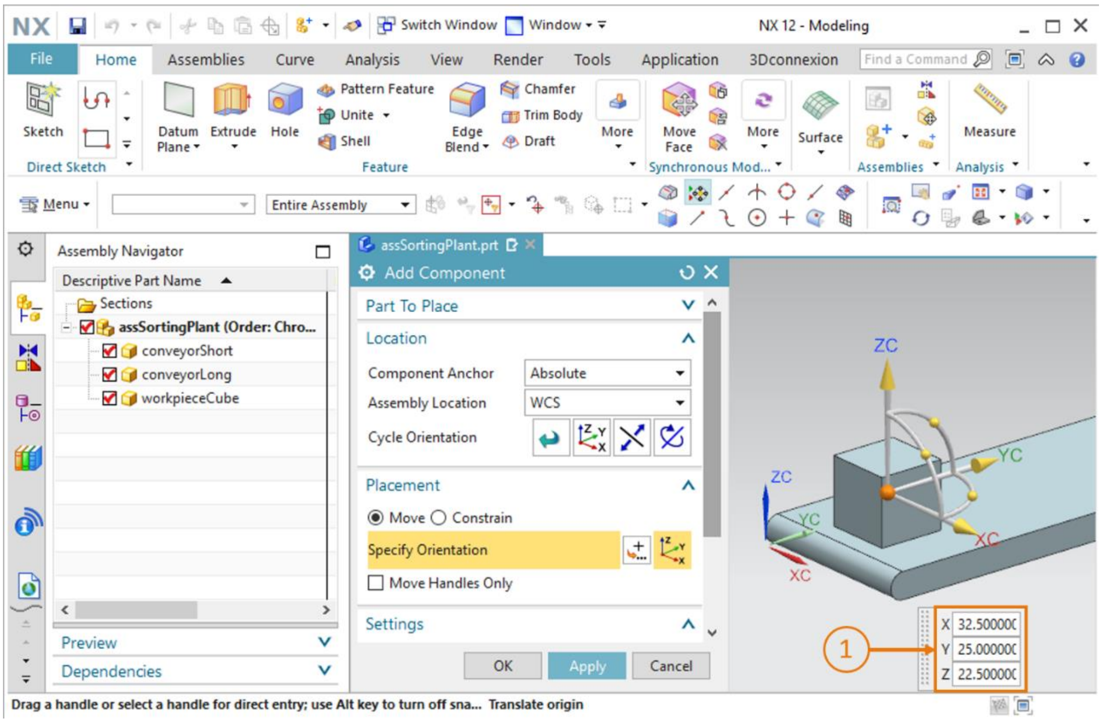


Figure 58: Positioning the "workpieceCube" model on the conveyor belt

Remember to save the sorting plant after completing this chapter.

7.2.5 Inserting and positioning the cylinder workpiece

In this chapter, the cylinder workpiece is to be placed on the short conveyor belt. The same position in the assembly is chosen for this as for the cube workpiece in Chapter 7.2.4. The reason for this, is that the starting positions are to be the same for both workpieces in the future dynamic model.

Therefore, the steps described in Chapter 7.2.2, "**Section: Inserting and positioning a model"** also apply to this component. Note the following changes, however:

→ In the "Part to Place" submenu, you need to select the "**workpieceCylinder**" model from your working directory.

→ Enter the following coordinates for the placement, as shown in Figure 59, step 1:

- X value = **32.5 mm**
- Y value = **25.0 mm**
- Z value = **15.0 mm**

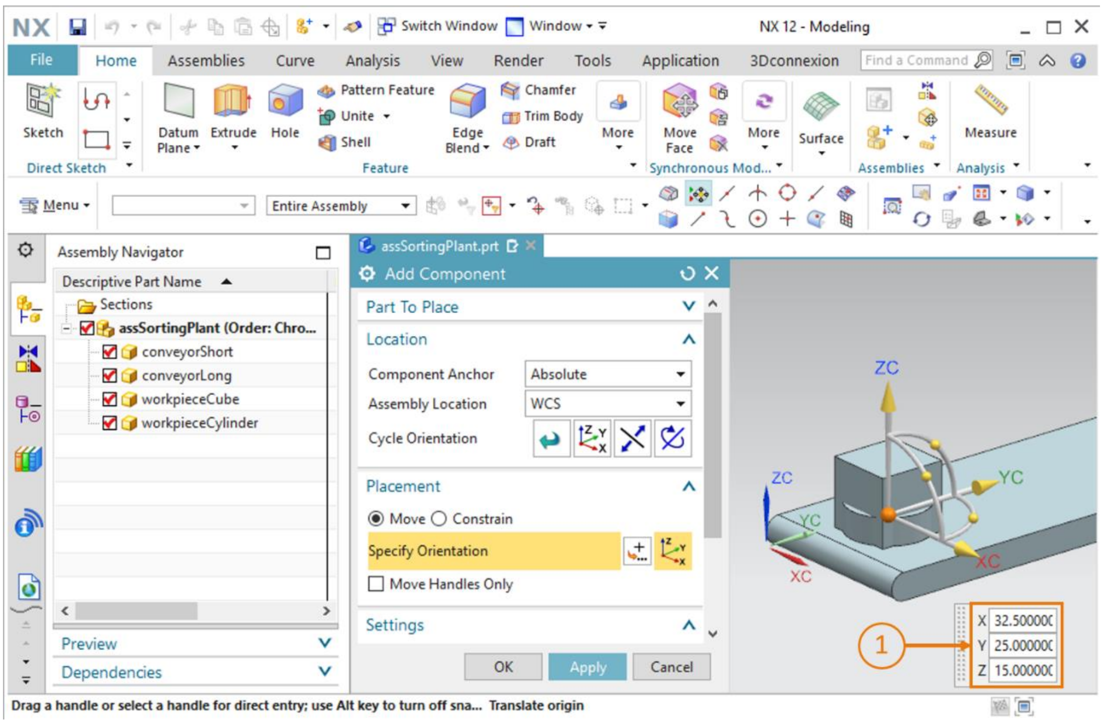


Figure 59: Positioning the "workpieceCylinder" model on the conveyor belt

Once again, you should save the assembly at this point.

7.2.6 Inserting and positioning the thrust cylinder

As you saw in Chapter 7.1.6 and 7.1.7, the thrust cylinder consists of two components: the base and the head.

Inserting and positioning the base in the assembly:

For adding the base, a couple of additional steps are required compared with the steps described in Chapter 7.2.2, "**Section: Inserting and positioning a model"**:

→ Here, as well, you need to open the window for adding new components and select your "**cylinderLiner**" model in the "Part To Place" submenu.

→ You can see here that the base is perpendicular to the transport surfaces. In this position, the thrust cylinder cannot eject any workpieces. Therefore, you should start by rotating the component. First, similar to the action in Chapter 7.2.2, select the "**Move**" method in the "Placement" submenu of the "Add Component" window (see Figure 60, step 1) and click on "**Specify Orientation**" (see Figure 60, step 2). For changing the orientation, first select the point between the X and Z axis in the three-dimensional graphics window, as shown in Figure 60, step 3. This allows you to rotate the component around the Y axis.

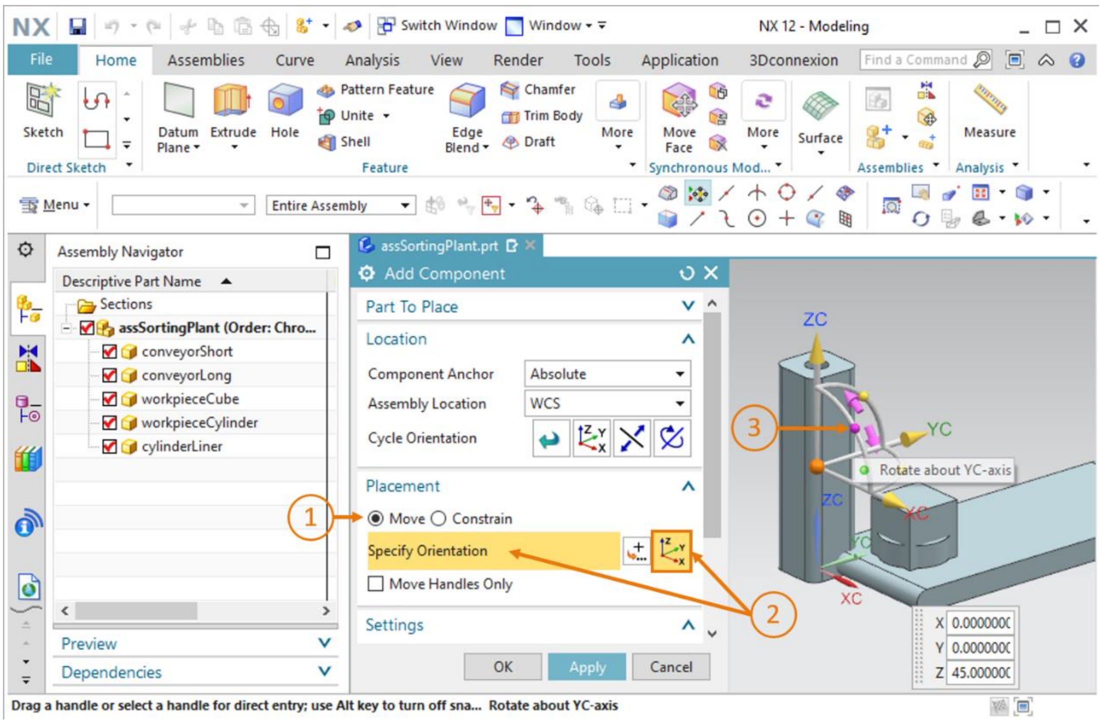


Figure 60: Rotating the "cylinderLiner" component – Selecting the axis

→ A new input window appears in the graphics window. For the required horizontal orientation, enter an angle of **270.0°**, as shown in Figure 61, step 1. Then click on the center point of the body in the graphics window again (see Figure 61, step 2), so that the origin can be moved again via the space coordinates.

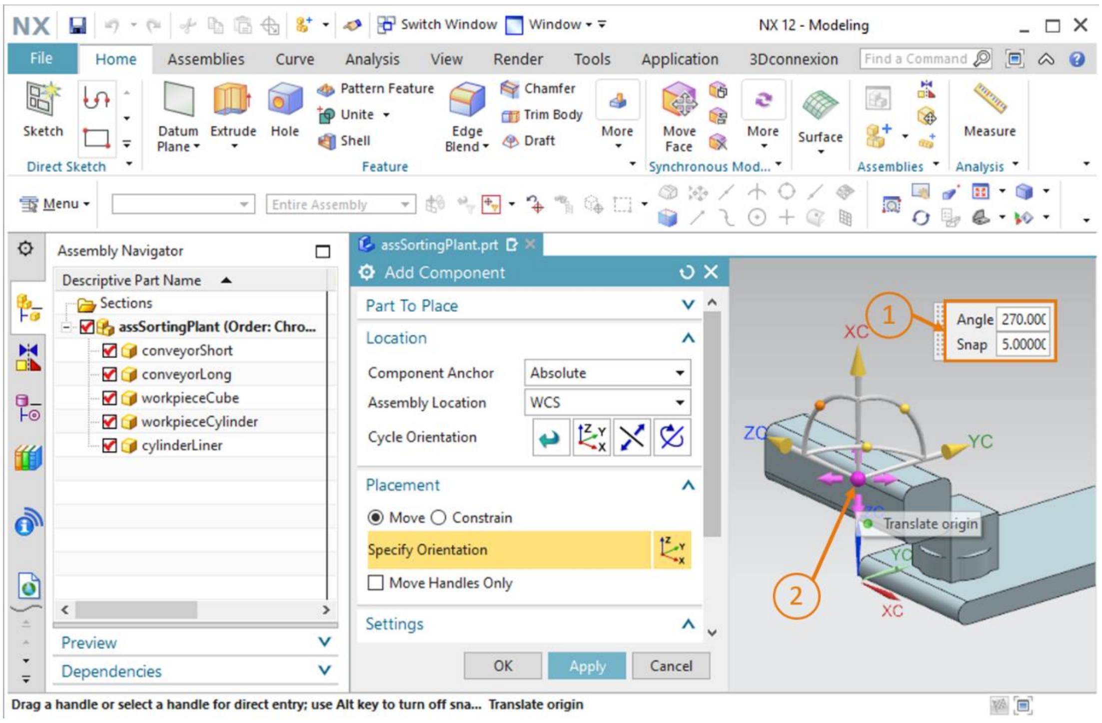


Figure 61: Rotating the "cylinderLiner" component – Specifying the rotation angle

→ Specify the following values for the base of the thrust cylinder (see Figure 62, step 1):

- X value = **125.5 mm**
- Y value = **307.5 mm**
- Z value = **24.0 mm**

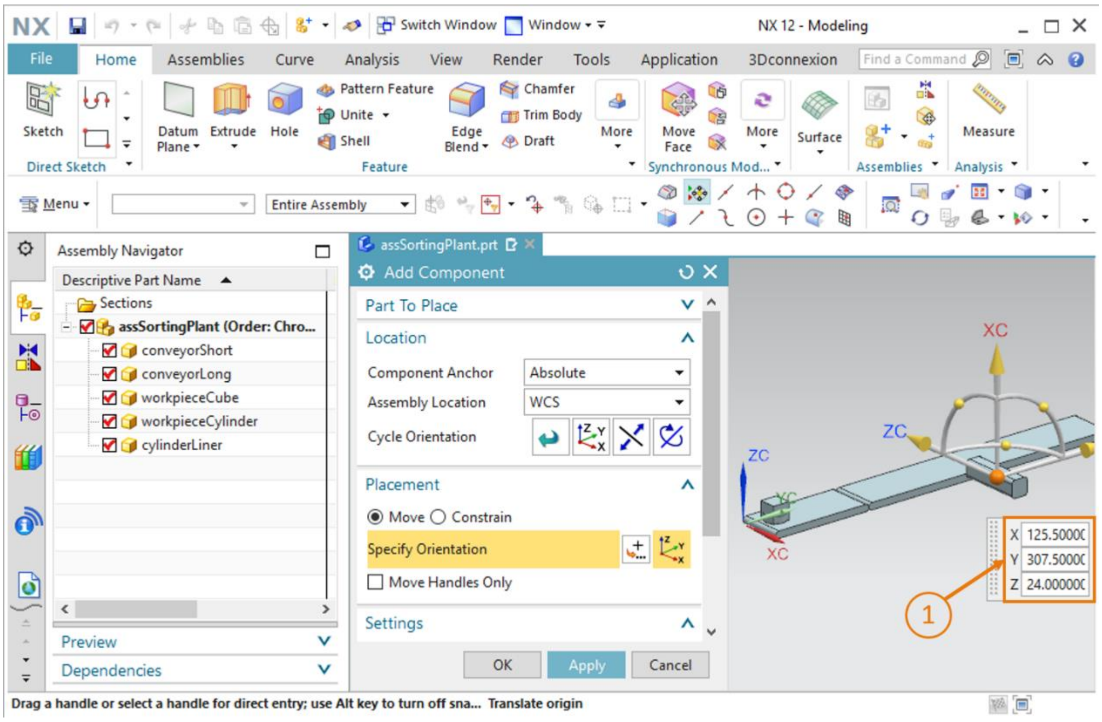


Figure 62: Positioning the "cylinderLiner" model in the assembly

→ Finally, make sure again that only the **Model** is selected as the **Reference Set** in the "Settings" submenu.

When you have successfully added the base to the assembly, save the sorting plant.

Inserting and positioning the head in the assembly:

The procedure used for positioning the base can also be used for positioning the head.

→ In the "Part To Place" submenu of the "Add Component" window, select the "**cylinderHead**" model from your working directory.

→ Next, rotate the head around the Y axis by 270°, as described previously for the base of the thrust cylinder.

→ Then, move the component to the following space coordinates (see Figure 63, step 1).

- X value = **112.0 mm**
- Y value = **307.5 mm**
- Z value = **24.0 mm**

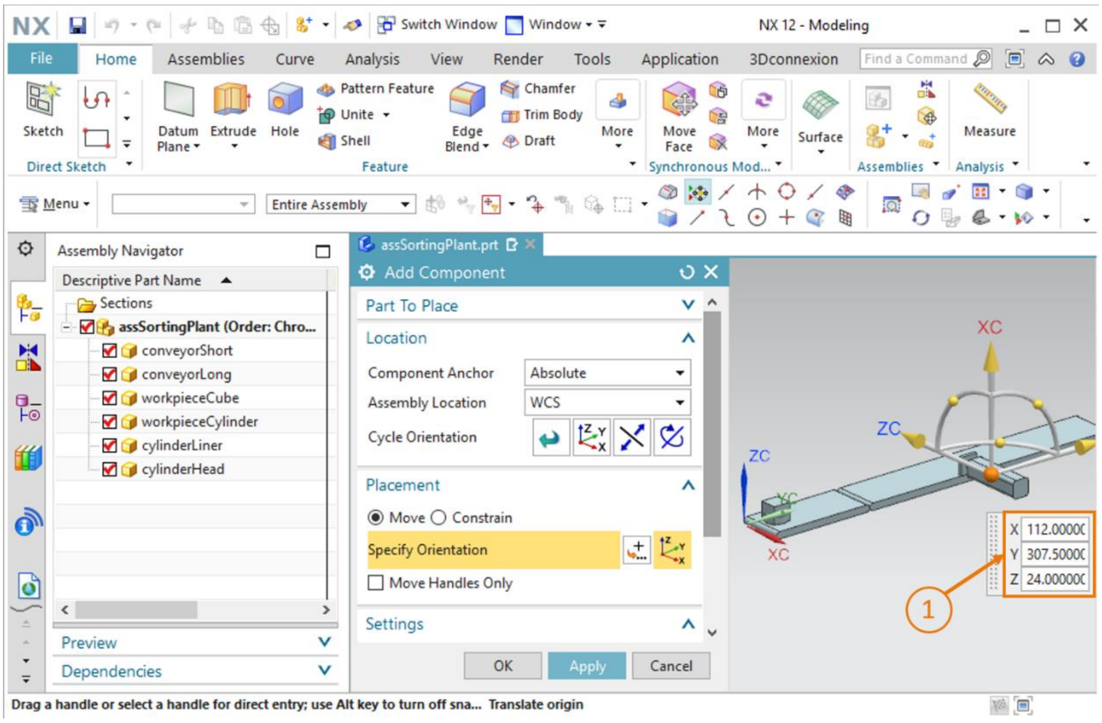


Figure 63: Positioning the "cylinderHead" model in the assembly

→ Here, as well, the selected reference set in the "Settings" submenu should contain only the model.

You have now inserted the thrust cylinder as a static model in the assembly. You should save the assembly to conclude this chapter.

NOTE

Besides the option of specifying fixed space coordinates and a fixed rotation, it is also possible to use so-called "Constraints" to give models a certain orientation. For example, these allow two circular surfaces to be positioned concentric to one another. You can also specify which surfaces are to be parallel or orthogonal to one another.

This is not trivial, however, and requires proficiency with NX. For further information you can search in the online help of NX (see Chapter 9, Link [2]).

7.2.7 Inserting and positioning both containers

Two identical storage containers are used for sorting the different workpieces. You have already prepared a model for these with the name "container" in Chapter 7.1.5. You are to now insert this model into the sorting plant.

Positioning the first container for the "workpieceCylinder" workpieces:

The first container is to be placed directly next to the conveyor belt at the position where, the thrust cylinder ejects the cylinder workpieces. To insert the first container, follow the procedure described in Chapter 7.2.2, "**Section: Inserting and positioning a model"**, with the following adjustments:

→ Select your "**container**" model for the part to be placed.

→ Enter the following coordinates for the position is space, as shown in Figure 64, step 1:

- X value = **-32.0 mm**
- Y value = **307.5 mm**
- Z value = **-42.0 mm**

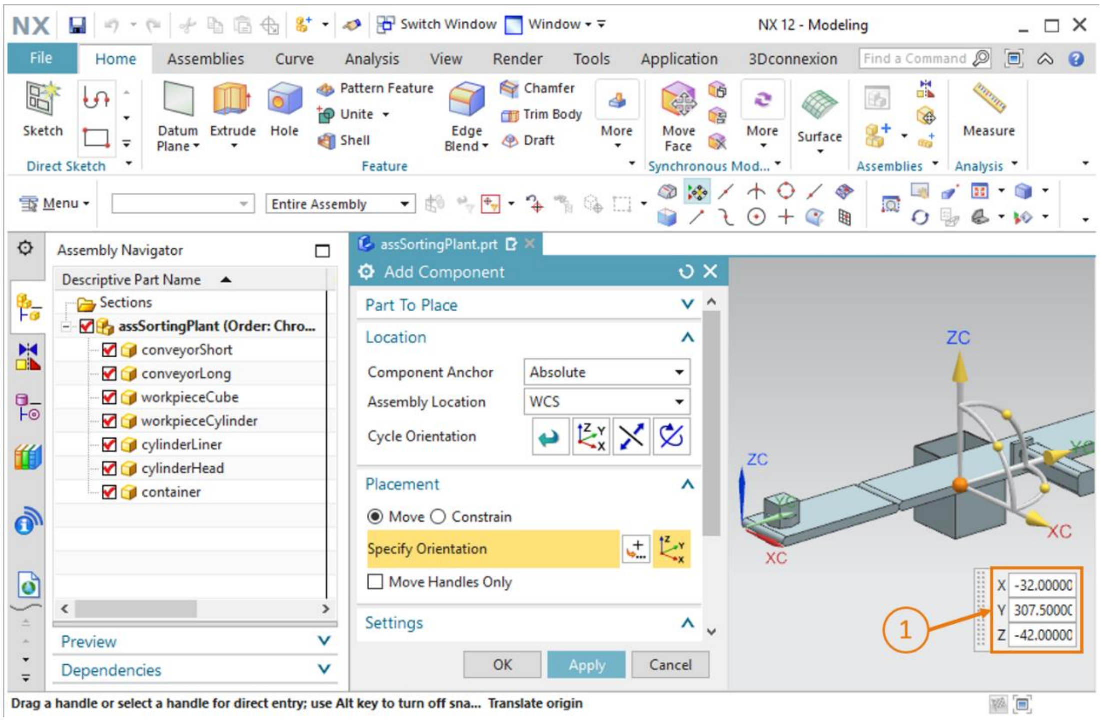


Figure 64: Positioning the "container" model in the assembly

Positioning the second container for the "workpieceCube" workpieces:

To add a second container, you can copy the first container. This is possible because the model is already a known part within the assembly. Proceed as follows for this:

→ In the Assembly Navigator, select your inserted "container" module in your "assSortingPlant" assembly (see Figure 65, step 1). Then, click on the "Add Component" button again (see Figure 65, step 2). In this case, the "Part To Place" has already been automatically selected. Now, follow the usual procedure in the "Location" and "Placement" submenus (see Figure 65, steps 3-5) but replace the position coordinates as follows:

- X value = **32.5 mm**
- Y value = **572.5 mm**
- Z value = **-42.0 mm**

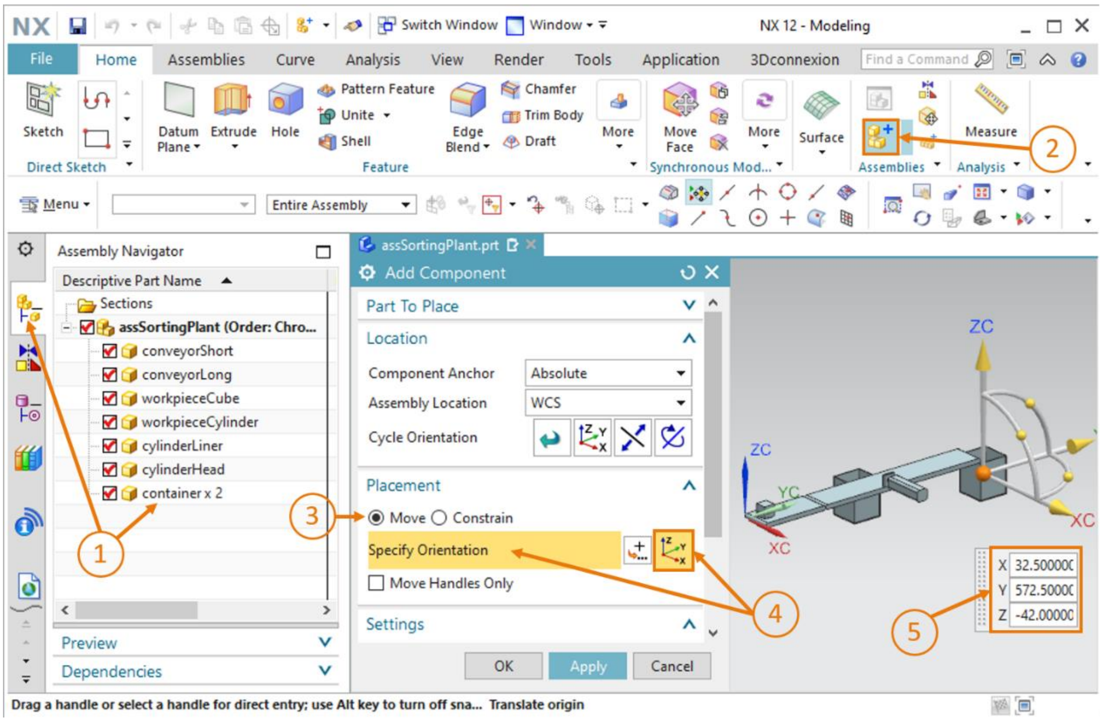


Figure 65: Copying and positioning the second "container" model in the assembly

You have inserted the two storage containers as static models into the sorting plant. Conclude by saving the assembly.

7.2.8 Inserting and positioning the "Workpieces" light sensor

Various light sensors must also be added to the assembly for detecting the different workpieces. At the end of the short conveyor belt "ConveyorShort" there is a light sensor for counting all the workpieces that run through the sorting process.

For this, you use the "**lightSensor**" and "**lightRay**", models that were prepared for you (see Chapter 7.1.8). In this case, the "lightSensor" model is used twice as a photoelectric sensor with counterpart, which act as a transmitter and receiver for example. The "lightRay" model is needed once for representing a light beam. Proceed as follows for this:

→ Add the first part of the light sensor to the assembly. Follow the procedure described in Chapter 7.2.2, "**Section: Inserting and positioning a model**". This time, use the "**lightSensor**" model as the "Part To Place" and orient it according to the following space coordinates (see Figure 66, step 1):

- X value = **70.0 mm**
- Y value = **130.0 mm**
- Z value = **15.0 mm**

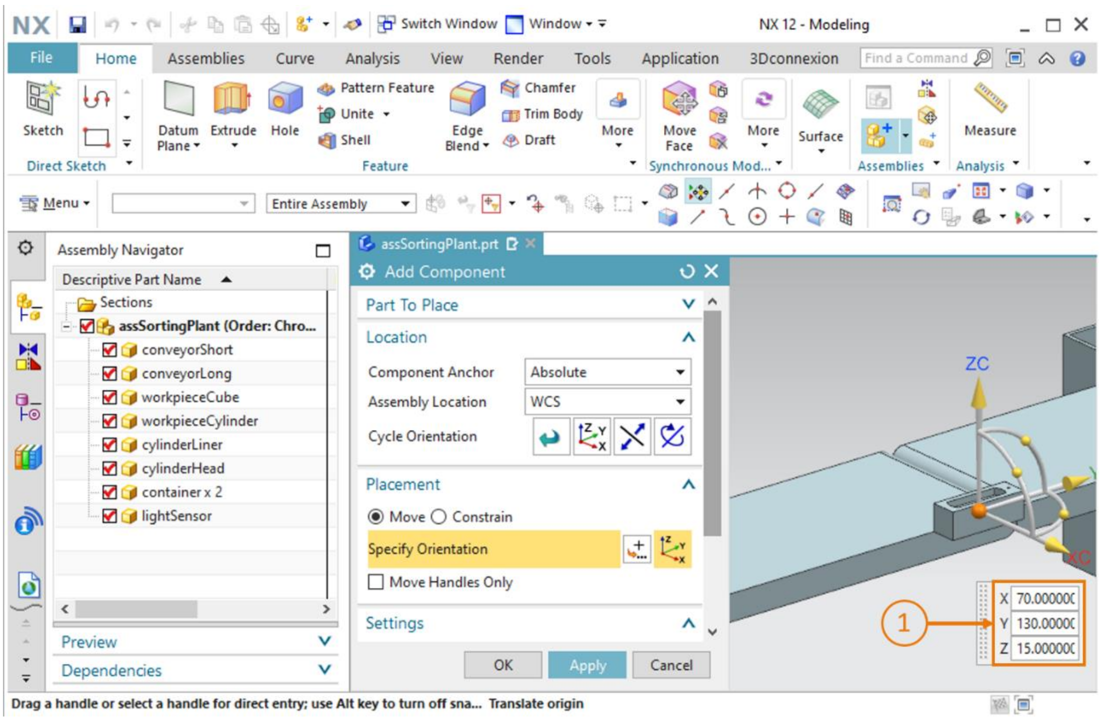
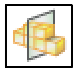


Figure 66: Positioning the first "lightSensor" model in the assembly

→ Next, you are to insert the counterpart for the photoelectric sensor. This is inserted using the same model from the preceding step. However, this model is now to be mirrored. For this, you open the "**Mirror Assembly**" command  in the assembly functions. The "Mirror Assemblies Wizard" appears, which will guide you through the mirroring process. Click on the "**Next**" button in the Welcome window, as shown in Figure 67, step 1.

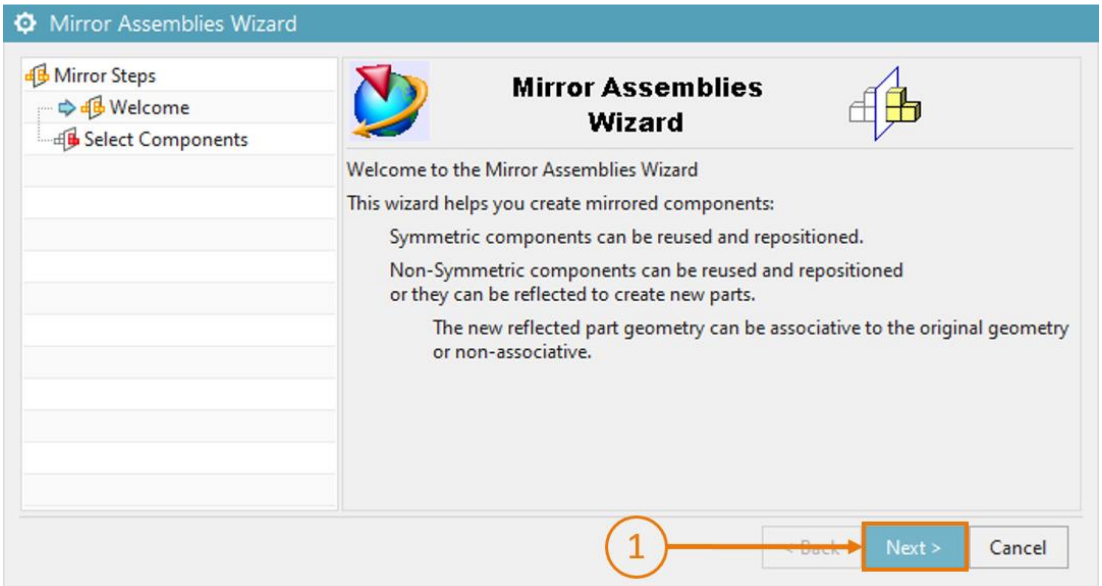


Figure 67: Mirroring the light sensor – Welcome page

→ You are to select the component to be mirrored in the next window. Navigate to the Assembly Navigator in the Resource bar, and select the "**lightSensor**" model there, (see Figure 68, step 1). This should now be indicated as a selected component in the wizard. Continue by clicking the "**Next**" button (see Figure 68, step 2).

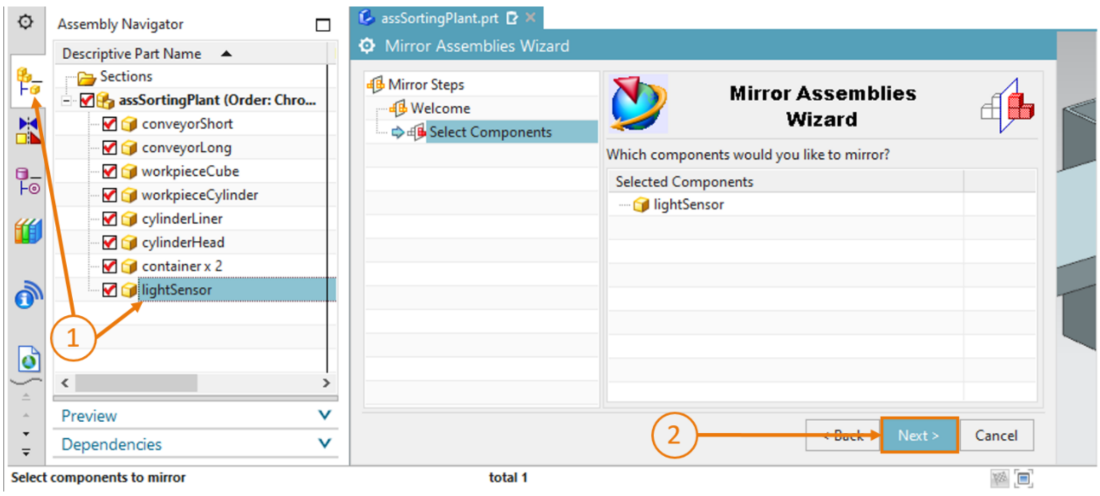
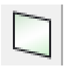


Figure 68: Mirroring the light sensor – Selecting a component

→ In the next window, you select a plane on which the previously selected component is to be mirrored. For this, you click on the mirror symbol  (see Figure 69, step 1) so that you can select an appropriate plane in the three-dimensional graphics window.

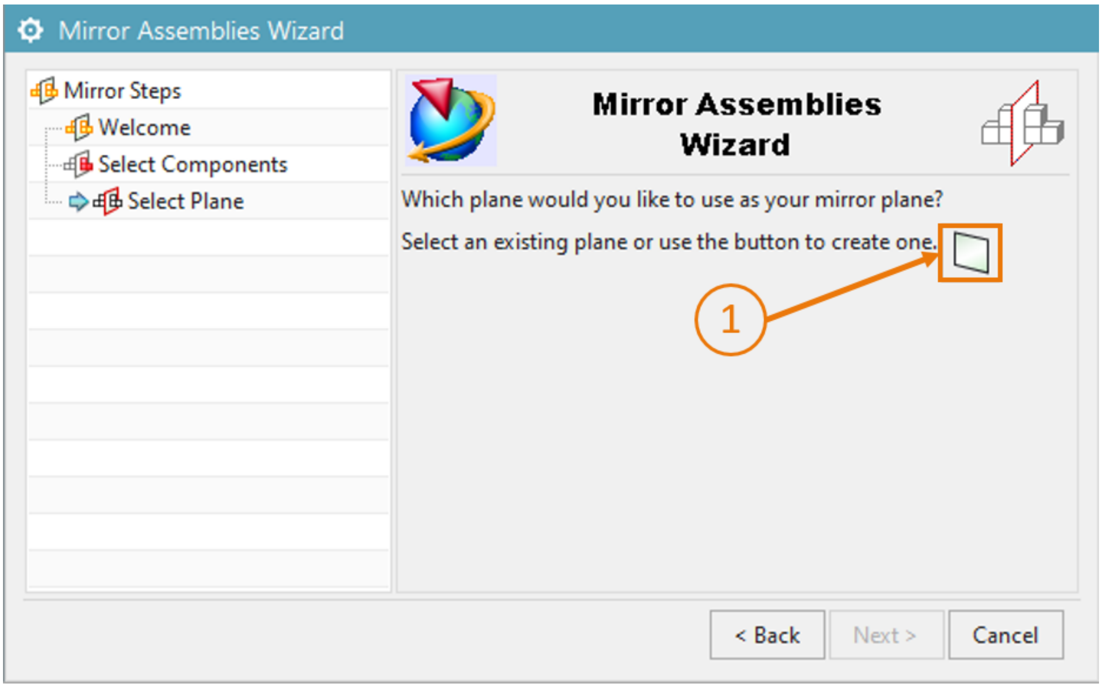


Figure 69: Mirroring the light sensor – Selecting the mirror plane

→ You now see the "**Datum Plane**" command window. First, switch from the trimetric view to the "**Front**" view (see Figure 70, step 1). Specify the "**YC-ZC Plane**" as the datum plane (see Figure 70, step 2). In the "**Offset and Reference**" submenu, define "**WCS**" as the input method with a defined distance of **32.5 mm** (see Figure 70, step 3). This corresponds to half the width of the conveyor belts. Confirm your selection by clicking "**OK**" (see Figure 70, step 4).

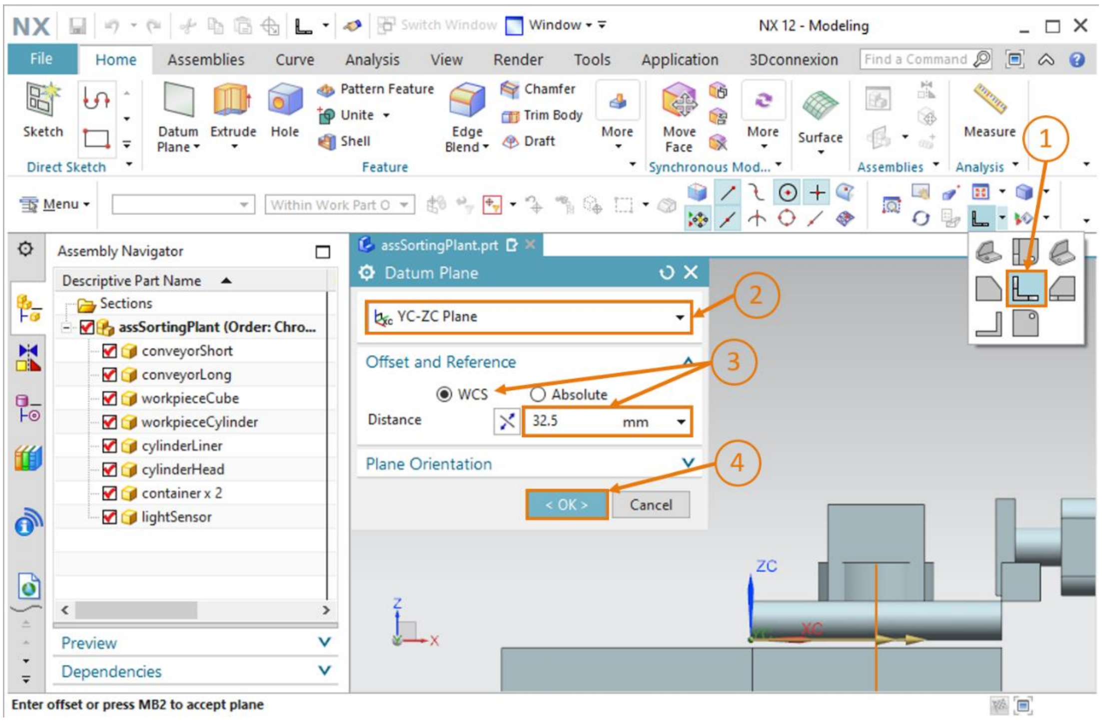


Figure 70: Mirroring the light sensor – Setting the datum plane

→ You are then returned to the Select Plane window shown in Figure 68. Click "**Next**" in order to select the previously selected datum plane for this mirroring operation.

→ In the next window, you now need to specify how the mirrored body is to be named because it will be saved as a new model in a separate file. For the naming rule, specify that the **suffix** "**_mirror**" is to be added to the original file name (see Figure 71, step 1). Store the model in the same working directory as the model of the light sensor that is being mirrored (see Figure 71, step 2) and continue by clicking the "**Next**" button (see Figure 71, step 3).

Figure 71: Mirroring the light sensor – Naming policy for new model

→ You must now specify the mirror type. This is necessary for adapting the sketches prepared for the source to the mirrored model. For this, you select the "**lightSensor**" model in the selection menu and click on the button for "**Associative mirroring**" (see Figure 72, step 1). Continue by clicking the "**Next**" button (see Figure 72, step 2).

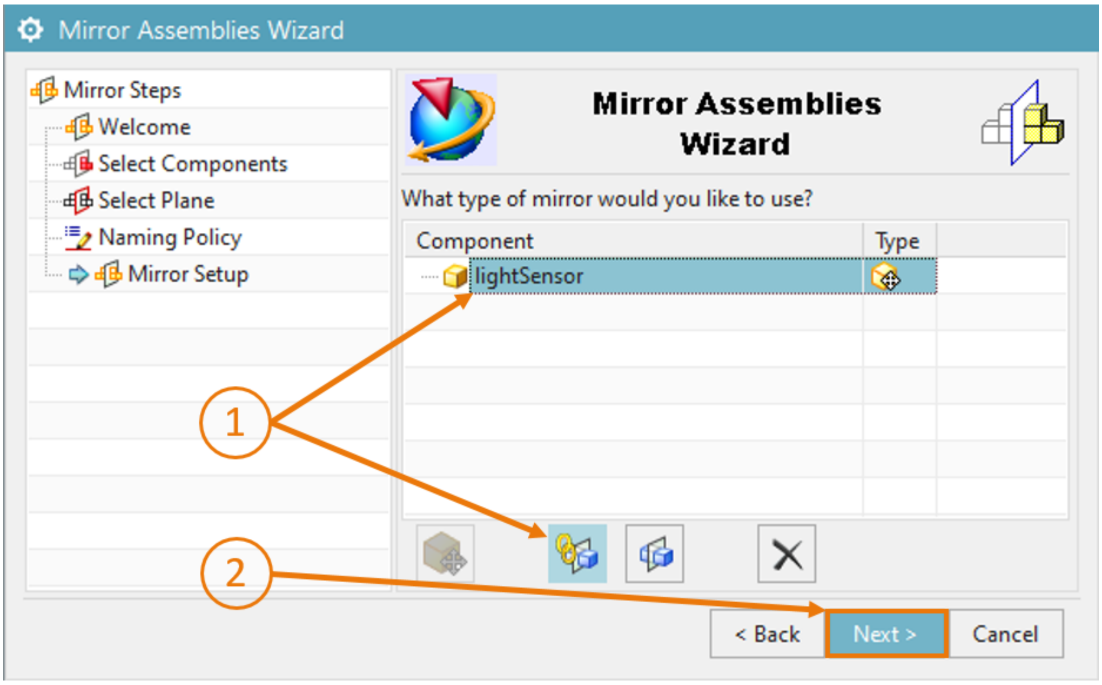


Figure 72: Mirroring the light sensor – Specifying the mirror type

→ A notice then appears that new part files will be generated. Confirm this by clicking on "**OK**".

NOTE

You may get an information window with several warnings indicating that the parent relationships of planes, vectors and points of various prepared sketches have been suppressed or removed. As a result of the mirroring, references to the coordinate system of the model may be lost.

You can ignore these warnings in this example because no other adjustments to the model are needed.

In the next window, you must confirm the positioning of the mirrored components again. To do so, check the position in the three-dimensional graphics window again and confirm it by clicking "**Next**", as shown in Figure 73, step 1.

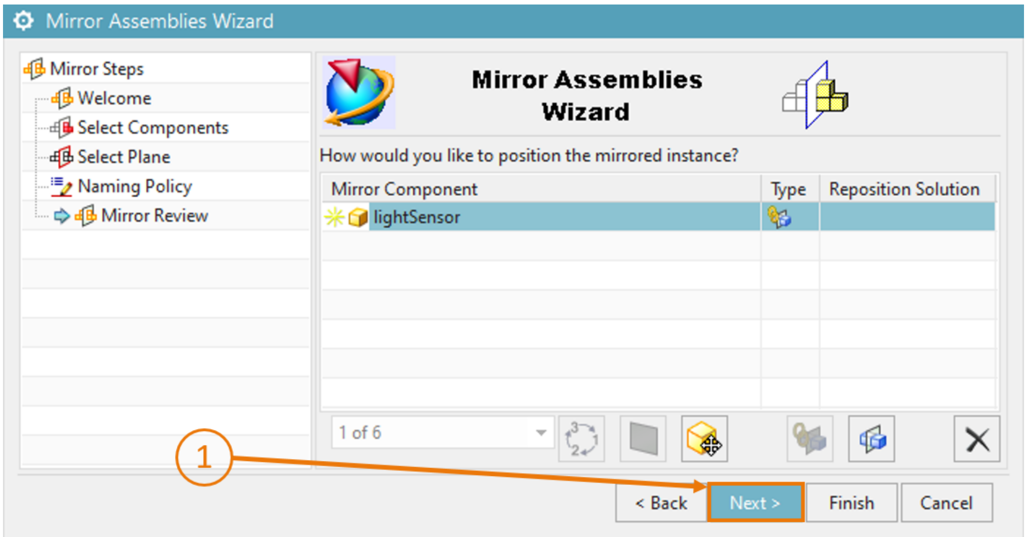


Figure 73: Mirroring the light sensor – Checking the positioning of the mirrored component

→ The new file name and the name of the source file are then displayed. If the new file name is "lightSensor_mirror" and stems from the source file "lightSensor", close this wizard by clicking the "**Finish**" button (see Figure 74, step 1). If there are discrepancies in the new file name, correct your naming rules from before. If you have selected the wrong source file, you need to select a different component in the "Select Components" step of the wizard (see also Figure 68).

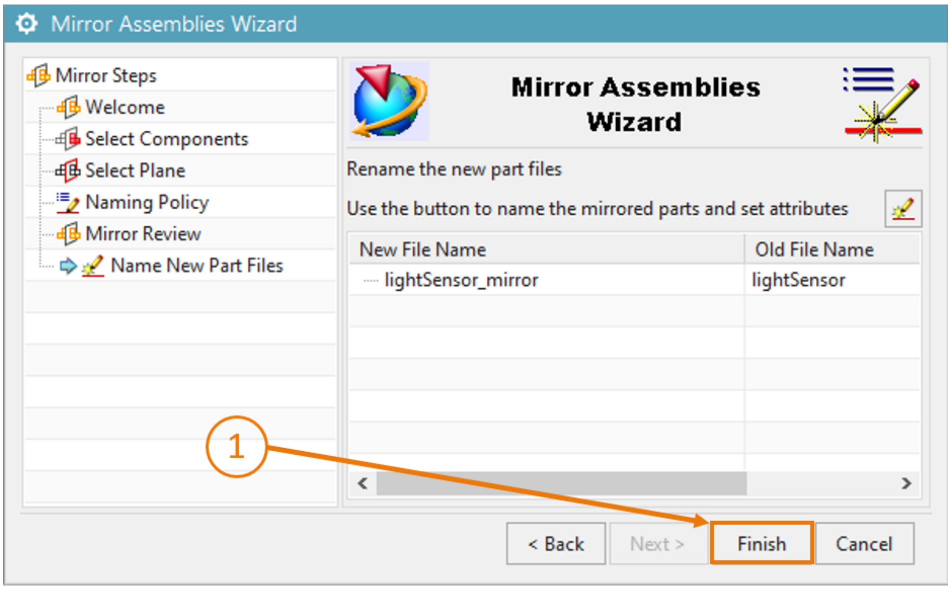


Figure 74: Mirroring the light sensor – Checking the new model name

→ To complete the light sensor, you also need the light beam. This is to be positioned between the light sensor and its counterpart. For this purpose, the "lightRay" model must be added and arranged.

Again, begin by first adding the component. You can use Chapter 7.2.2 as a guide again here. But this time, select the "**lightRay**" model as the "Part To Place". Because the added light beam is perpendicular to the conveyor belt surface, this model must first be rotated, as explained previously in Chapter 7.2.6. Select the "Move" method in the "Placement" submenu and click on "Specify Orientation" (see Figure 75, steps 1+2). You now see the orientation coordinates in space in the three-dimensional graphics window. To rotate the model, first select the point between the X and Z axis (see Figure 75, step 3), which allows you to rotate the component around the Y axis.

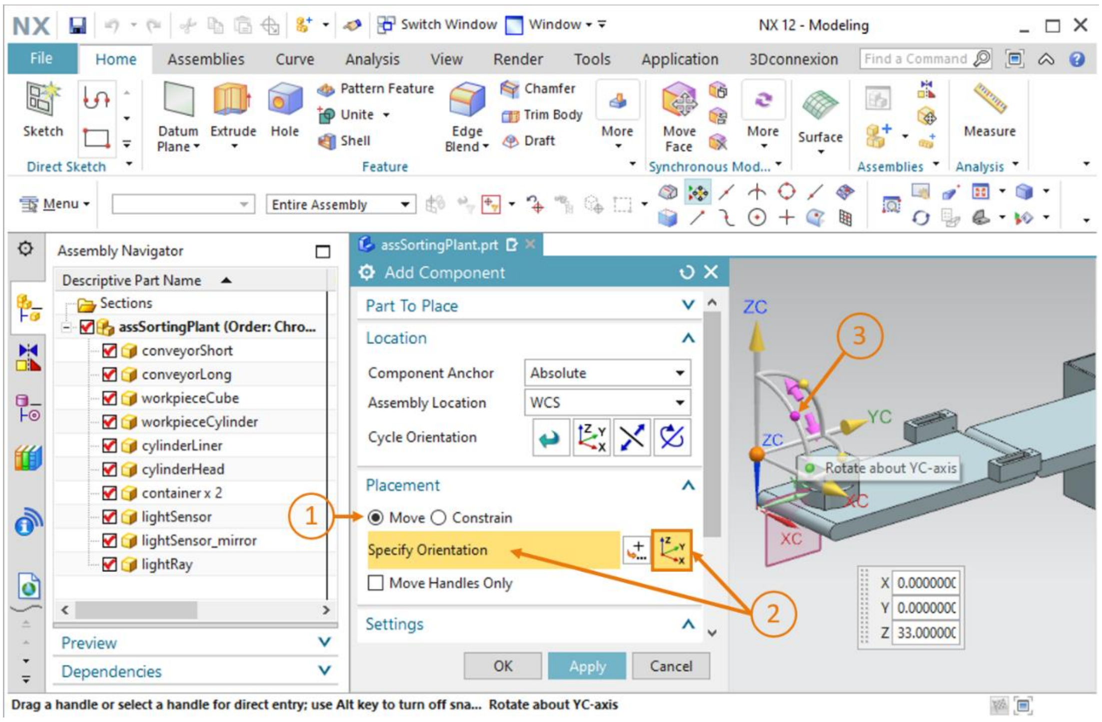


Figure 75: Adding the light beam for the light sensor – Selecting a rotation

→ You must then bring the light beam to a horizontal position. For this, you specify an angle of **90°** (see Figure 76, step 1). Now click on the center point again in order to move the component within the world coordinate system (see Figure 76, step 2).

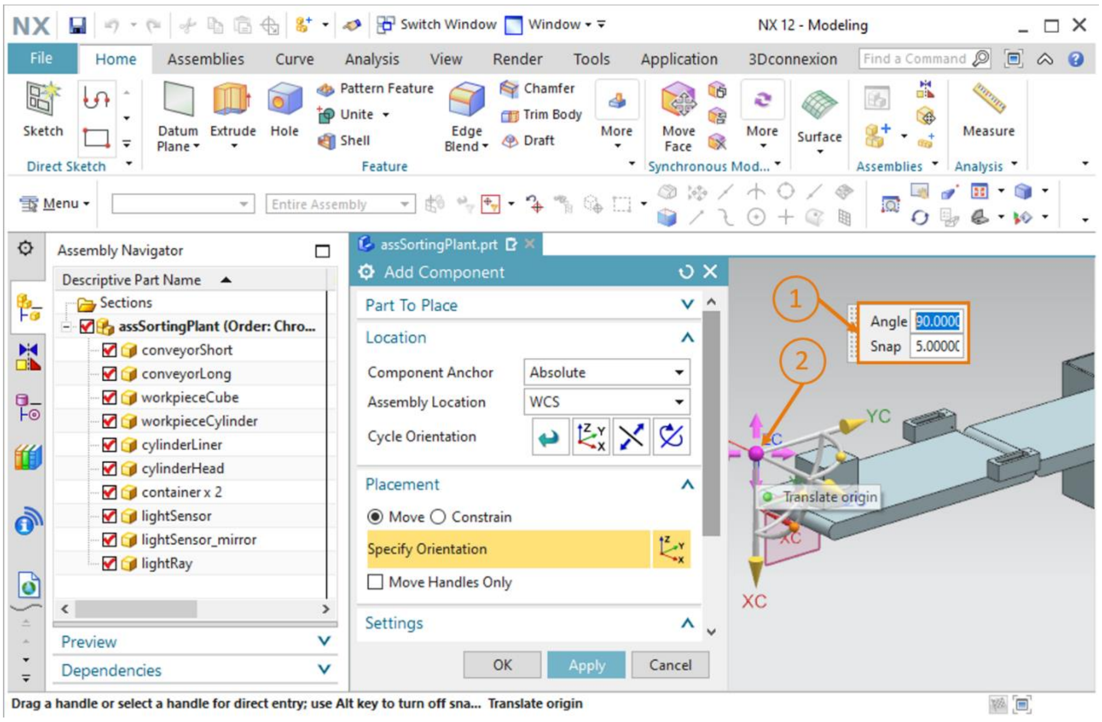


Figure 76: Adding the light beam for the light sensor – Specifying a rotation

→ Then, specify the following space coordinates for the light beam, as shown in Figure 77, step 1.

- X value = **32.5 mm**
- Y value = **142.5 mm**
- Z value = **12.5 mm**

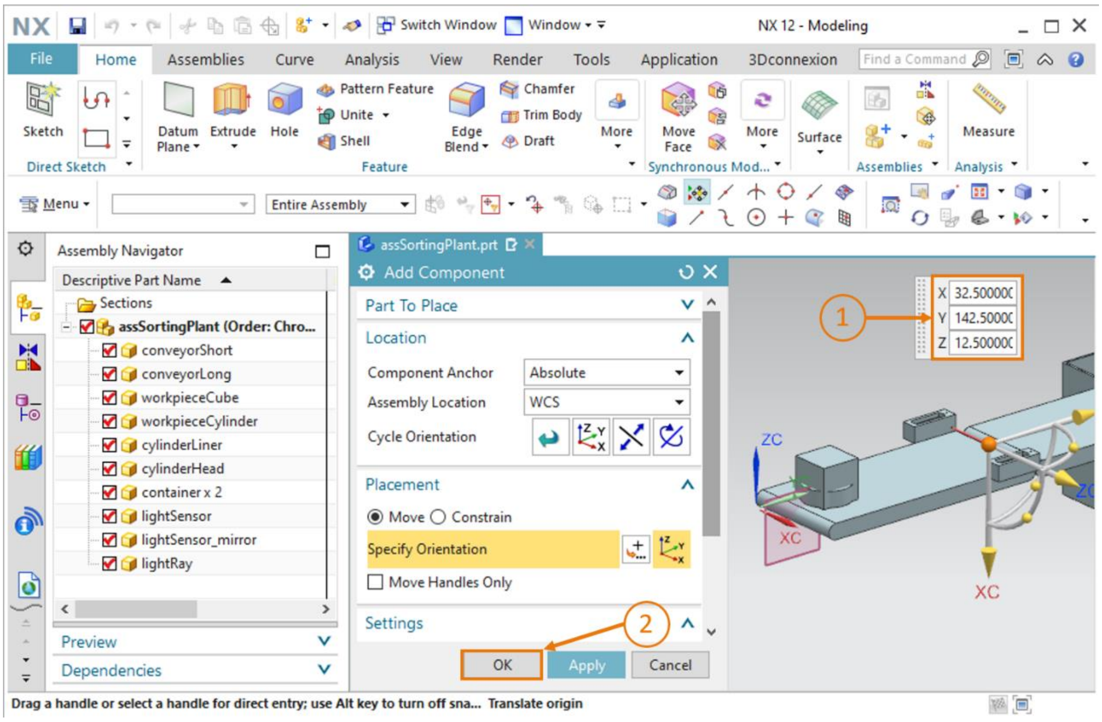


Figure 77: Adding the light beam for the light sensor – Specifying a position

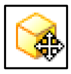
→ Confirm you input by clicking the "**OK**" button (see Figure 77, step 2).

You have now successfully inserted the first light sensor and you should save the sorting plant.

7.2.9 Inserting and positioning the cylinder light sensor system by inserting and positioning

As already described in the preceding modules of this workshop series, two light sensors arranged one on top of the other are needed directly in front of the ejector for detecting the "workpieceCylinder" workpieces. Based on the height difference of the two workpieces, the "workpieceCylinder" workpieces can be identified with absolute certainty by this light sensor system.

Because you have already inserted a finished static model of the light sensor at the end of the short conveyor belt in Chapter 7.2.8, you can use this for creating the light sensor system for detecting the "workpieceCylinder" workpieces. You are therefore to copy and move the model at the end of the short conveyor belt.

→ To do this, open the "**Move Component**" command  in the assembly functions (see Figure 78, step 1) and click on the "**Select Components**" button in the "Components to Move" submenu (see Figure 78, step 2). Then, in the Assembly Navigator, select the models you added in Chapter 7.2.8, (see Figure 78, step 3): **lightSensor**, **lightSensor_mirror** and **lightRay**.

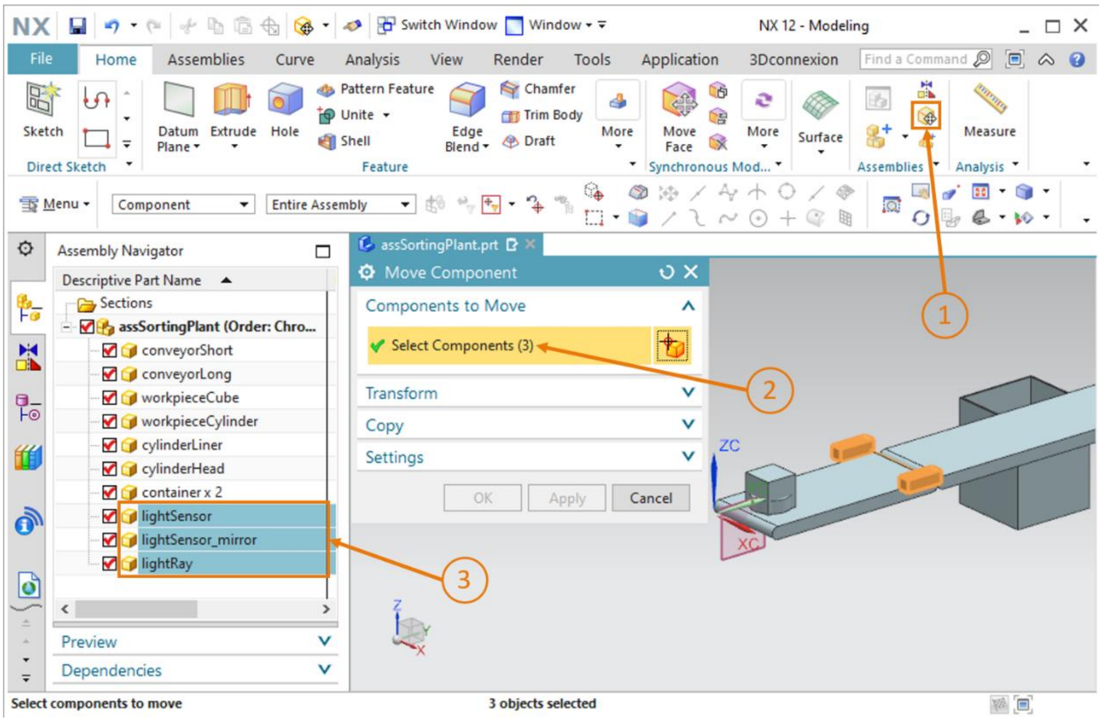


Figure 78: Moving a light sensor – Selecting the model

→ You must now activate the copying of components. Appropriate settings for copying can be made in the "**Copy**" submenu. Select "**Copy**" mode (see Figure 79, step 1) and click on the "**Select Components**" button in the "Components to Copy" menu command (see Figure 79, step 2). Then, once again select the three models for the light sensor in the Assembly Navigator so that all three models will be copied for moving (see Figure 79, step 3).

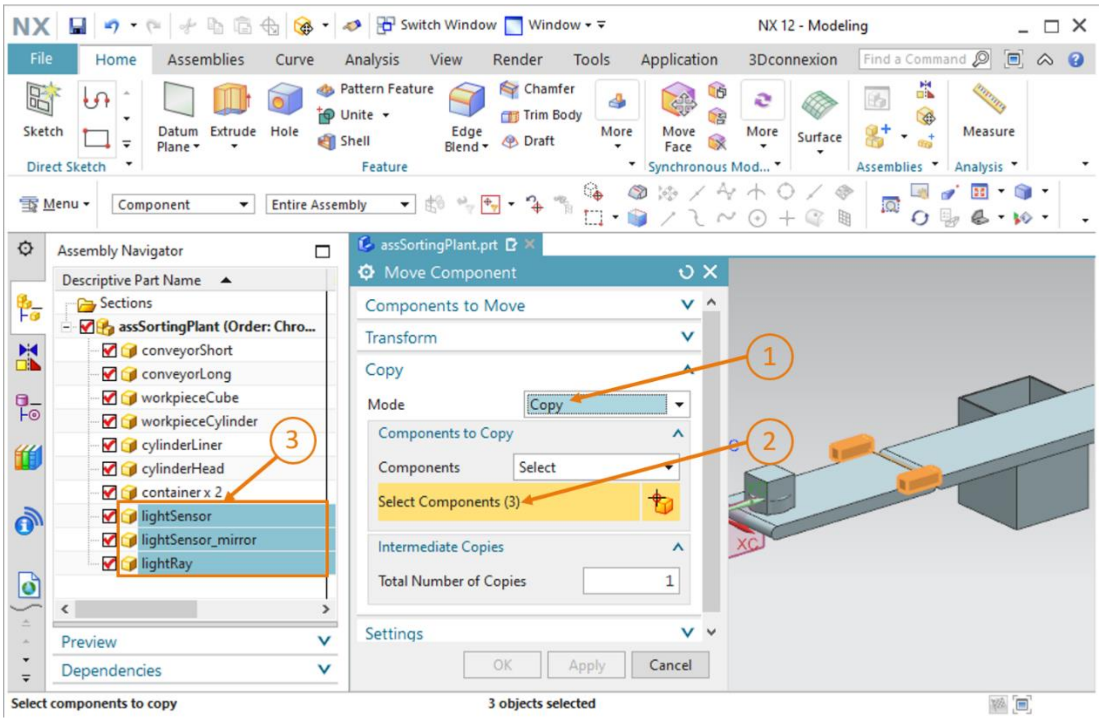


Figure 79: Preparation for copying the light sensor

→ Then, go to the "**Transform**" submenu and click on "**Specify Orientation**" (see Figure 80, step 1). Here, you should be able to set the space coordinates in the three-dimensional graphics window. Use the following coordinate values for this as indicated in Figure 80, step 2.

- X value = **32.5 mm**
- Y value = **260.0 mm**
- Z value = **15.0 mm**

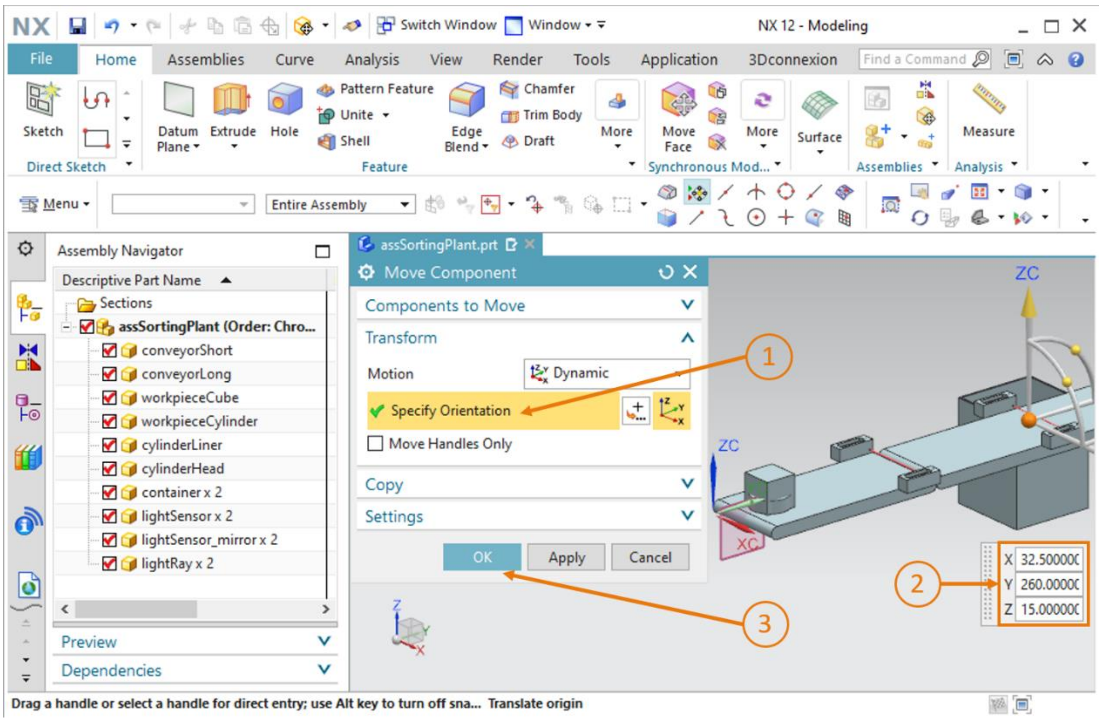


Figure 80: Copying the light sensor to a new position

→ Confirm you inputs by clicking the "**OK**" button (see Figure 80, step 3). You have now copied the first of the two light sensors for detecting the "workpieceCylinder" workpieces to the appropriate position.

→ You can proceed as described above for the second light sensor. However, when selecting the suitable models in the Assembly Navigator, the addition of "**x 2**" to the model name should make you aware that you are now selecting both light sensors at the same time. This is due to more than one model of the same type being packed. To unpack these models again, select all the corresponding models in the Assembly Navigator, as shown in Figure 81, step 1. Then, right-click the selection, and select the "Unpack" command in the shortcut menu. This applies to the **lightSensor**, **lightSensor_mirror** and **lightRay** models in this case.

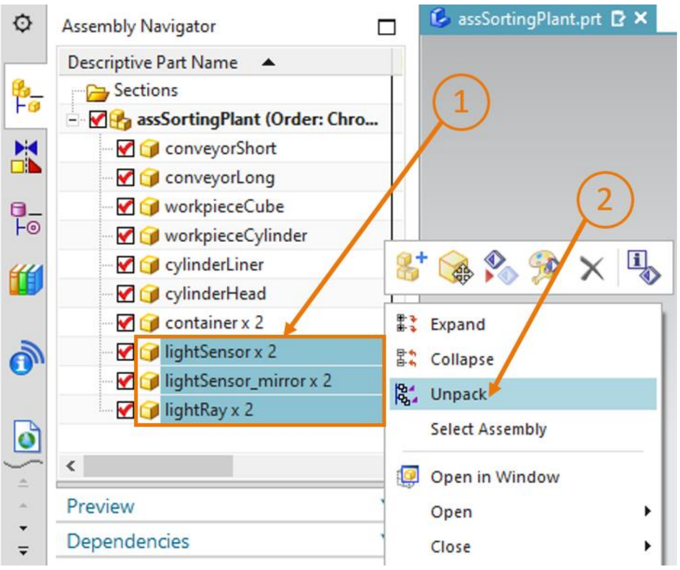


Figure 81: Unpacking models of the same type in an assembly

→ You should then find the unpacked components in the Assembly Navigator. Instead of the models being listed once with the addition "x 2", the models are now listed twice. First, select the models that you have created for the first light sensor of this light sensor system (see Figure 82, step 1). You can verify the selections in the graphics window because the selected parts appear in orange color, as highlighted on the right side in Figure 82.

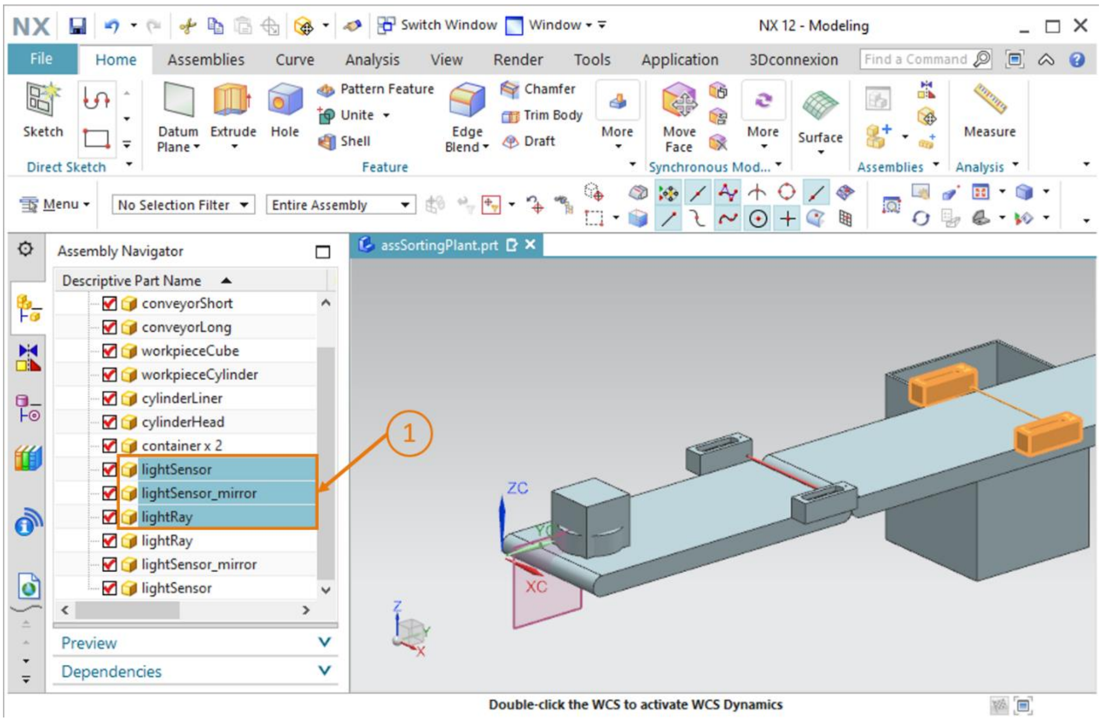


Figure 82: Selection of the components to be copied

Finally, open the "**Move Components**" command window again and follow the same procedure as for the first light sensor of the light sensor system. The submenu "Components to Move" is already automatically defined based on your previous selection of models. Select precisely these models as the "**Components to Copy**", and specify the following spatial arrangement in the "Transform" submenu (see Figure 83, steps 1+2):

- X value = **32.5 mm**
- Y value = **260.0 mm**
- Z value = **25.0 mm**

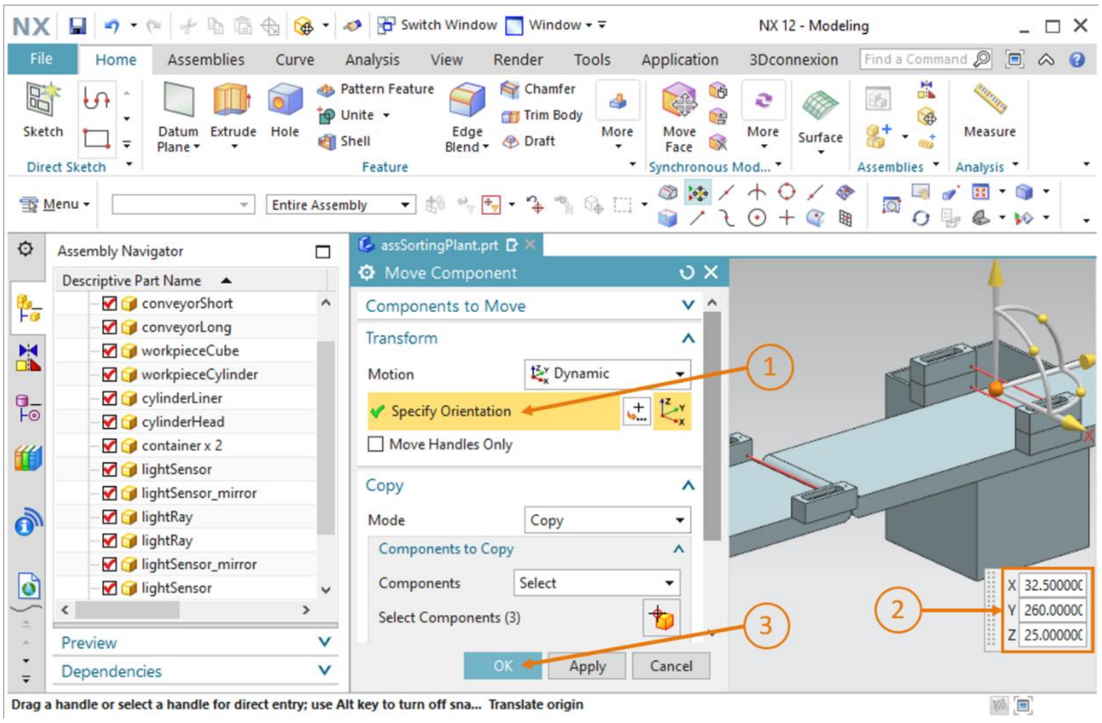


Figure 83: Copying and positioning the second light sensor above the first light sensor

→ Confirm your arrangement by clicking the "OK" button (see Figure 83, step 3).

You have now defined the light sensor system for detecting the "WorkpieceCylinder" workpieces as a static model. Save the assembly in its current state.

7.2.10 Inserting and positioning the cube light sensor

As the final static model, you need to insert the cube light sensor that is to count the "workpieceCube" workpieces at the end of the long conveyor belt. Because all "workpieceCylinder"-type workpieces will have already been detected and sorted out with the cylinder light sensor system, only "workpieceCube"-type workpieces can reach the end of the conveyor belt. Therefore, only a single light sensor is required.

→ For this, you copy the light sensor at the end of the short conveyor belt following the same principle as described in Chapter 7.2.9. But specify the following coordinates for the placement of the light sensor (see Figure 84, steps 1+2):

- X value = **32.5 mm**
- Y value = **520.0 mm**
- Z value = **15.0 mm**

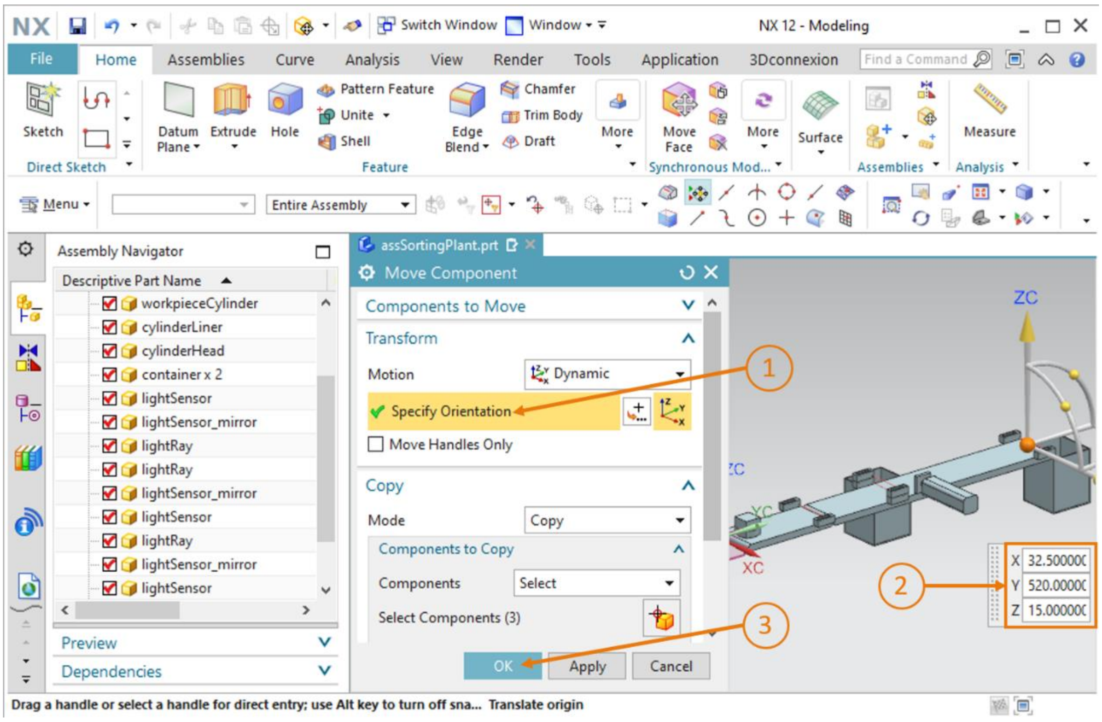


Figure 84: Copying and positioning the cube light sensor

Confirm the copying process by clicking the "**OK**" button (see Figure 84, step 3).

You have now modeled the sorting plant as a static model completely independently and arranged it appropriately in space (see Figure 85). Conclude this module by saving the assembly.

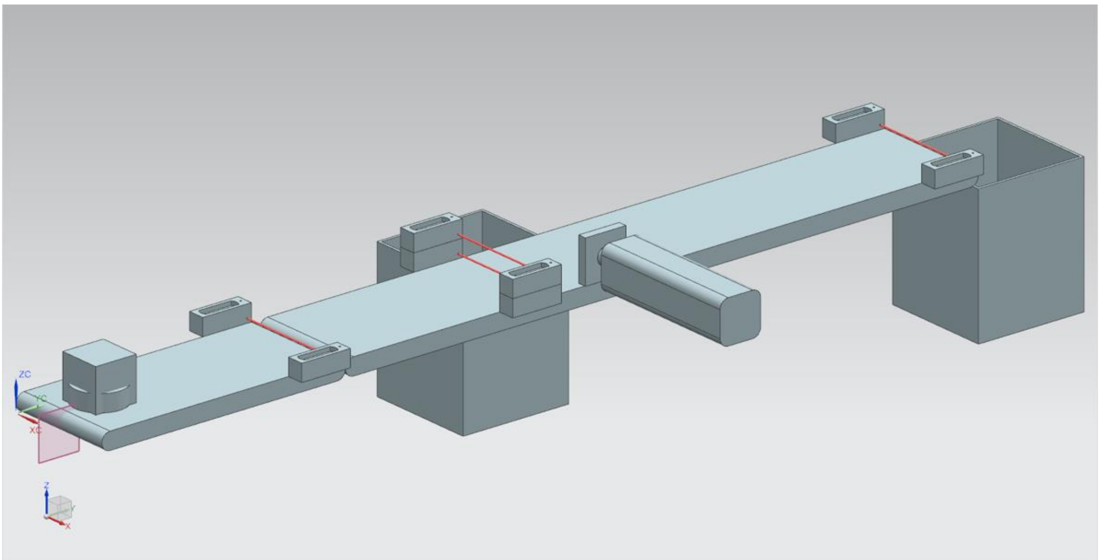


Figure 85: Full view of the static model of the sorting plant in NX

You have now finished working with the basic components of NX, and in future modules you will be able to add dynamic behavior to your static model using the NX Mechatronics Concept Designer application. In this way, you will obtain your complete digital twin of the sorting plant.

7.2.11 Inserting and positioning the limit switches

To conclude, you are to apply various techniques from the preceding chapters for adding the limit switches for the thrust cylinder.

Positioning of the first limit switch:

The first limit switch is to be positioned at the end of the thrust cylinder in order to detect if the head of the thrust cylinder is fully extended. The following steps are needed for this:

→ Open the window for adding a new component (see Figure 86, step 1). Select your "**limitSwitchSensor**" model in the "Part To Place" submenu.

→ The "**limitSwitchSensor**" model is in a vertical position. Rotate it to a horizontal position instead. Select the "**Move**" method in the "Placement" submenu (see Figure 86, step 2) and click on "**Specify Orientation**" (see Figure 86, step 3). For rotating the body, select the point between the Y and Z axis in the three-dimensional graphics window, as shown in Figure 86, step 4. This allows you to rotate the component around the X axis.

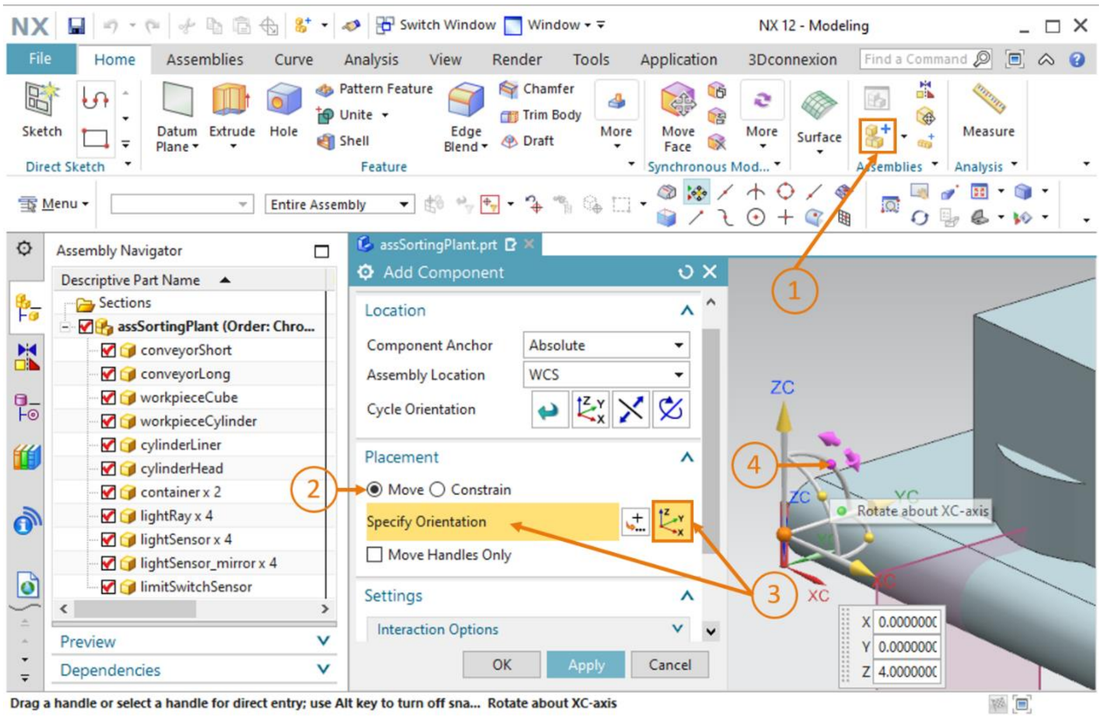


Figure 86: Rotating the "limitSwitchSensor" component – Selecting the rotation axis

→ In the input window displayed, enter an angle of **90.0°** for the horizontal orientation (see Figure 87, step 1). Then click on the center point of the coordinate system in the graphics window again (see Figure 87, step 2), so that the body can be moved via the space coordinates.

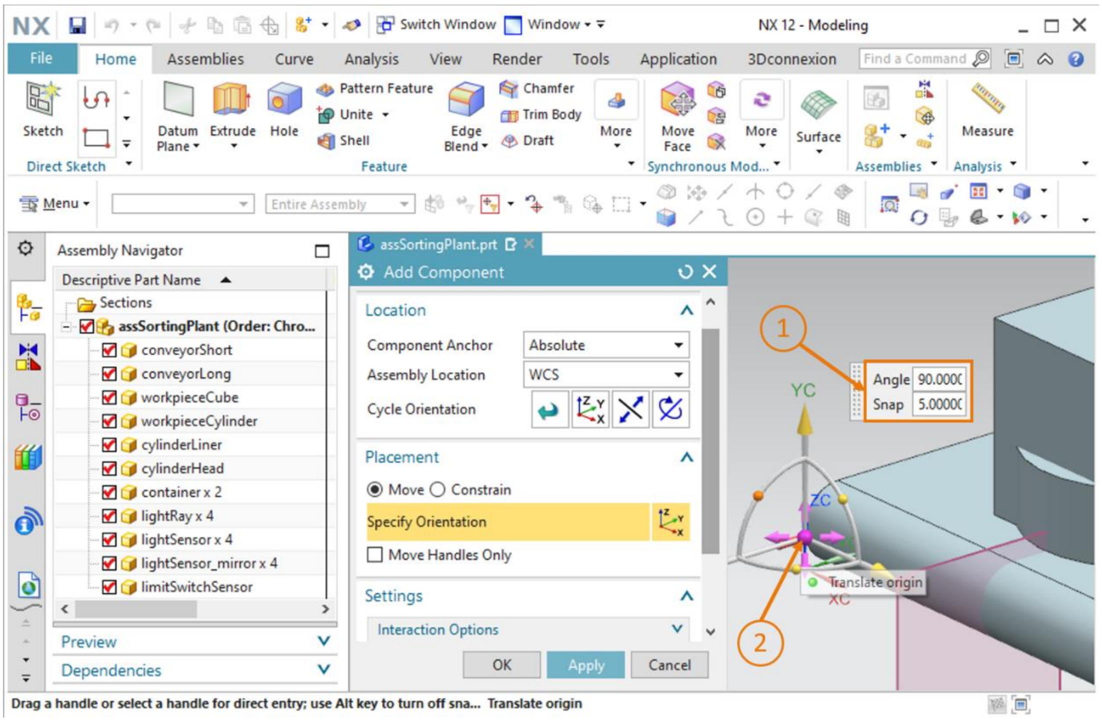


Figure 87: Rotating the "limitSwitchSensor" component – Specifying the rotation angle

→ Specify the following space coordinates for the first limit switch of the thrust cylinder (see Figure 88, step 1):

- X value = **82.0 mm**
- Y value = **307.5 mm**
- Z value = **22.5 mm**

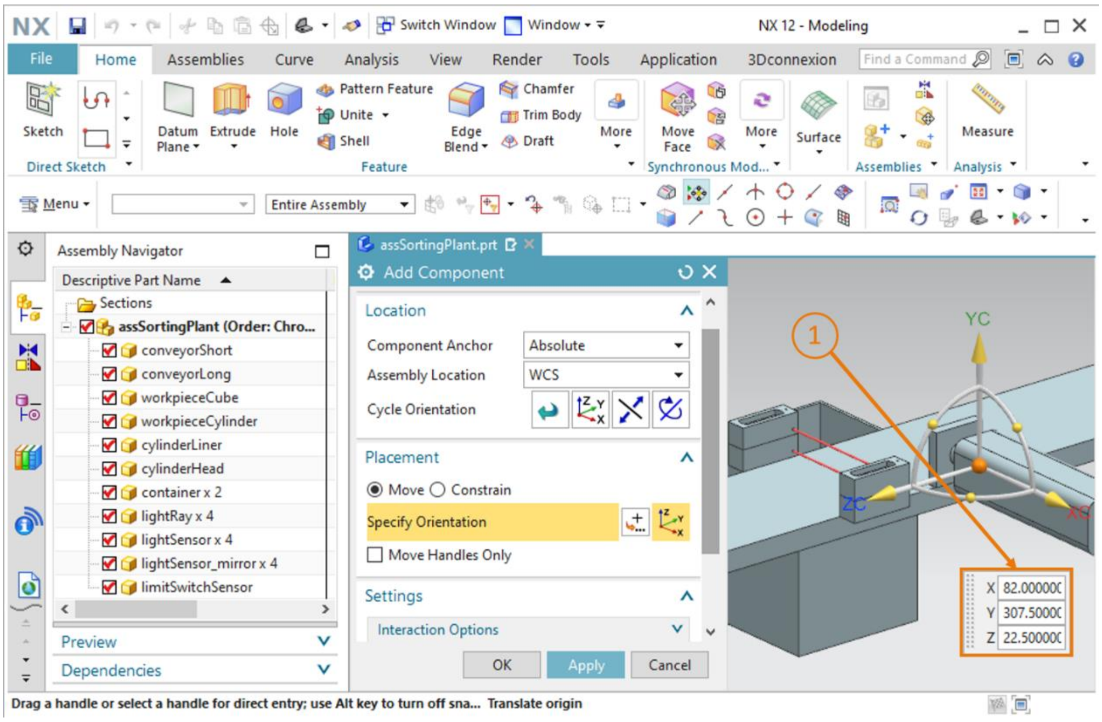


Figure 88: Positioning the "limitSwitchSensor" model in the assembly

→ Finally, make sure again that only the **Model** is selected as the **Reference Set** in the "Settings" submenu.

Positioning the second limit switch:

→ Now insert the second limit switch by copying the first limit switch to your assembly. This is to detect if the head of the thrust cylinder is fully retracted. Follow the procedure explained in Chapter 7.2.9 for copying the model. Select your "**limitSwitchSensor**" model in this case. Also, when selecting the orientation (see Figure 89, step 1), specify the following space coordinates, as shown in Figure 89, step 2:

- X value = **160.0 mm**
- Y value = **307.5 mm**
- Z value = **22.5 mm**

Then confirm your entries with "**OK**" (see Figure 89, step 3).

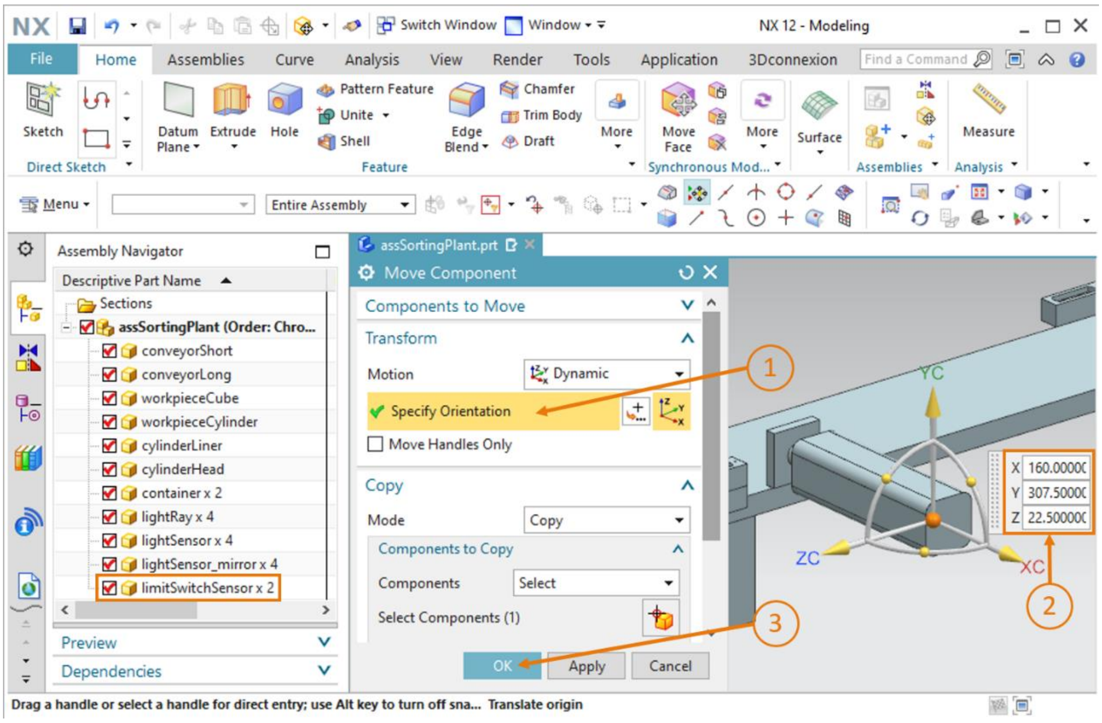


Figure 89: Copying the "limitSwitchSensor" model

The two limit switches for the thrust cylinder have now been successfully added to the assembly. Switch back to the trimetric view and save the project.

You have now modeled the sorting plant as a static model completely independently and arranged it appropriately in space (see Figure 90). Conclude this module by saving the assembly.

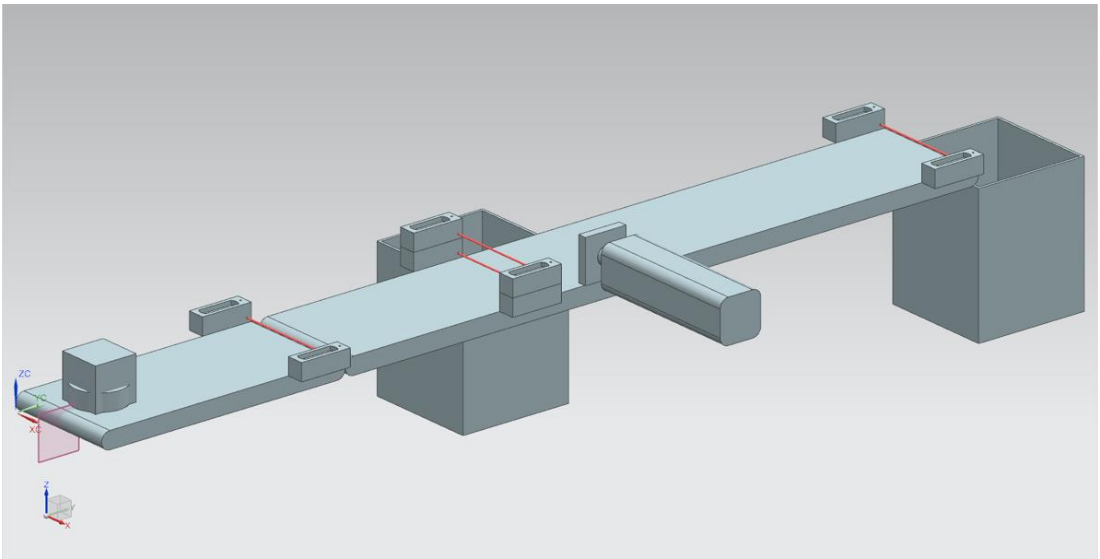


Figure 90: Full view of the static model of the sorting plant in NX

You have now finished working with the basic components of NX, and in future modules you will be able to add dynamic behavior to your static model using the NX Mechatronics Concept Designer application. In this way, you will obtain your complete digital twin of the sorting plant.

8 Checklist – Step-by-step instructions

The following checklist helps students/trainees to independently check whether all tasks in the step-by-step instructions have been carefully completed and enables them to successfully finish the module on their own.

No.	Description	Checked
1	The "workpieceCube" model was successfully modeled in NX.	
2	The modeling of the "workpieceCylinder" model was successfully completed.	
3	The short conveyor belt was successfully modeled.	
4	The long conveyor belt "ConveyorLong" was modeled as a model.	
5	The "Container" was modeled.	
6	The base of the thrust cylinder was successfully modeled.	
7	The head of the thrust cylinder was modeled.	
8	The template files for the photoelectric sensors were copied to your working directory.	
9	An assembly for the overall sorting plant was successfully created.	
10	The model of the "ConveyorShort" conveyor belt was inserted and positioned in the assembly.	
11	The model of the "ConveyorLong" conveyor belt was inserted and positioned in the assembly.	
12	The model of the cube workpiece was placed on the "ConveyorShort" conveyor belt in the assembly.	
13	The model of the cylinder workpiece was placed on the "ConveyorShort" conveyor belt in the assembly.	
14	The thrust cylinder, consisting the head and base, was inserted in the assembly and arranged appropriately.	
15	The container in the "container" model was inserted and placed twice in the assembly.	
16	The workpiece light sensor was added to the assembly and positioned at the end of the short conveyor belt.	
17	The cylinder light sensor system was created by inserting it into the assembly and placing it shortly in front of the thrust cylinder.	
18	The cube light sensor was inserted into the assembly and arranged at the end of the long conveyor belt.	
19	The assembly with the finished static model was saved.	

Table 1: Checklist for "Creation of a Static 3D Model Using the NX CAD System"

9 Additional information

You can find additional information as an orientation aid for initial and advanced training, for example: Getting Started, videos, tutorials, apps, manuals, programming guidelines and trial software/firmware, at the following link:

Preview "Additional information" – In preparation

Here are some interesting links:

[1] support.industry.siemens.com/cs/de/en/view/90885040

[2] support.industry.siemens.com/cs/de/en/view/109756737

[3] omg.org/spec/UML/2.5.1/PDF

[4] geeksforgeeks.org/unified-modeling-language-uml-activity-diagrams/

[5] geeksforgeeks.org/unified-modeling-language-uml-state-diagrams/

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