BCK
SOLUTIONS

GUIDELINE

V3.261 User Manual

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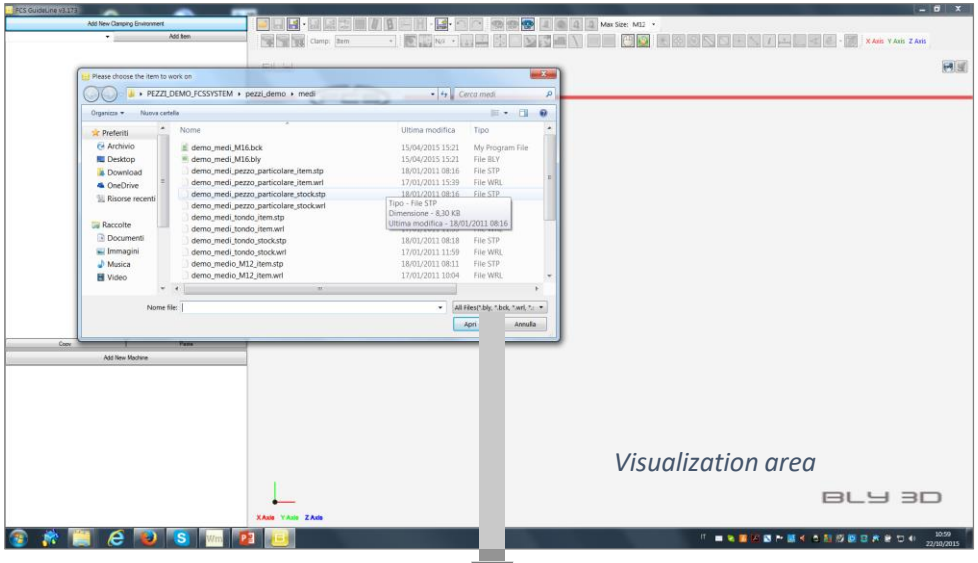
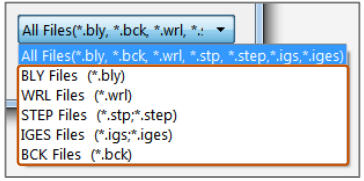
REFERENCE GUIDE - ADDITIONAL COMMANDS

- *Starting a Guideline session*

- *Opening the workpiece's 3D model*



- *Supported file format*

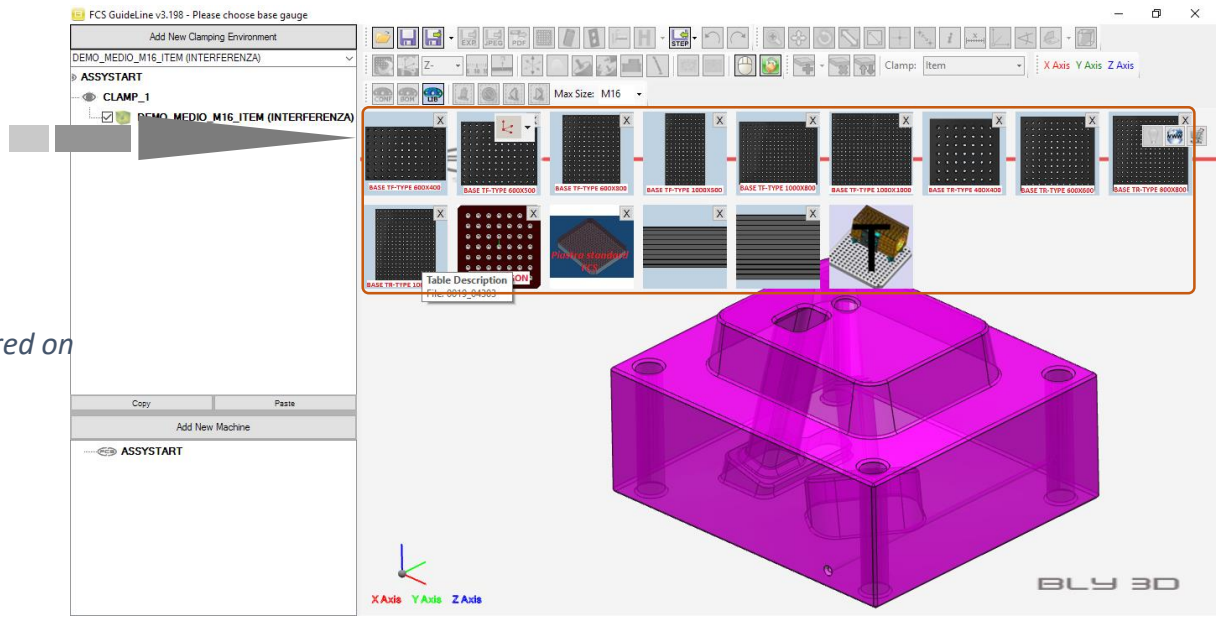


✓ *A 3D model can also be opened by drag&drop it on the desktop Guideline icon or in the graphic area*

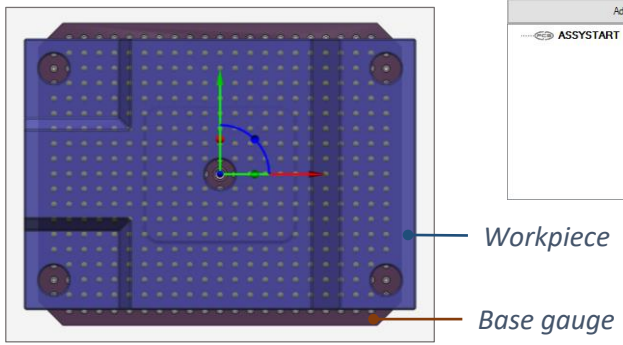
PHASE 1: PLACEMENT SEATS DESIGN

❑ *Base gauge selection*

Base gauges included in GL library are displayed at the top of the graphic area, ready to be selected by clicking them

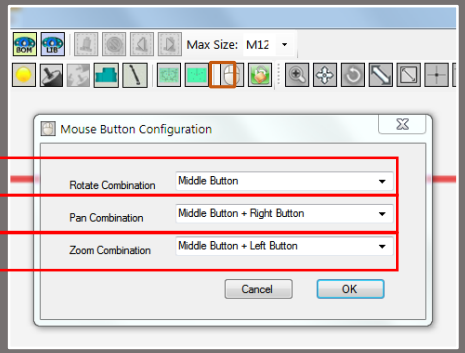


✓ *Once selected, the base gauge is centered on the workpiece (Bounding box)*



❑ *Visualization control: Rotate, Pan, Zoom, Fit, Zoom window*

✓ *Mouse buttons:*



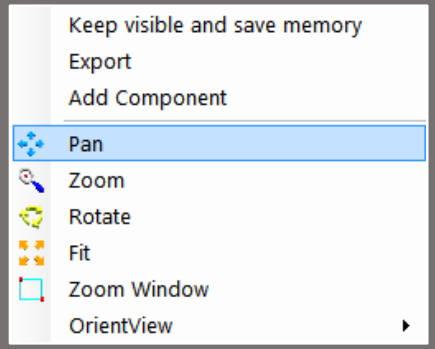
Rotate

Pan

Zoom

Mouse wheel scrolling

✓ *Right mouse button:*



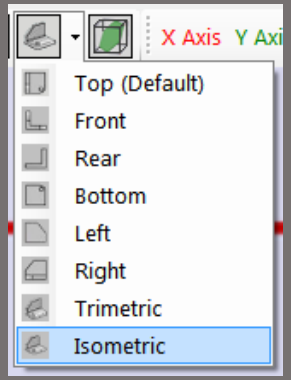
✓ *Tool bar:*

✓ *Press and hold central mouse button on the workpiece = that point becames the visualization center*

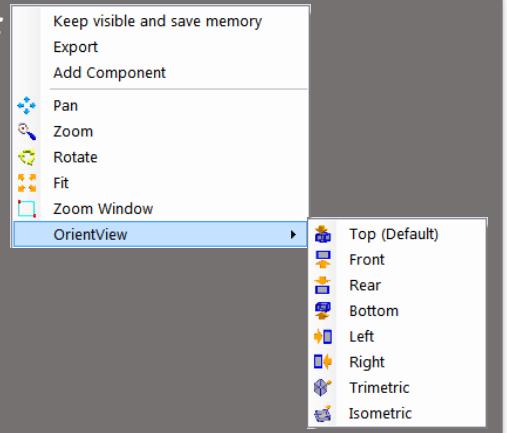
❑ *Visualization control: the views*

✓ *Keyboard buttons:*

✓ *Tool bar buttons:*



✓ *Right mouse button:*



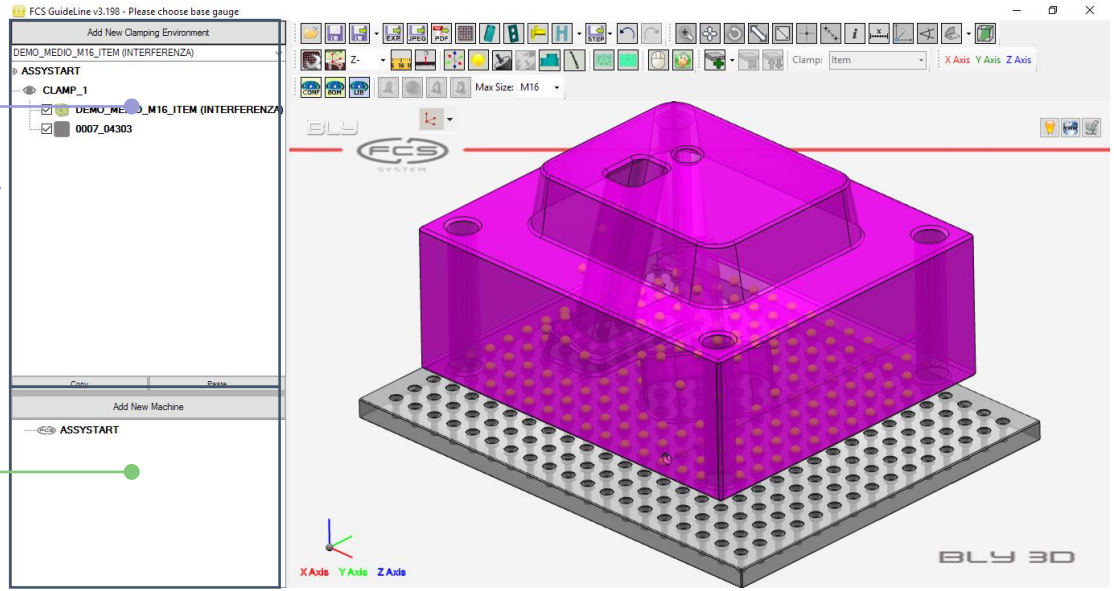
The two working environments:

- *Clamping environment (CAD)*

  Target:
  make the placement seats on the workpiece. A first clamping, named by default «CLAMP_1», is created as soon as the base gauge is selected and made working environment.

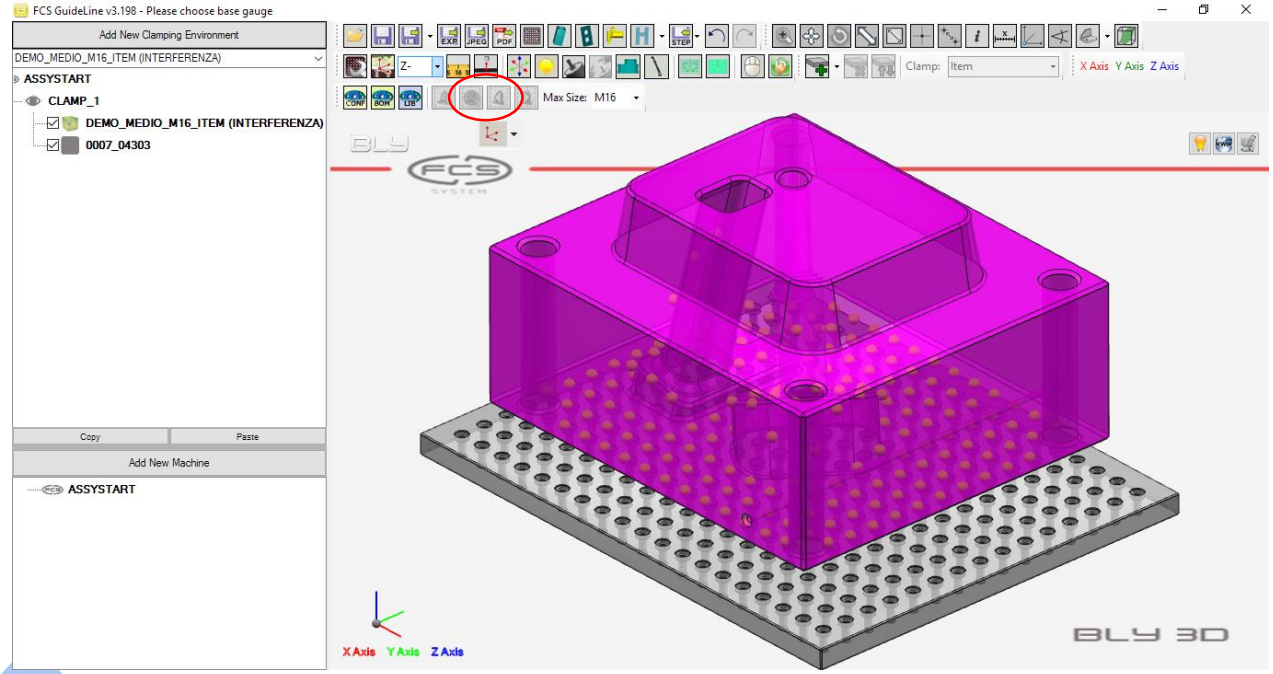
- *Machining environment (CAM)*

  Target:
  make the complete clamping, starting from the clamping holes already defined in the previous environment (or in any case already included in the workpiece geometry).



❑ *Workpiece orientation*

Guideline places by default the workpiece on the gauge using the workpiece coordinate system and aligning the Z axis toward the vertical direction. User can change the orientation using main directions or manually place the workpiece



❑ *Workpiece orientation*

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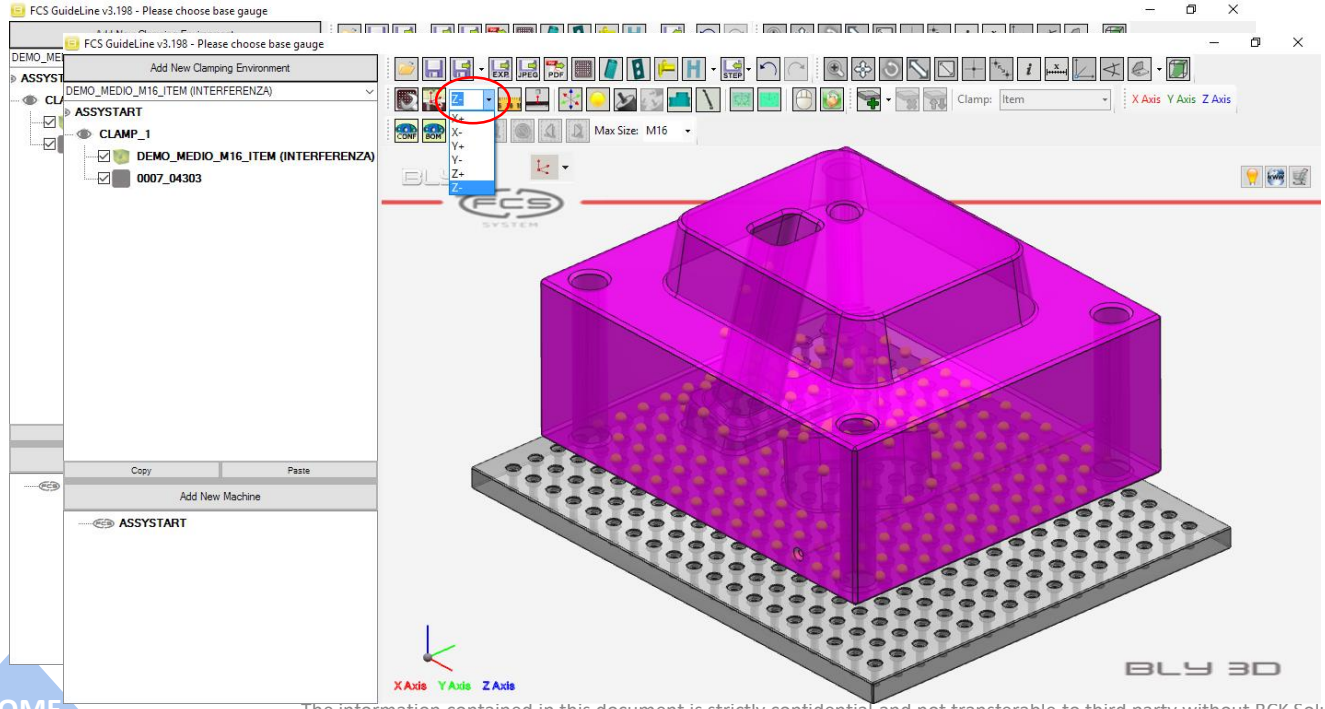
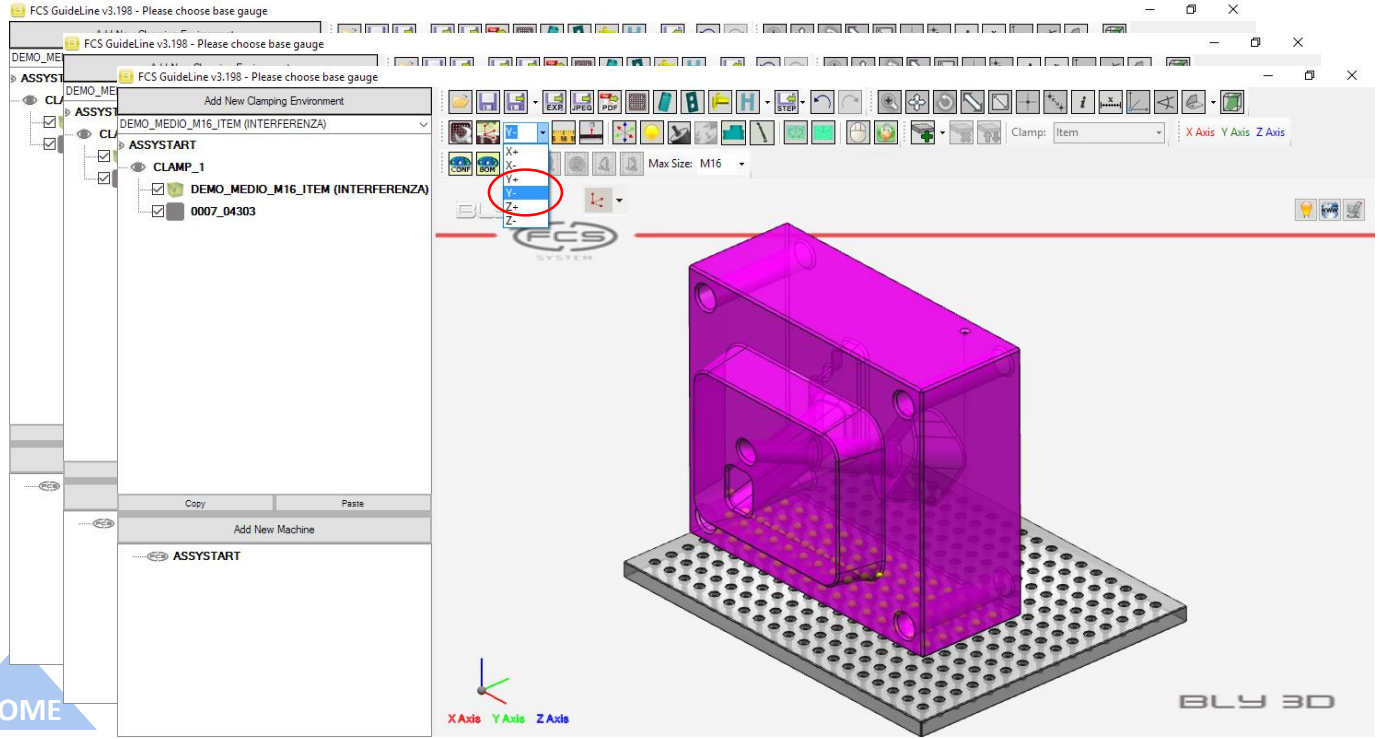


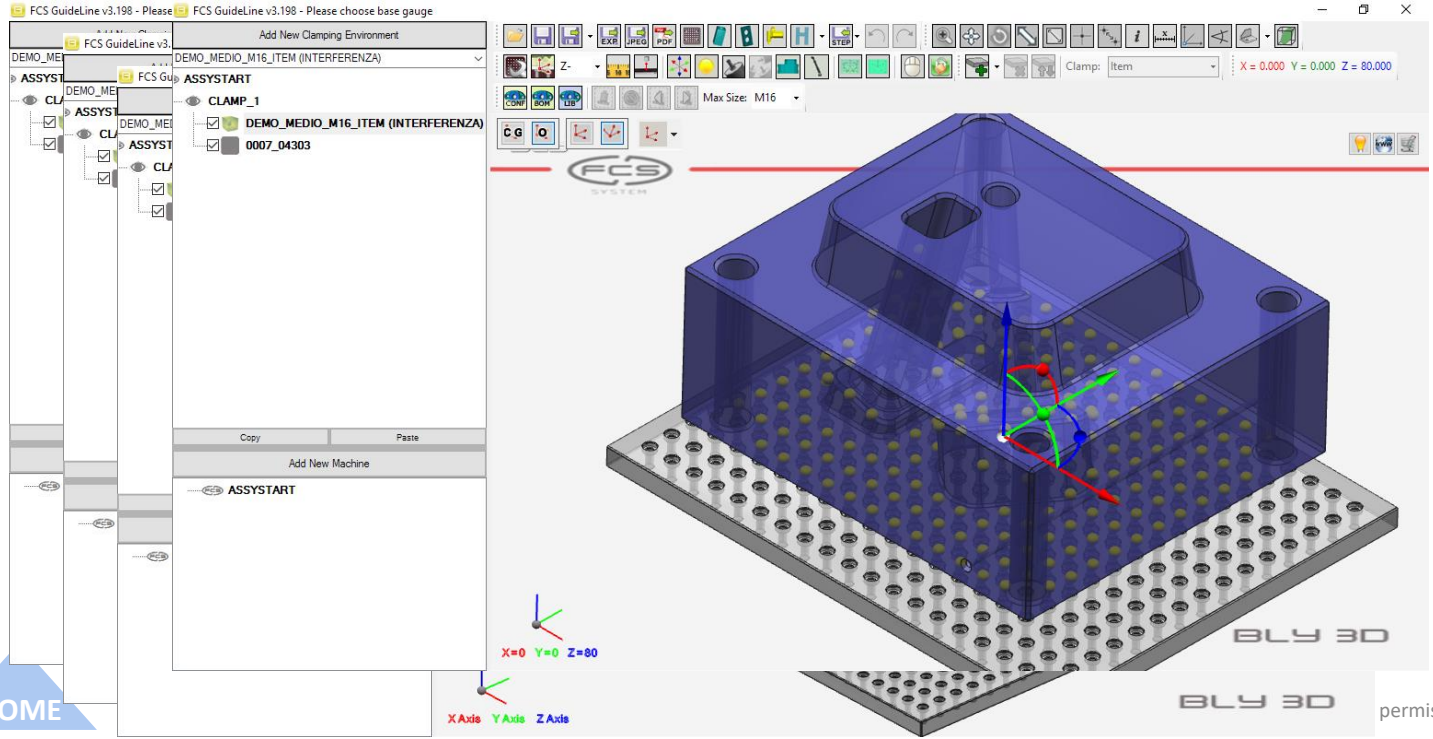
❑ *Workpiece orientation*

Guideline places by default the workpiece on the gauge using the workpiece coordinate system and aligning the Z axis toward the vertical direction. User can change the orientation using main directions or manually place the workpiece



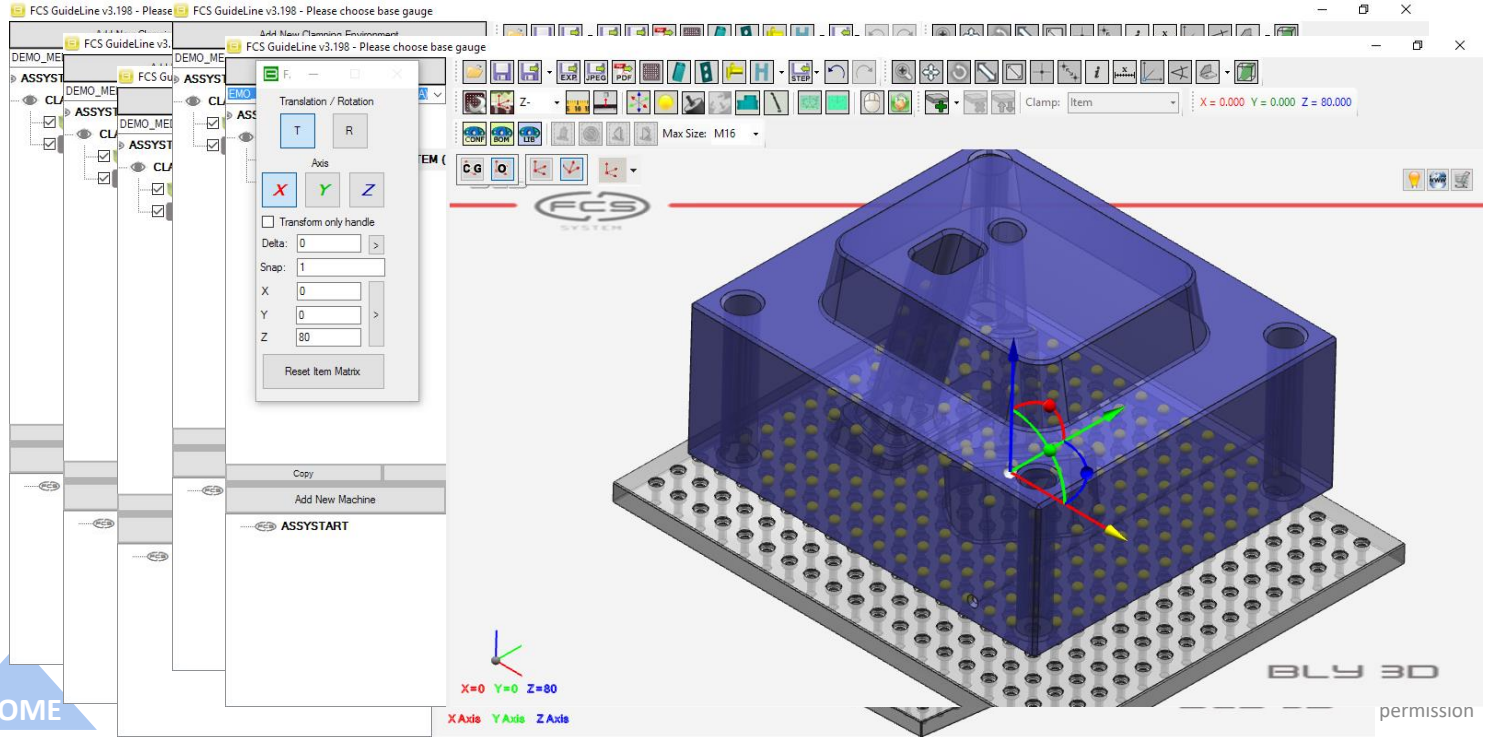
❑ *Workpiece orientation (cont'd)*

It is possible to orient the workpiece freely using a manipulator: just click once on the workpiece and it will be displayed



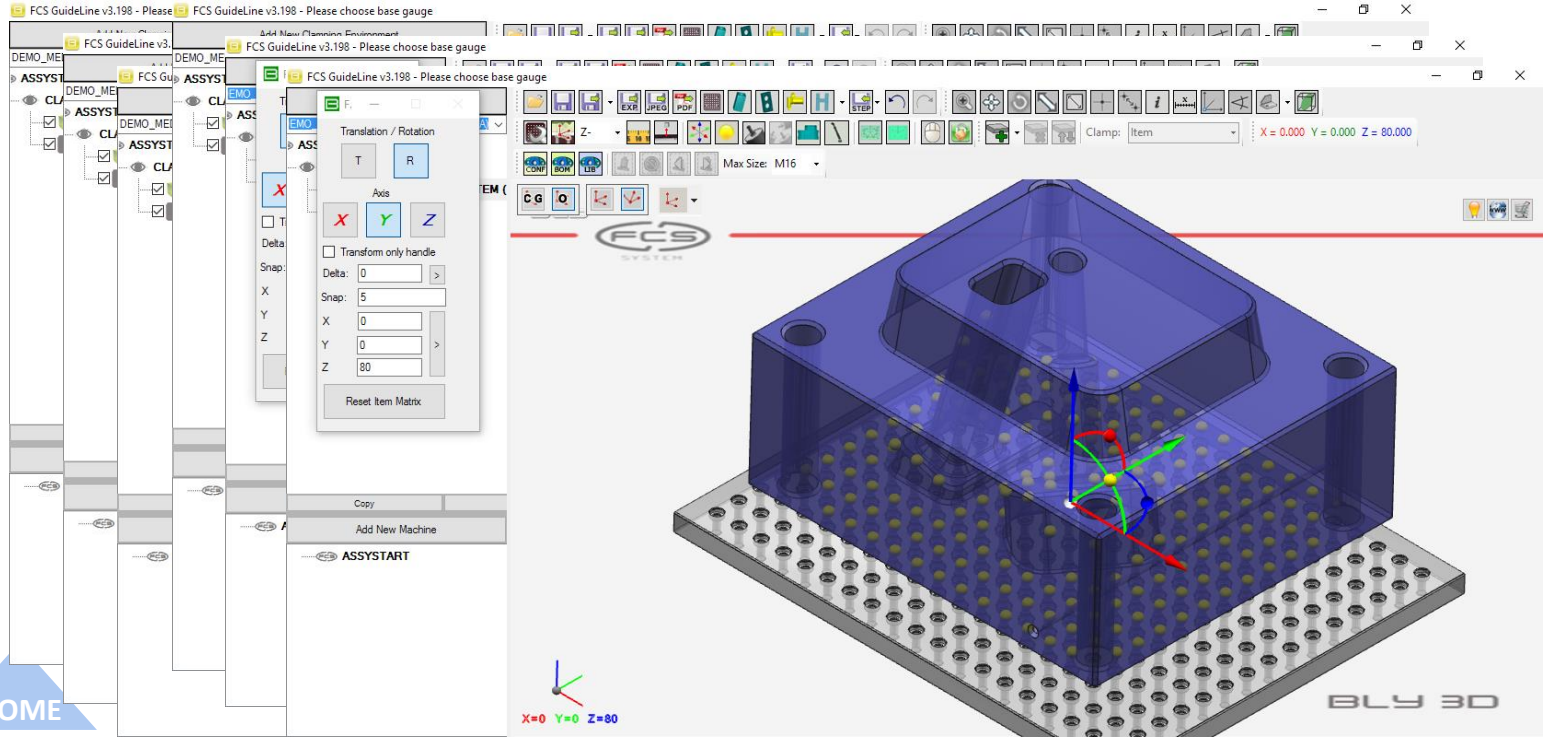
❑ *Workpiece orientation (cont'd)*

You can now translate the piece dragging it along the arrows on the manipulator (i.e. yellow in the picture) or using the window that appears onto the screen



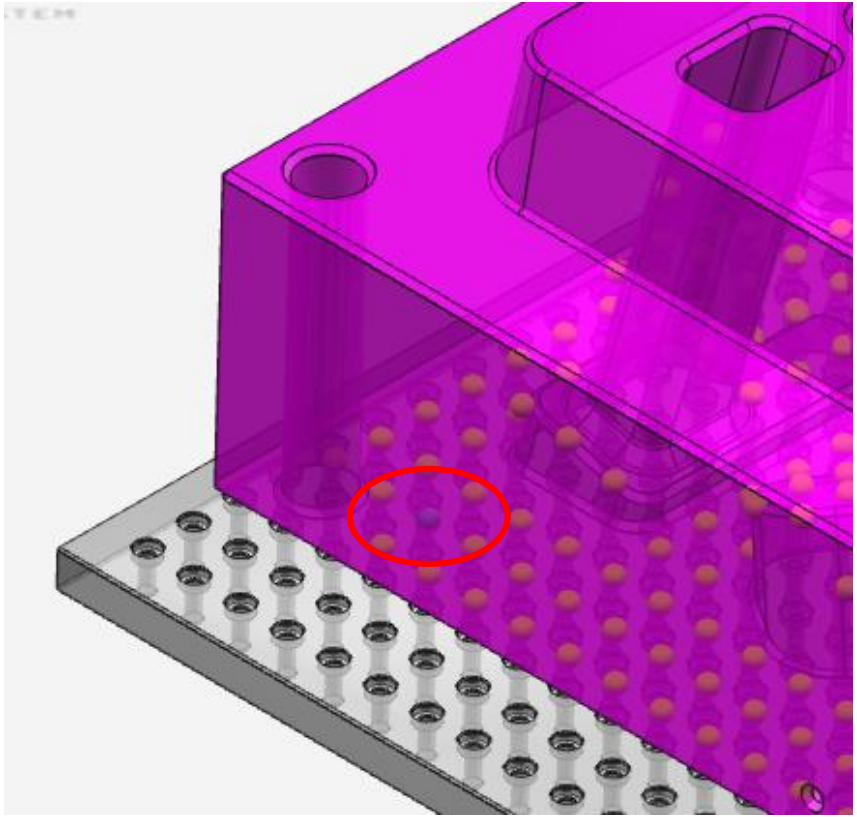
❑ *Workpiece orientation (cont'd)*

Similarly, the piece can be rotated using the spheres on the manipulator (i.e. yellow in the picture) or again typing the values on the translation/rotation window



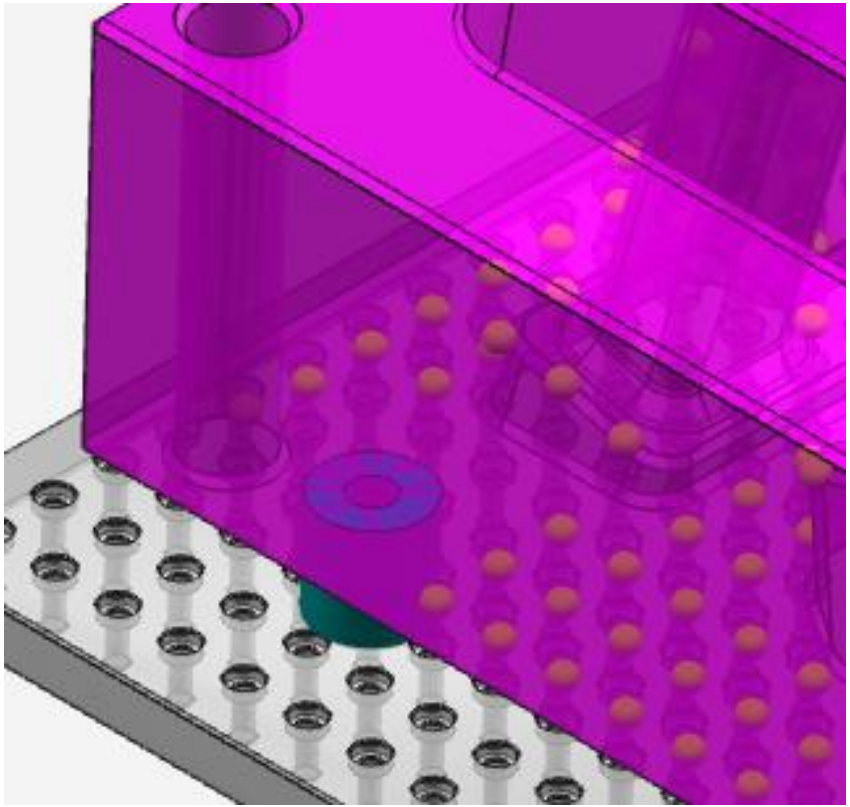
❑ *Stacks placement*

Stacks can be placed by clicking the spheres on the workpiece: those are the base gauge's placement holes projected onto the workpiece.



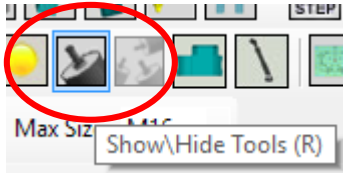
❑ *Stacks placement*

Stacks can be placed by clicking the spheres on the workpiece: those are the base gauge's placement holes projected onto the workpiece.
Guideline creates the stack and places the 3D models of the FCS system in the selected points.
The software allows the user to place as many stacks as desired, with no limitations.
To delete one stack, simply click on it.



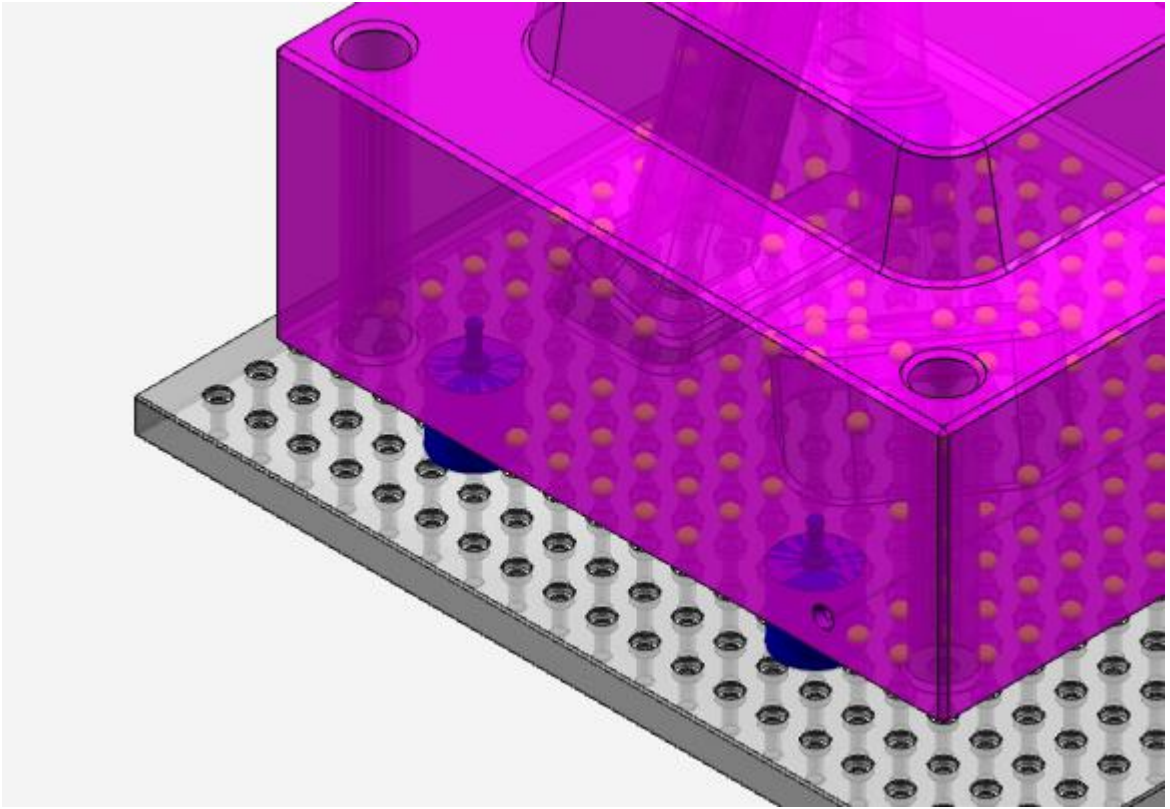
❑ *Stacks placement (cont'd)*

In addition, Guideline creates and places the geometry defining the seats that are needed to be created in the workpiece's 3D model.
It can be displayed pressing the button



on the toolbar

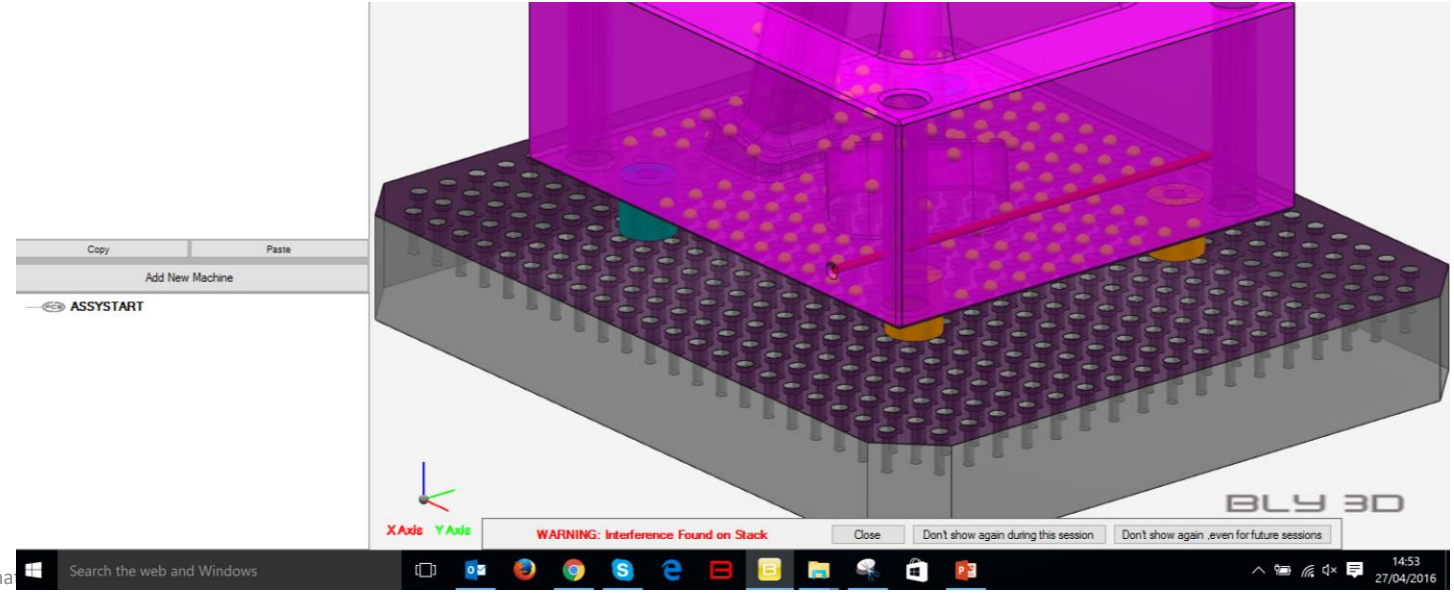
- ❑ This geometry can be exported into the CAD and used as reference to modify the model (see chap. *Export the clamping definition to CAD*)



❑ *Stacks placement (cont'd): interference analysis*

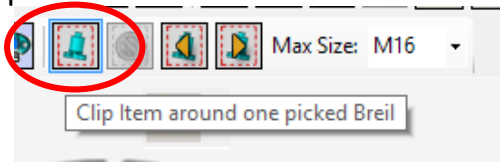
During the stacks' placement, Guideline analyzes the interfence that the component or the tool geometry may have with part of the workpiece that are supposed not to be modified, particularly holes, cooling, figure etc.
It is possible to set the minimum distance to be kept (see chap.XXX).
In case of violations, the software highlights the stack and the geometry involved, together with the stack's placement face on the piece. A warning is displayed also at the bottom of the graphic area

IMPORTANT: Guideline does not suggest any fixing to the problem, since this requires critical analysis by the designer. As an example, one possible solution is to delete the stack and place it somewhere else, while another possibility is modify the piece.

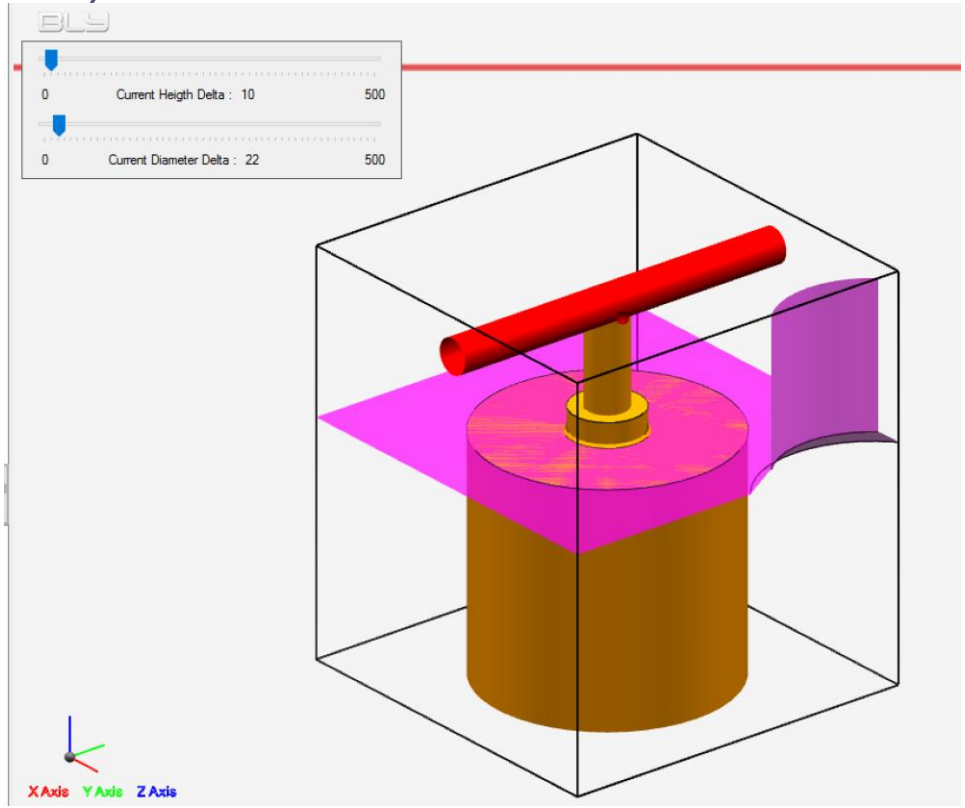


❑ *Stacks placement (cont'd): interference analysis*

To evaluate the interfence and the area around, press

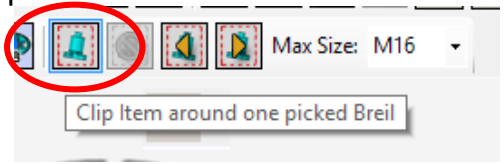


on the toolbar and then click the stack that you want to evaluate: Guideline will graphically isolate the area.
It is possibile to enlarge the volume evaluated using the sliders and execute measures of distance.



❑ *Stacks placement (cont'd): interference analysis*

To evaluate the interfence and the area around, press

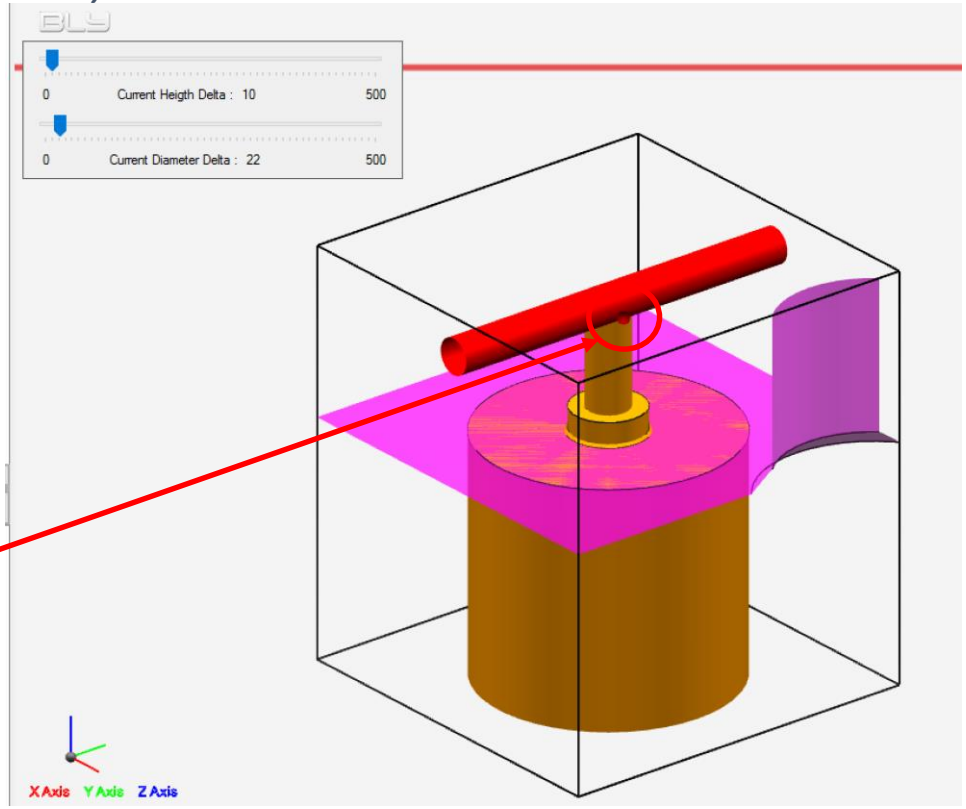


on the toolbar and then click the stack that you want to evaluate: Guideline will graphically isolate the area.

It is possibile to enlarge the volume evaluated using the sliders and execute measures of distance.

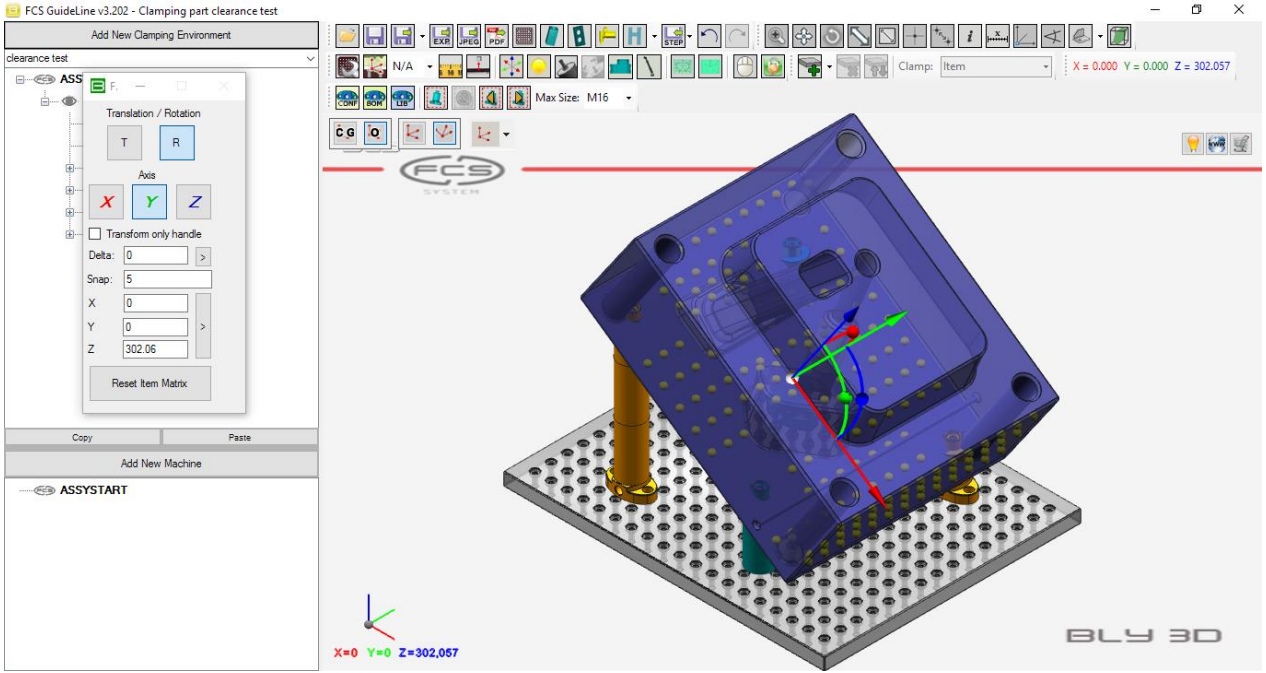
As a reference, a small red dot identifies the closest point violating the clearance.

To exit form the evaluation, press «esc» on the keyboard



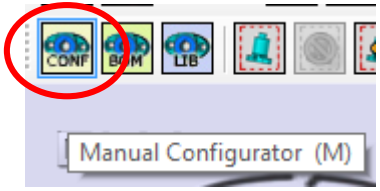
❑ *Stacks placement (cont'd): change the workpiece orientation*

As discussed before, the workpiece can be freely oriented in the space: this operation can be done before or after the stacks' placement.
In this second case, Guideline updates the stacks' configuration according to the workpiece orientation and respecting the table settings (see chap. XXX), with particular regards to the minimum distance value.
As shown in the picture, interference check is always active.

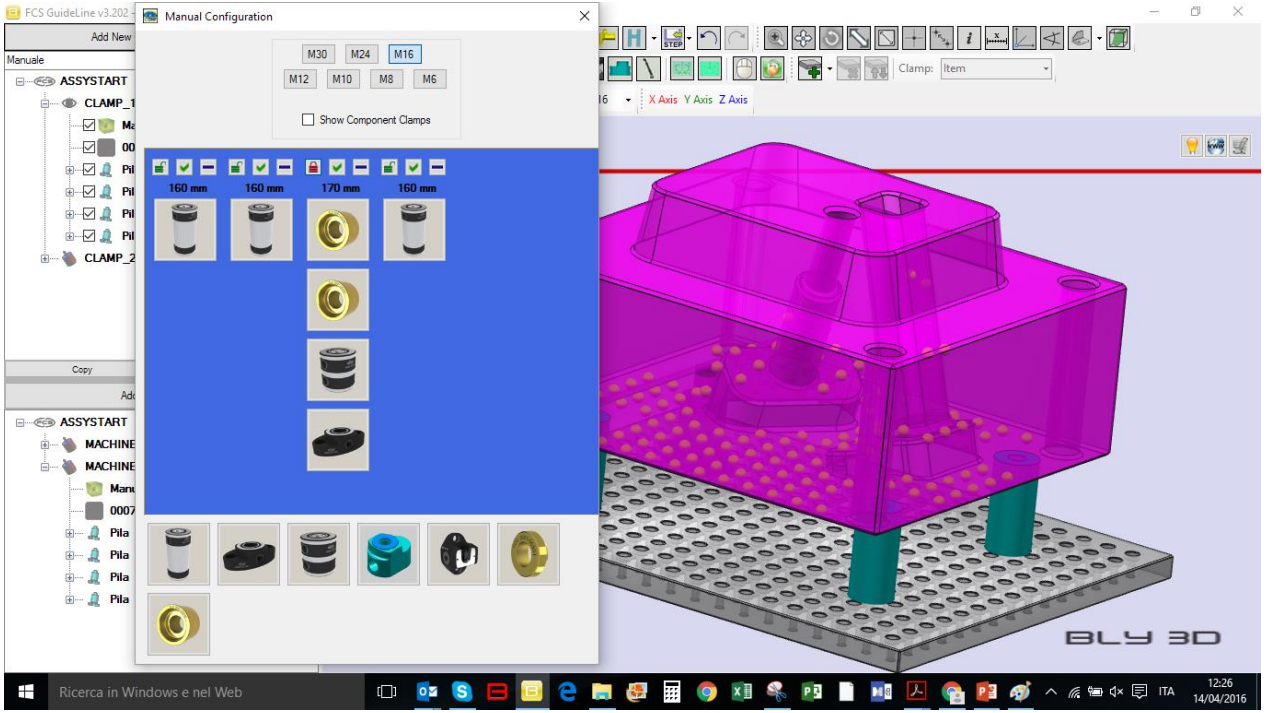


❑ *Stacks placement (cont'd): manual configuration*

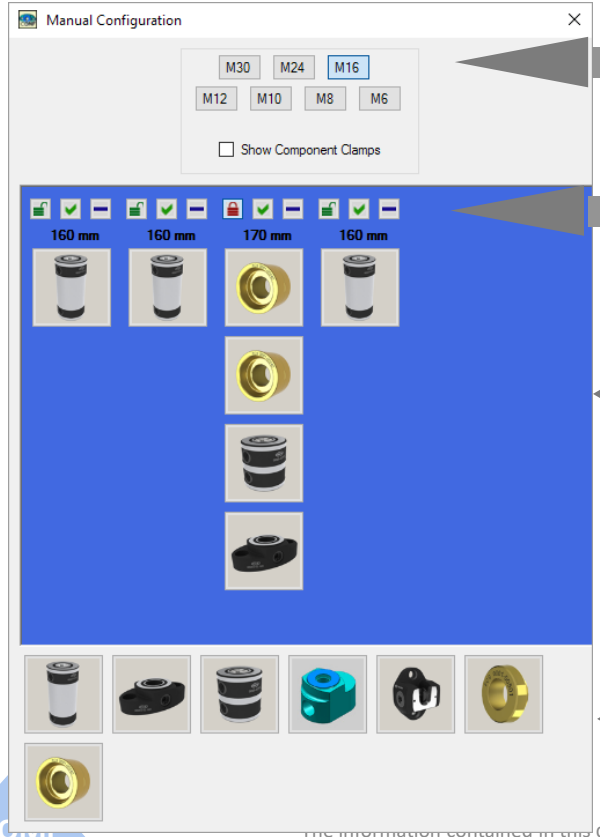
Should the combination(s) be not acceptable, you have the possibility to manually change it. To enter the Manual Configurator menu, press



on the toolbar.



❑ *Stacks placement (cont'd): manual configuration*



Dimension selection

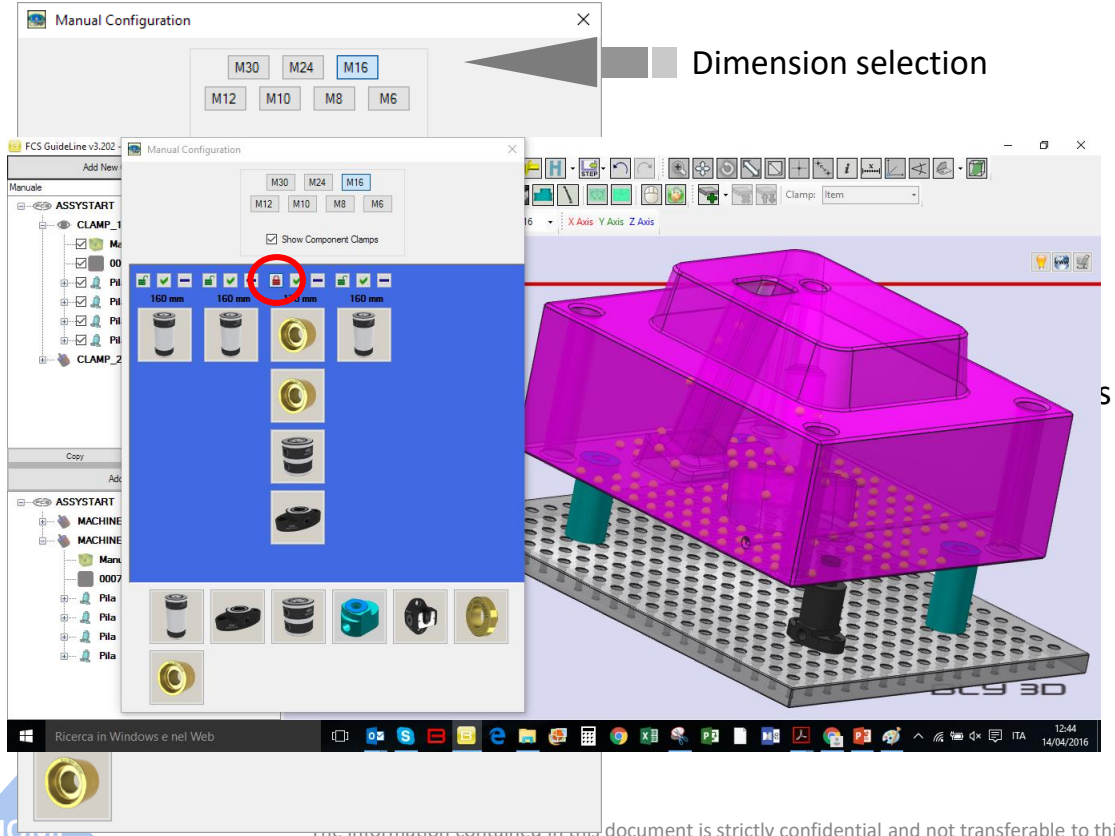
Checking, orienting and locking options

Actual stacks' configurations

Available components

In this window, you can manually define the configuration of each stack by dragging-and-dropping the components from the ones available into the library.
In order to prevent impossible combinations, the softwarecalculates the resulting height and checks the compatibility of the selected components and of the mutual heights between the stacks.
You can navigate the library by clicking the buttons in the Dimension selection area: Guideline will display the elements available in your library only.

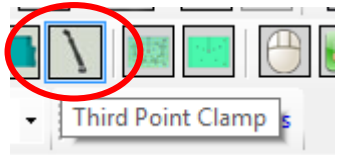
❑ *Stacks placement (cont'd): manual configuration*



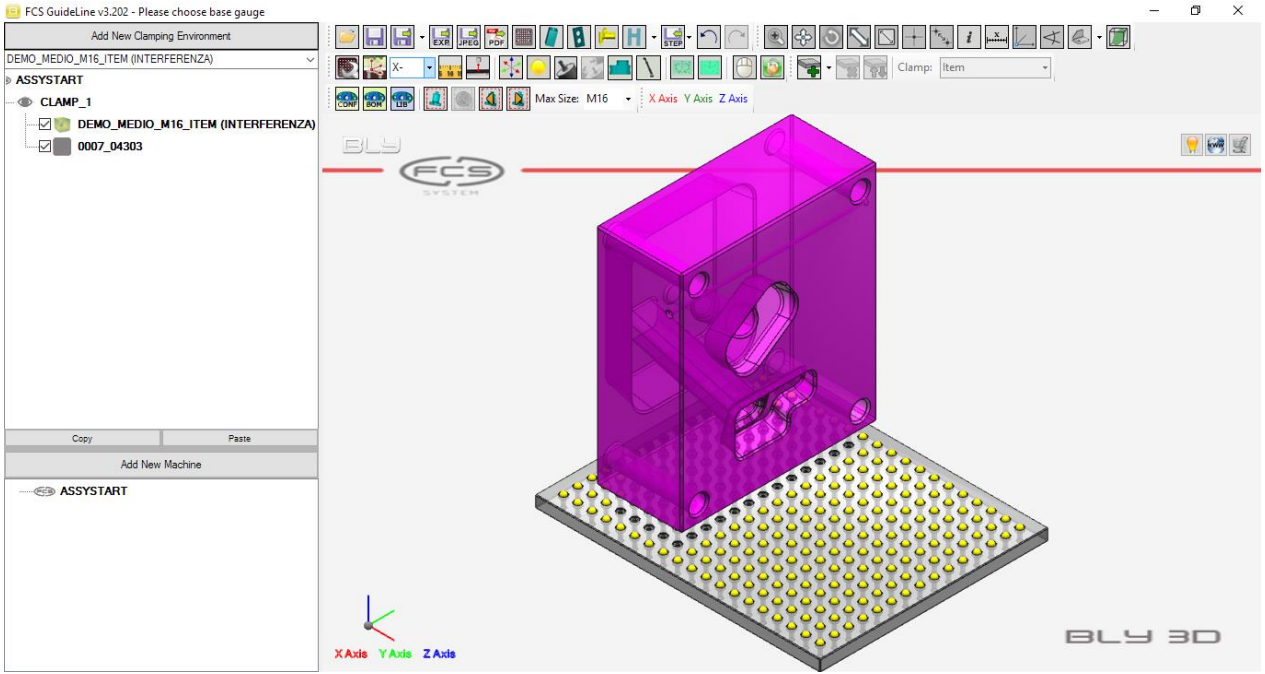
In this window, you can manually define the configuration of each stack by dragging-and-dropping the components from the ones available into the library.
In order to prevent impossible combinations, the softwarecalculates the resulting height and checks the compatibility of the selected components and of the mutual heights between the stacks.
You can navigate the library by clicking the buttons in the Dimension selection area: Guideline will display the elements available in your library only.
Once a combination is defined, you can check it, lock it (to prevent Guideline to recalculate it in case of workpiece repositioning) and orient it (useful for Extra Step Clamp).

❑ *Third Point Clamp*

To open the third point placement menu, press

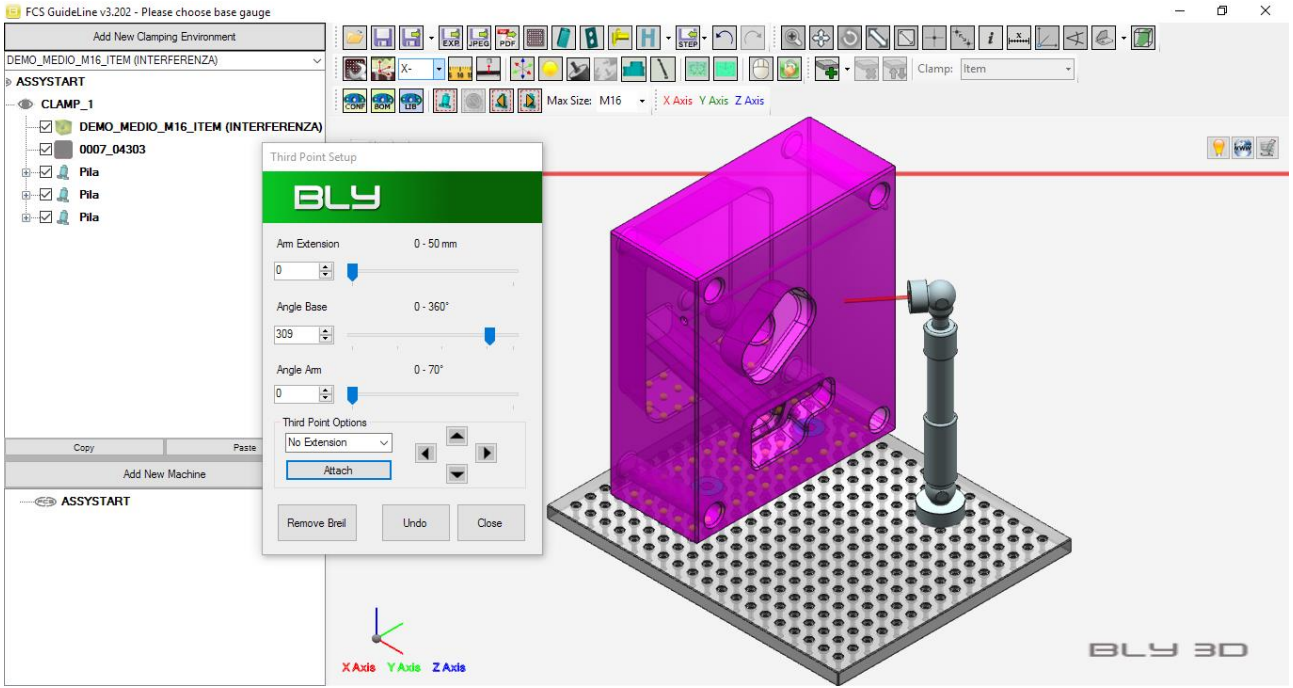


The software will display, on the base gauge, a new series of placement spheres, OUTSIDE the work piece projection.



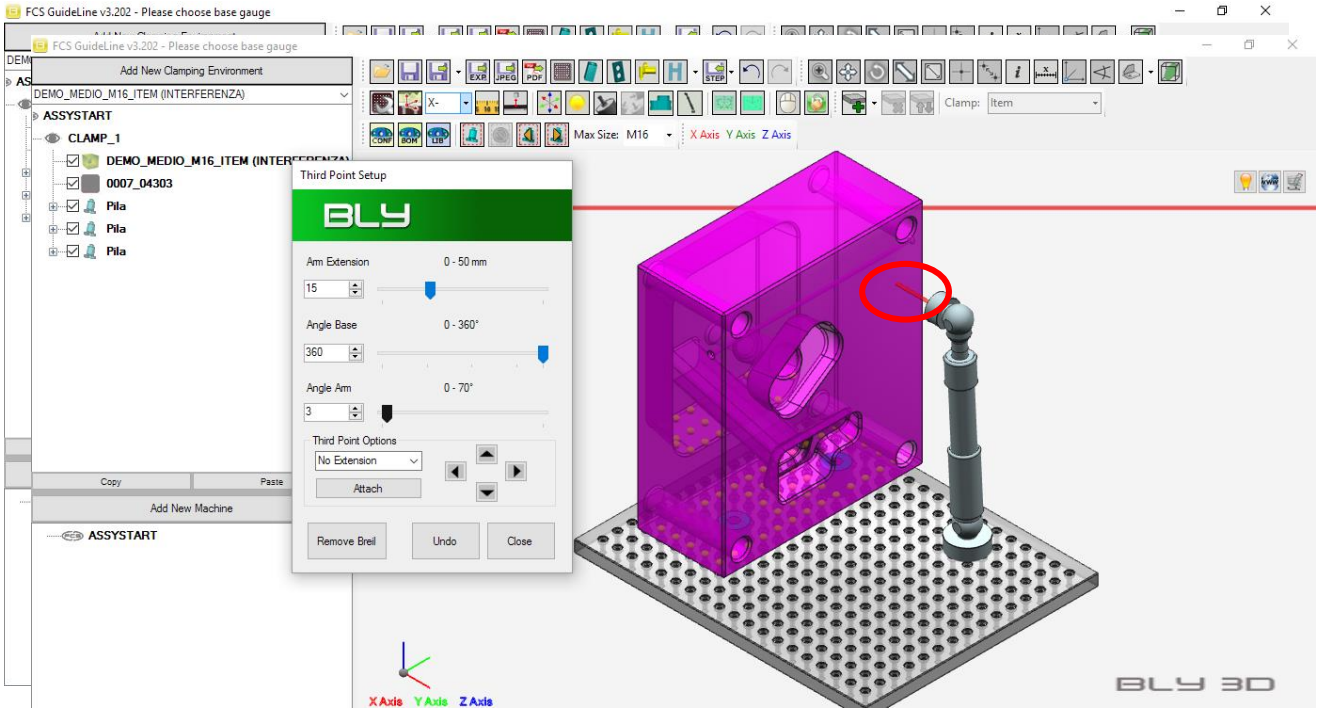
❑ *Third Point Clamp (cont'd)*

Clicking on one sphere, the software places the arm on the base gauge, vertical, and opens the Third Point Setup window.



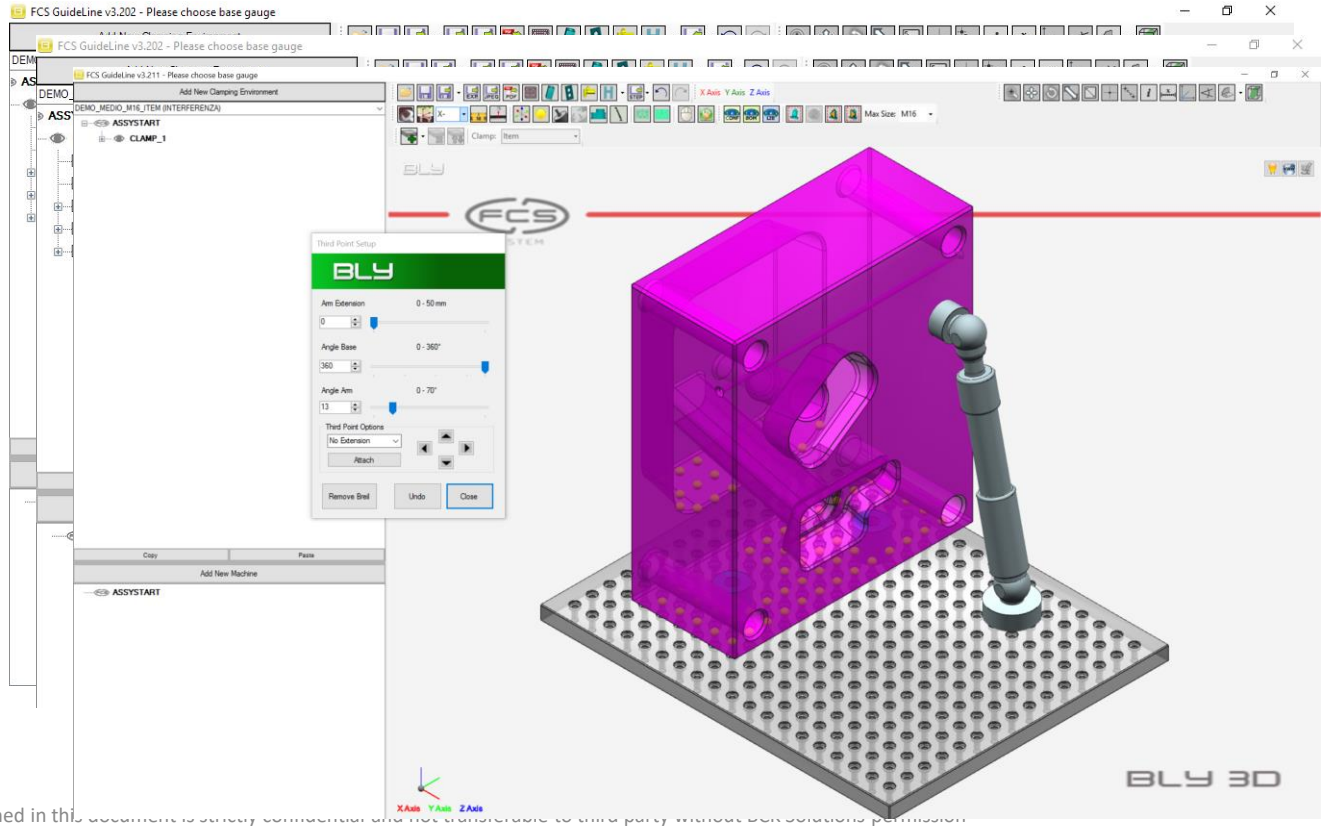
❑ *Third Point Clamp (cont'd)*

Clicking on one sphere, the software places the arm on the base gauge, vertical, and opens the Third Point Setup window.
Using the sliders, you can orient the arm until the red head axis (highligthed in the picture) touches the face that you want to attach the arm onto.

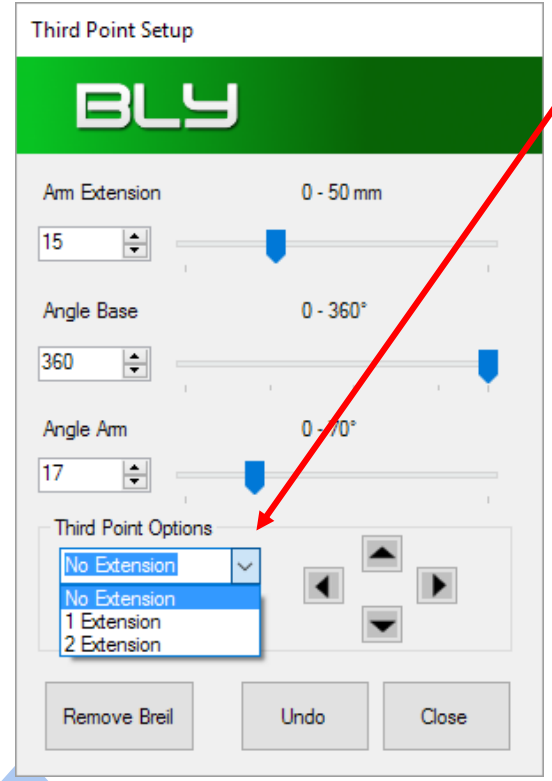


❑ *Third Point Clamp (cont'd)*

Clicking on one sphere, the software places the arm on the base gauge, vertical, and opens the Third Point Setup window.
Using the sliders, you can orient the arm until the red head axis (highligthed in the picture) touches the face that you want to attach the arm onto.
By pressing «Attach» button the clamping will be finalized.



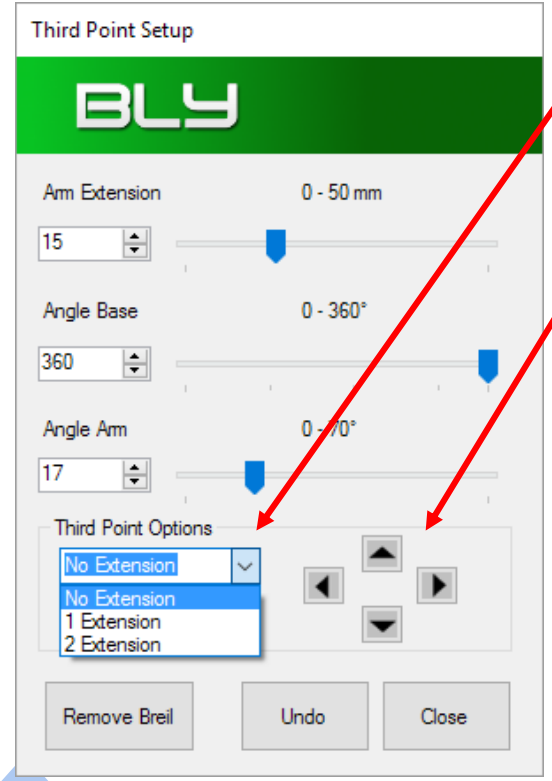
❑ *Third Point Clamp (cont'd)*



Additional options:

- DROP-DOWN MENU
  - No Extension: the arm placed is the standard one
  - 1 Extension: places the arm with one extension, which increases the lenght of…
  - 2 Extension: places the arm with two extensions

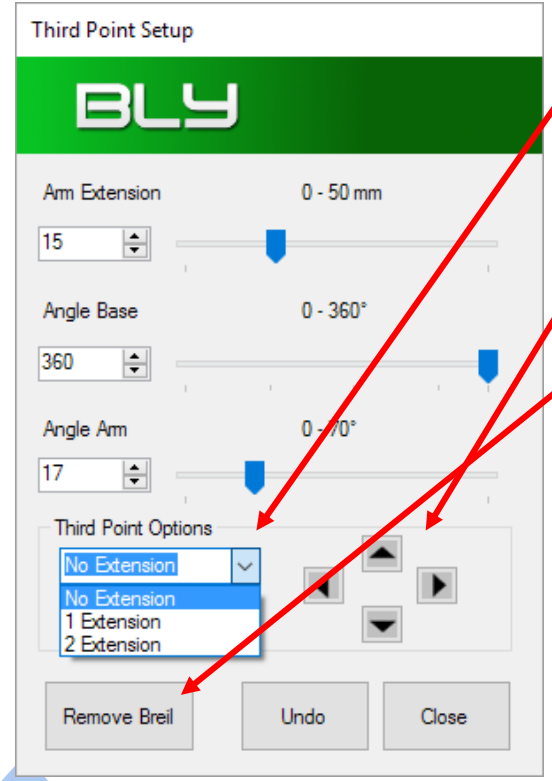
❑ *Third Point Clamp (cont'd)*



Additional options:

- DROP-DOWN MENU
  - No Extension: the arm placed is the standard one
  - 1 Extension: places the arm with one extension, which increases the lenght of…
  - 2 Extension: places the arm with two extensions
- ARROWS: move the arm on the base gauge from one placement hole to another, keeping the head attached to the workpiece;

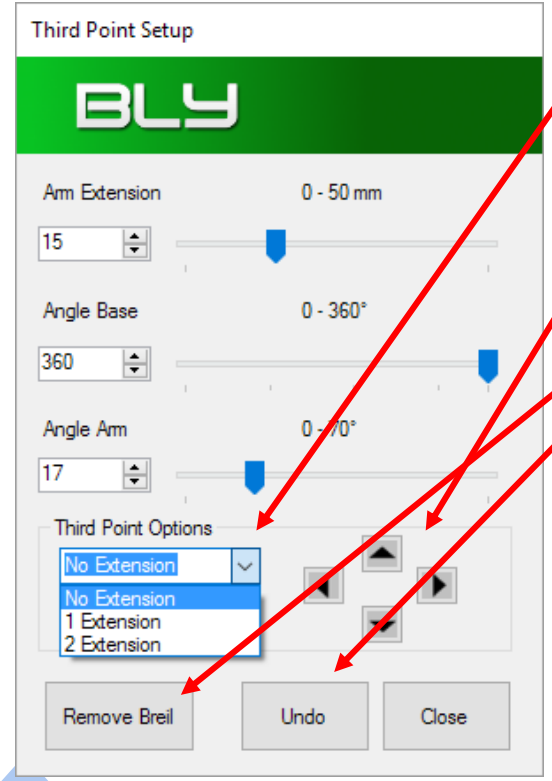
❑ *Third Point Clamp (cont'd)*



Additional options:

- DROP-DOWN MENU
  - No Extension: the arm placed is the standard one
  - 1 Extension: places the arm with one extension, which increases the lenght of…
  - 2 Extension: places the arm with two extensions
- ARROWS: move the arm on the base gauge from one placement hole to another, keeping the head attached to the workpiece;
- REMOVE BREIL: deletes the arm;

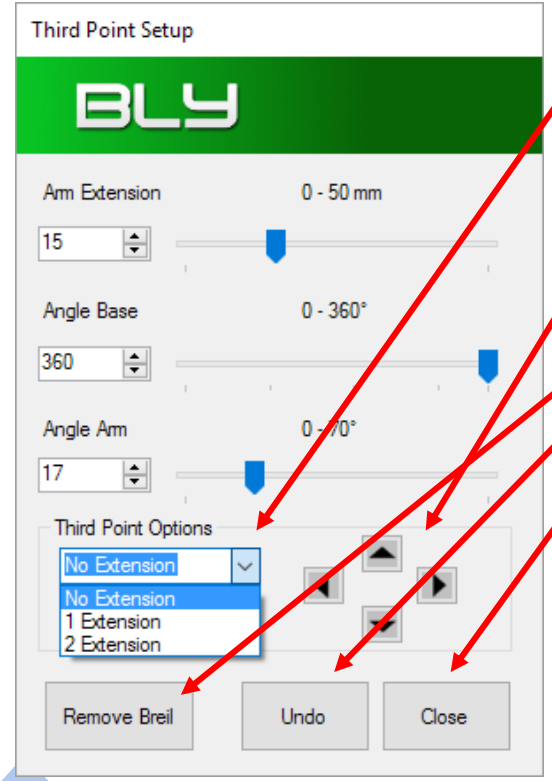
❑ *Third Point Clamp (cont'd)*



Additional options:

- DROP-DOWN MENU
  - No Extension: the arm placed is the standard one
  - 1 Extension: places the arm with one extension, which increases the lenght of…
  - 2 Extension: places the arm with two extensions
- ARROWS: move the arm on the base gauge from one placement hole to another, keeping the head attached to the workpiece;
- REMOVE BREIL: deletes the arm;
- UNDO: undo the last operation

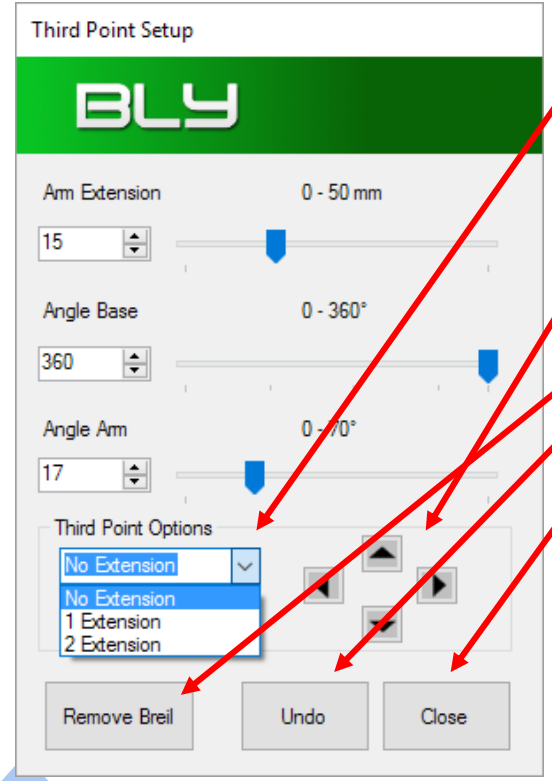
❑ *Third Point Clamp (cont'd)*



Additional options:

- DROP-DOWN MENU
  - No Extension: the arm placed is the standard one
  - 1 Extension: places the arm with one extension, which increases the lenght of…
  - 2 Extension: places the arm with two extensions
- ARROWS: move the arm on the base gauge from one placement hole to another, keeping the head attached to the workpiece;
- REMOVE BREIL: deletes the arm;
- UNDO: undo the last operation
- CLOSE: close the window

❑ *Third Point Clamp (cont'd)*



Additional options:

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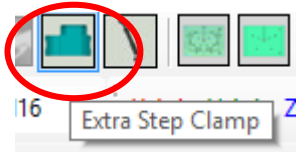
  - No Extension: the arm placed is the standard one
  - 1 Extension: places the arm with one extension, which increases the lenght of…
  - 2 Extension: places the arm with two extensions
- ARROWS: move the arm on the base gauge from one placement hole to another, keeping the head attached to the workpiece;
- REMOVE BREIL: deletes the arm;
- UNDO: undo the last operation
- CLOSE: close the window

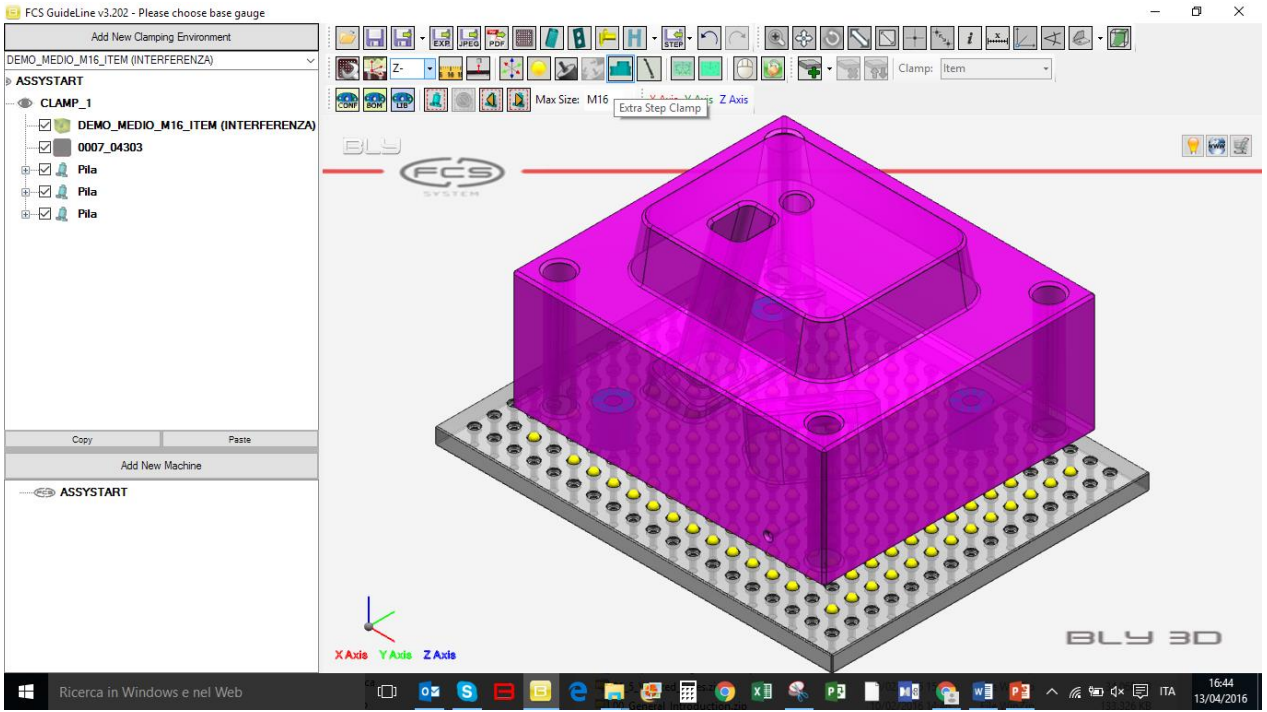
To modify an arm already placed, click it in the graphic area.

❑ *Extra Step Clamp*

To open the extra step placement, press

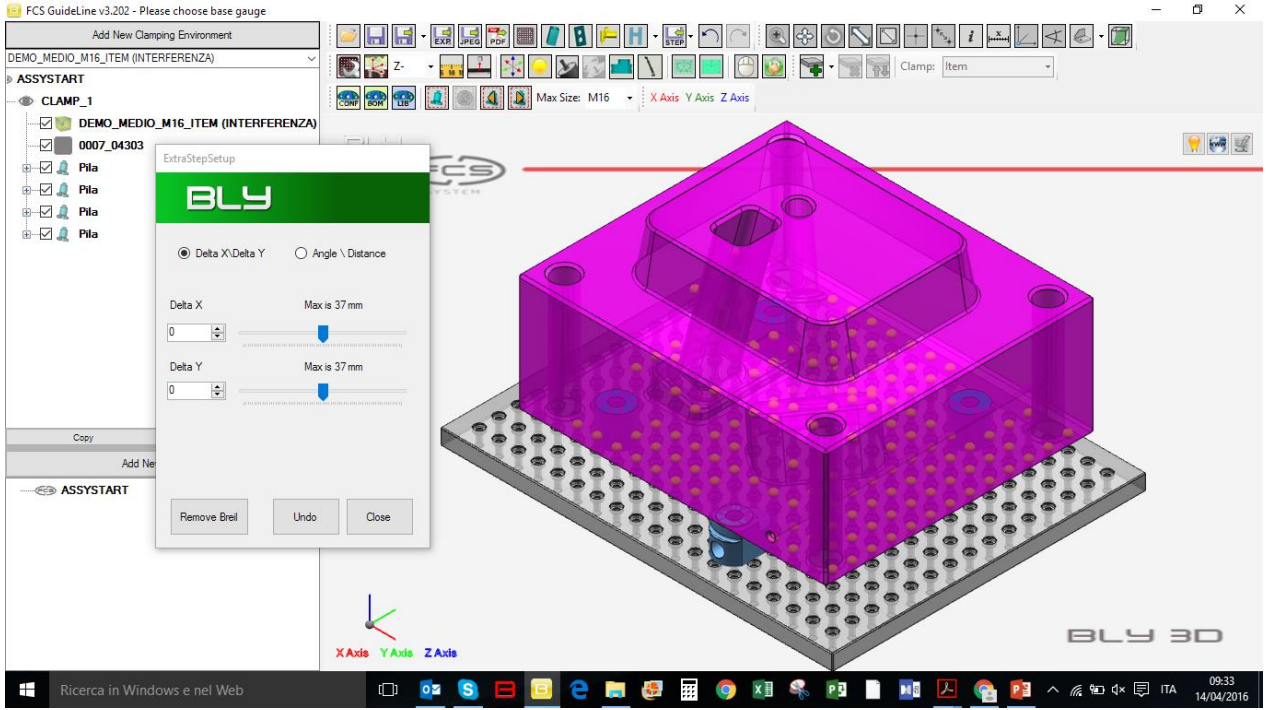


The software will display, on the base gauge, a new series of placement spheres, INSIDE the projection of the work piece.



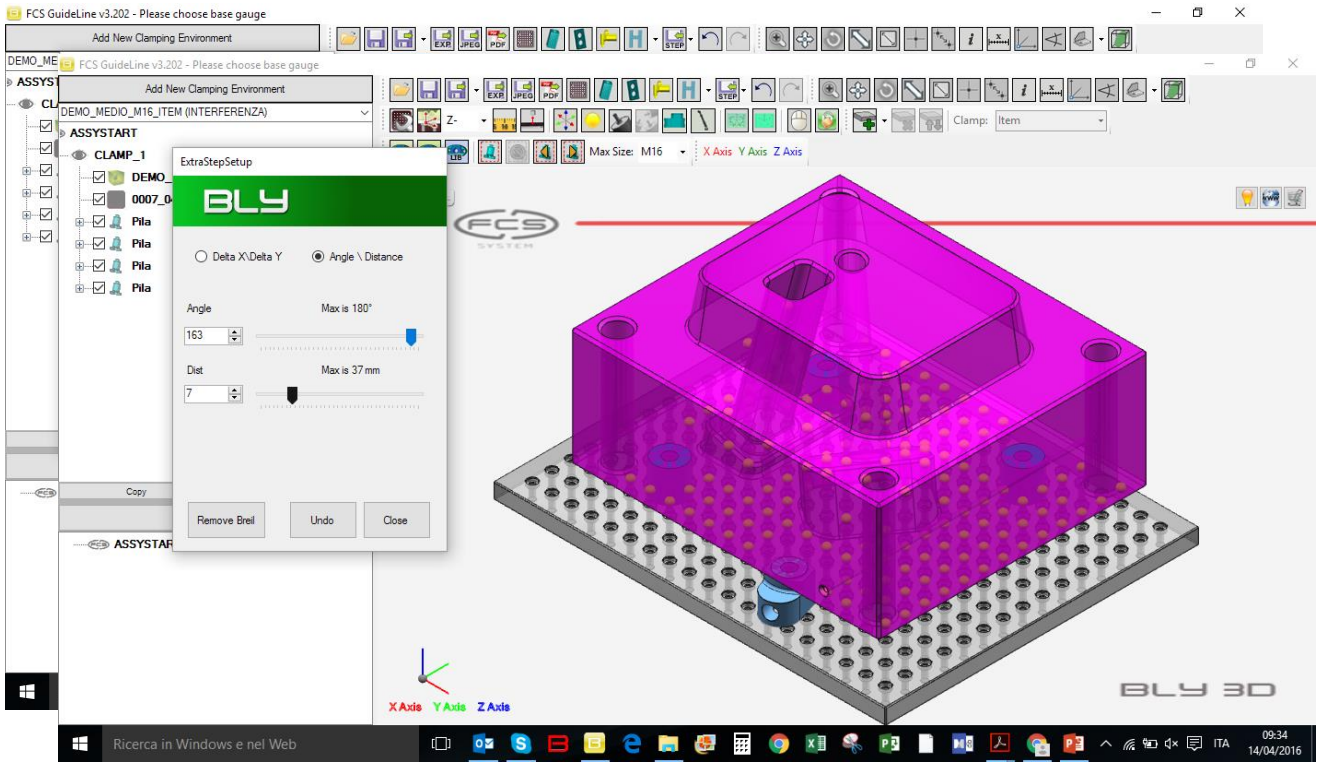
❑ *Extra Step Clamp (cont'd)*

Clicking one sphere, the software places the breyl on the base gauge, and opens the ExtraStepSetup window.

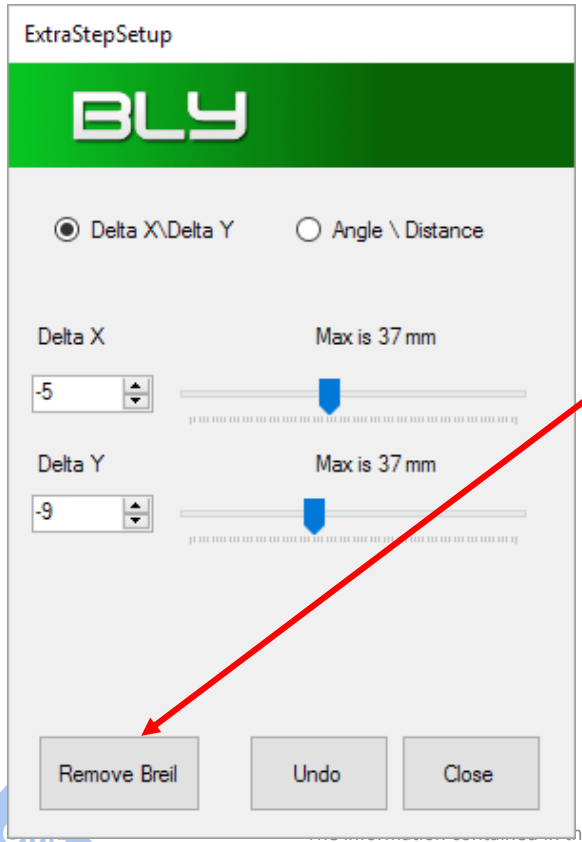


❑ *Extra Step Clamp (cont'd)*

Clicking one sphere, the software places the breyl on the base gauge, and opens the ExtraStepSetup window.
You can orient the breyl using the orthogonal placement «DeltaX\DeltaY» or the «Angle\Distance» options.
In both cases, either numbers can be typed or you can adjust the position using the sliders

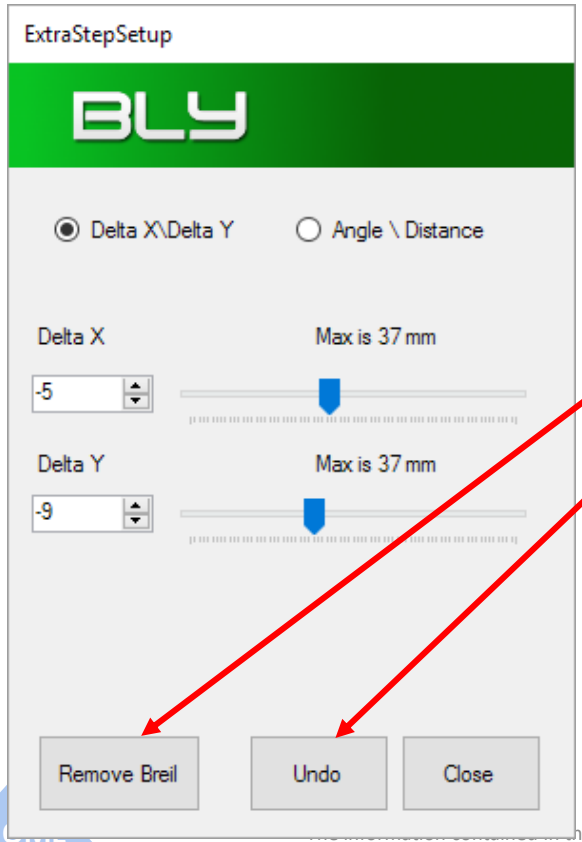


❑ *Extra Step Clamp (cont'd)*



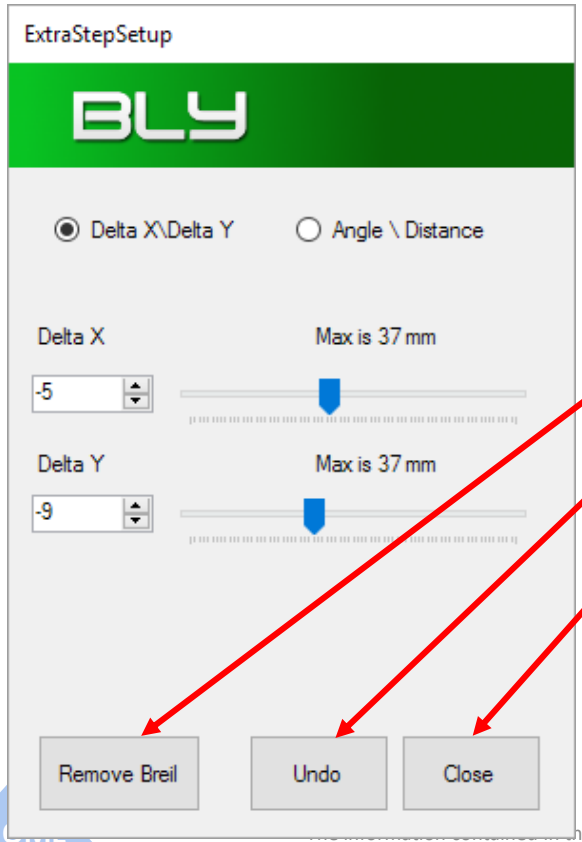
Additional options:
REMOVE BREIL: deletes the arm

❑ *Extra Step Clamp (cont'd)*



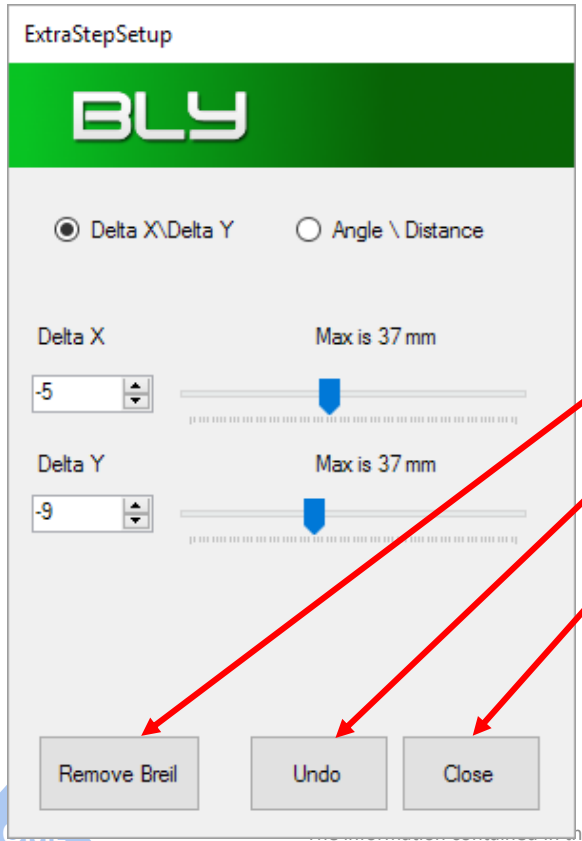
Additional options:
REMOVE BREIL: deletes the arm
UNDO: undo the last operation

❑ *Extra Step Clamp (cont'd)*



Additional options:
REMOVE BREIL: deletes the arm
UNDO: undo the last operation
CLOSE: close the window

❑ *Extra Step Clamp (cont'd)*



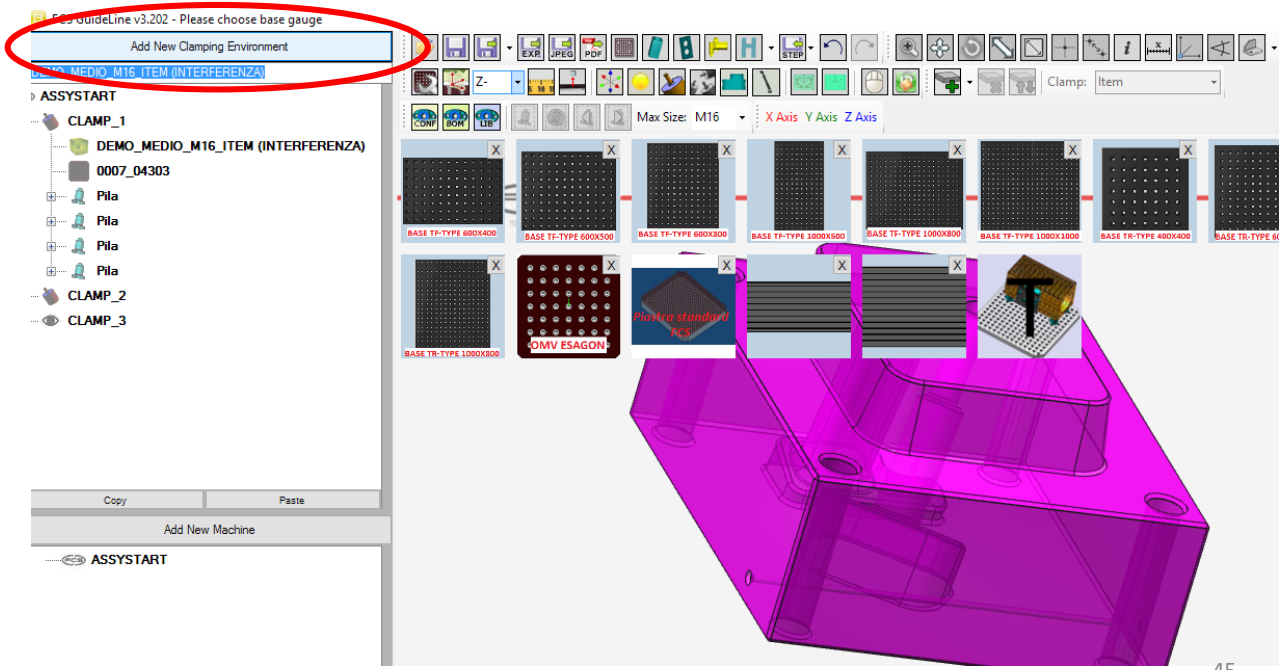
Additional options:
REMOVE BREIL: deletes the arm
UNDO: undo the last operation
CLOSE: close the window

To modify an extra step already placed, click it in the graphic area.

❑ *Create additional clamping environments*

It is very common the need of clamping the workpiece in more than one position in order to execute all the operations. In this case, FCS System requires to define more than one set of placement seats: Guideline reproduce this occurence giving you the possibility to define unlimited clamping environment, having its own setup each (base gauge, orientation, components).

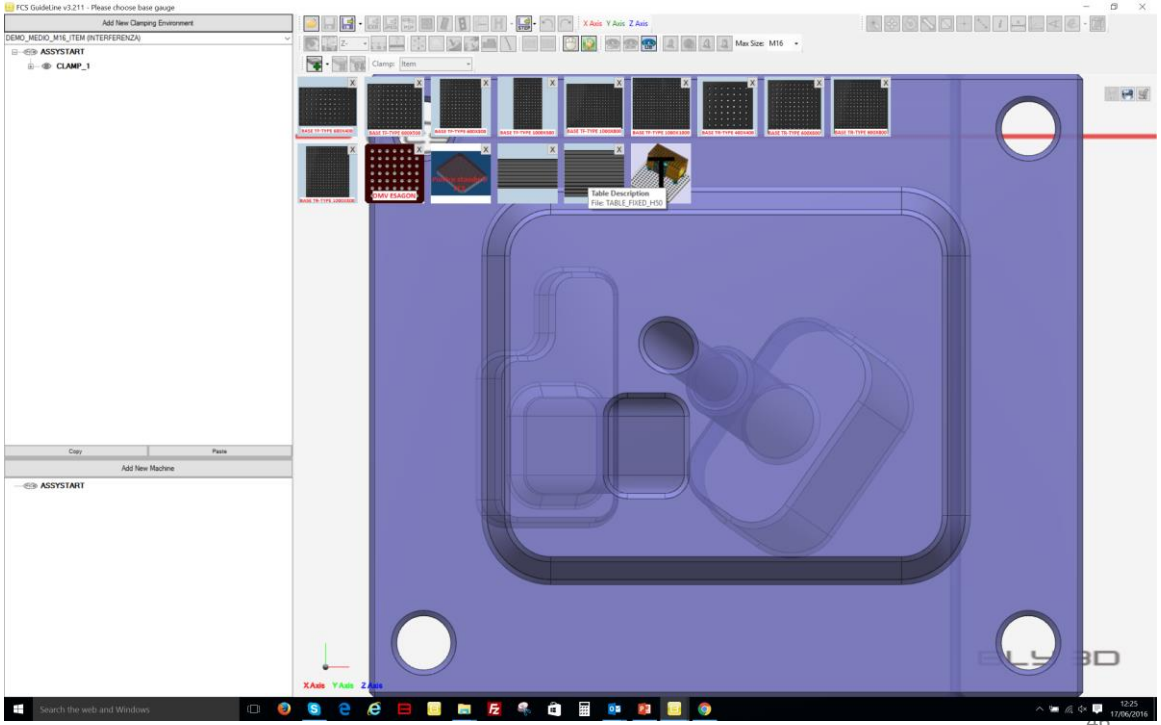
To create a new environment, click on «Add New Clampng Environment» on the tree: Guideline will add a new one and ask for a base gauge, as at the beginning. Once selected the base gauge, you can define the clamping as usual. You can activate each environment by clicking each one on the tree



❑ *Create clamping using rails*

FCS System offers the possibility to clamp on a T-Slot table also, without using a base gauge. To do that, you must mount at least two rails to the table (called fixed rails) and then 2 or more rails (called mobile rails) on top.

To use rails, select a T-Slot from the library.

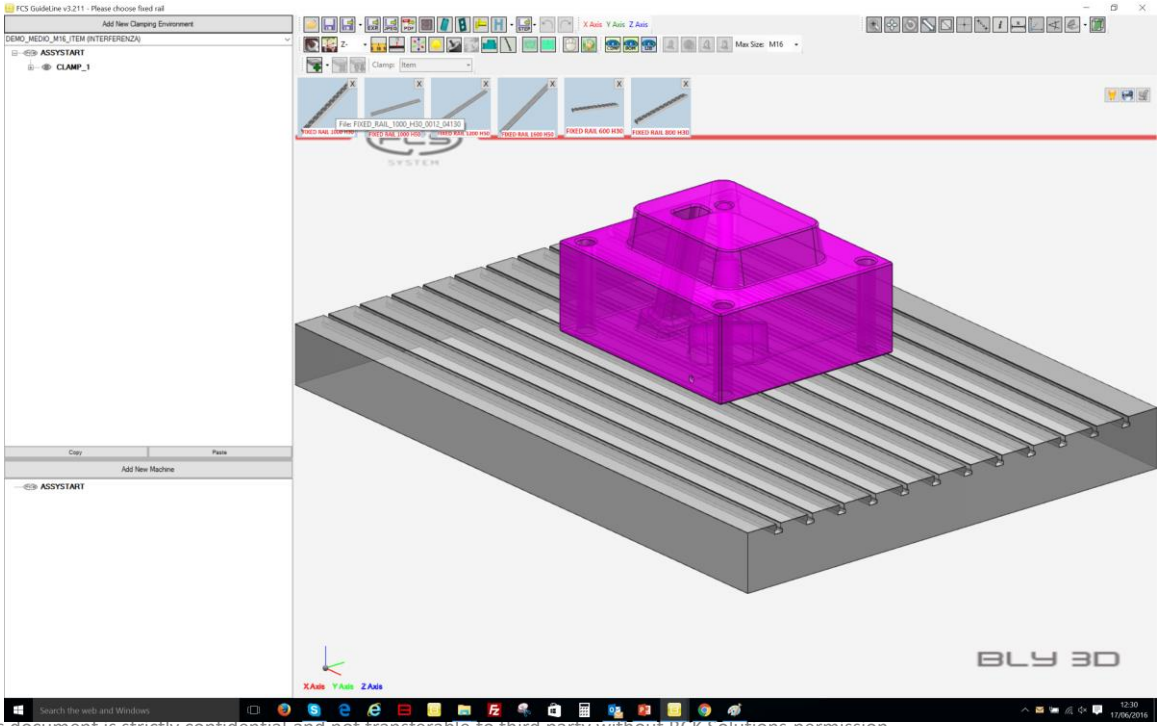


❑ *Create clamping using rails*

FCS System offers the possibility to clamp on a T-Slot table also, without using a base gauge. To do that, you must mount at least two rails to the table (called fixed rails) and then 2 or more rails (called mobile rails) on top.

To use rails, select a T-Slot from the library.
From the toolbar, select 

and then choose a fixed rail.



❑ *Create clamping using rails*

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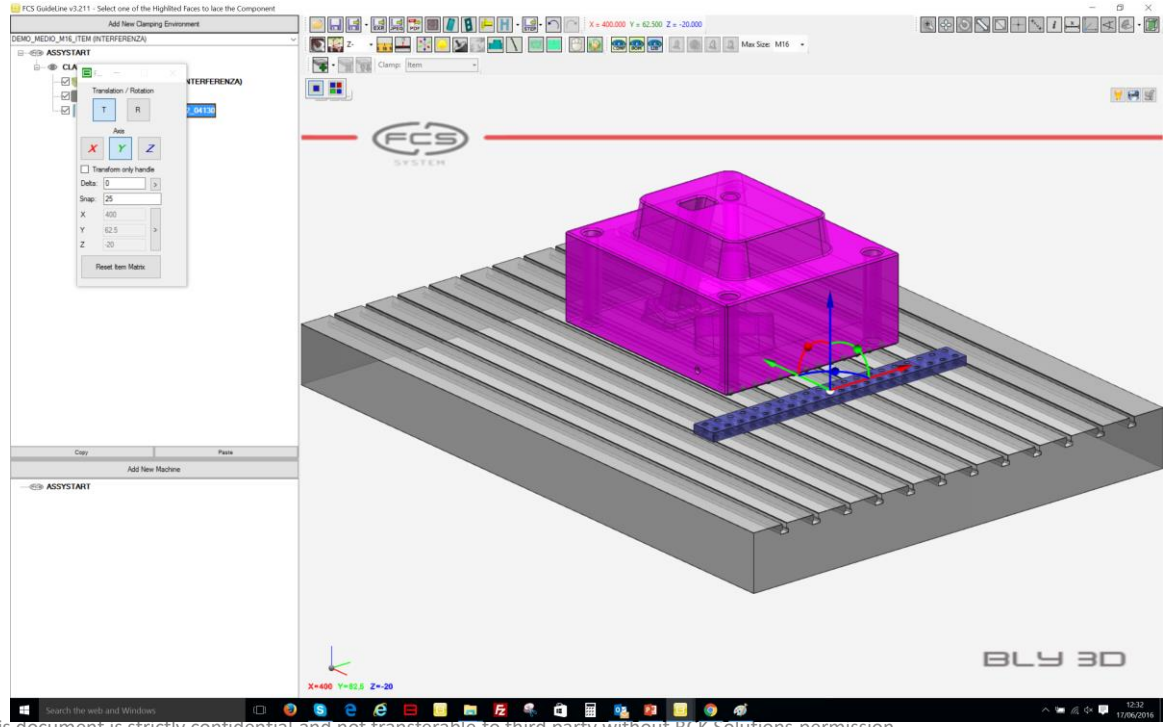
To use rails, select a T-Slot from the library.
From the toolbar, select 

and then choose a fixed rail.
Click the face of the T-Slot and then position the first rail in it.



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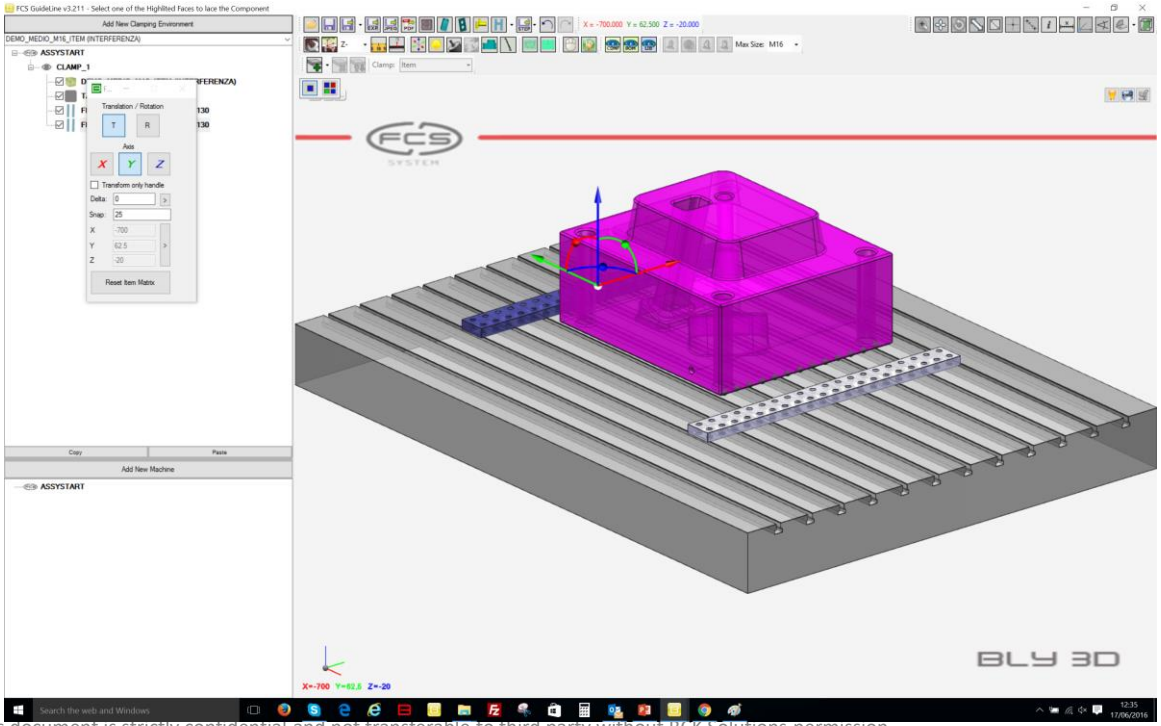
To use rails, select a T-Slot from the library.
From the toolbar, select 

and then choose a fixed rail.
Click the face of the T-Slot and then position the first rail in it.
Similarly, position the second fixed rail: please, note that, as in real life, Guideline does not execute any check at this stage, leaving to the user the responsibility of posistioning fixed rails at correct place, angle and pitch.



❑ *Create clamping using rails*

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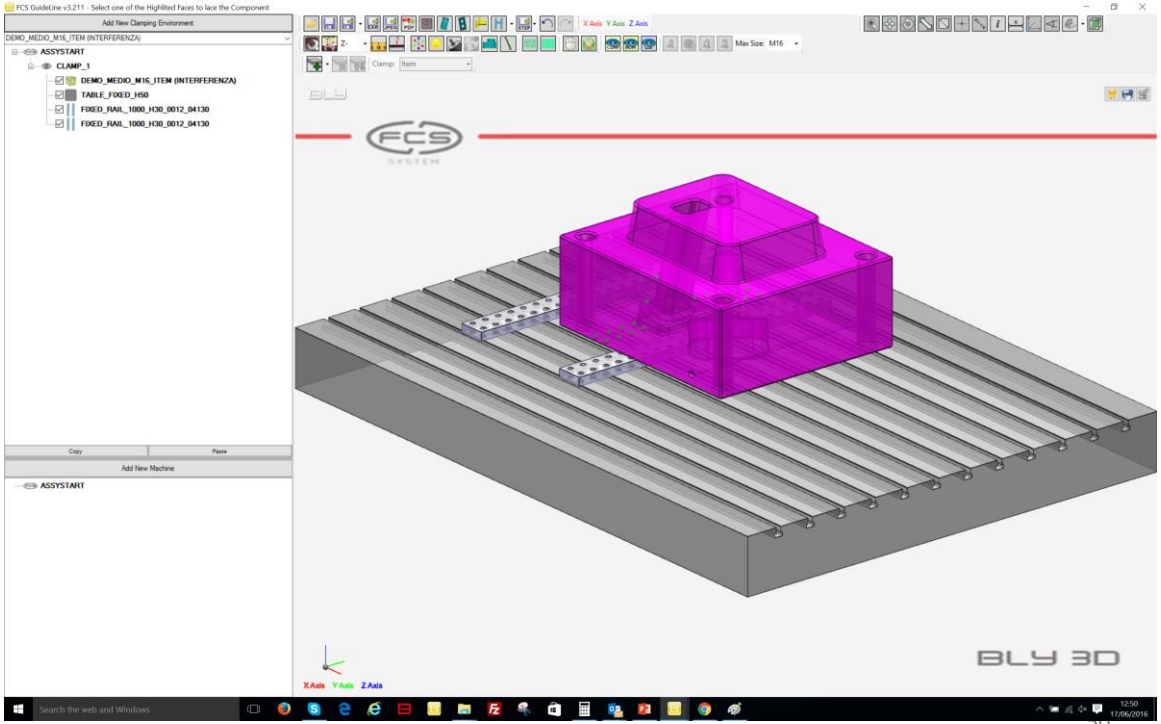
After placing two or more fixed rails, you can start placing the mobile ones.
From the toolbar, select 

and then choose a mobile rail.



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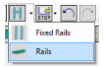
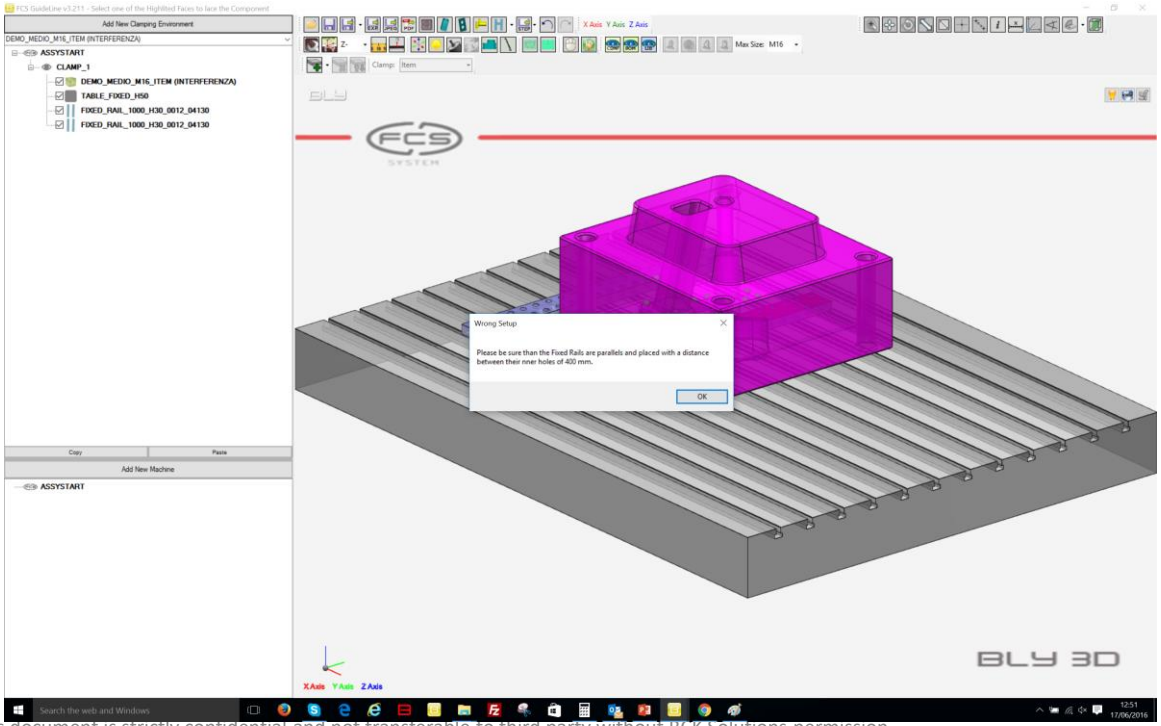
After placing two or more fixed rails, you can start placing the mobile ones.
From the toolbar, select 

and then choose a mobile rail.
Choose one upper face of a rail: Guideline checks the distance between fixed rails and does not place the mobile until that is not correct.



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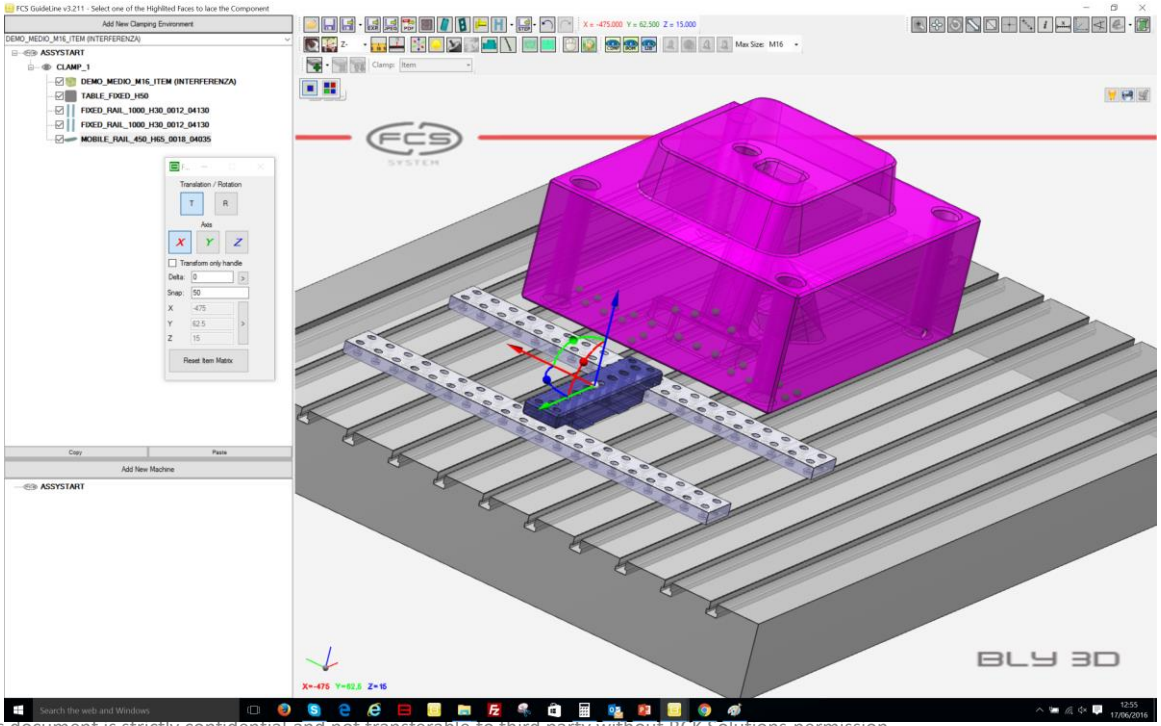
After placing two or more fixed rails, you can start placing the mobile ones.

From the toolbar, select 

and then choose a mobile rail.

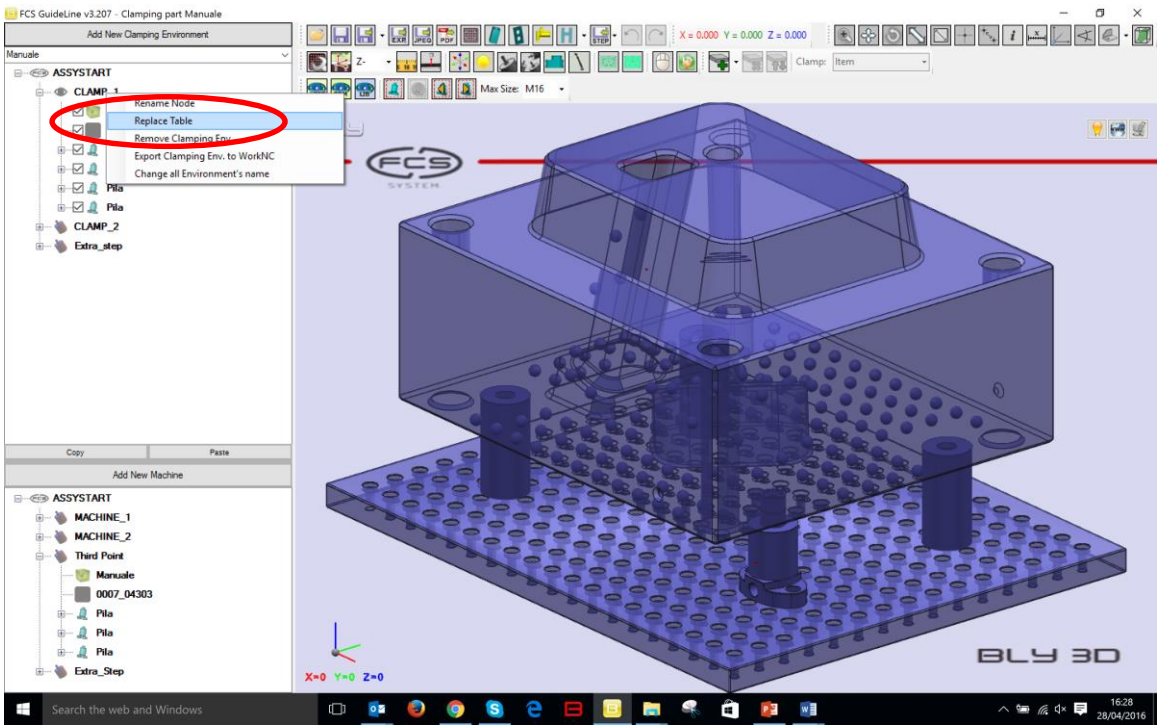
Choose one upper face of a rail: Guideline checks the distance between fixed rails and does not place the mobile until that is not correct.

Once placed, you can adjust mobile rail's position sliding it on the fixed rails and then clamp the piece.



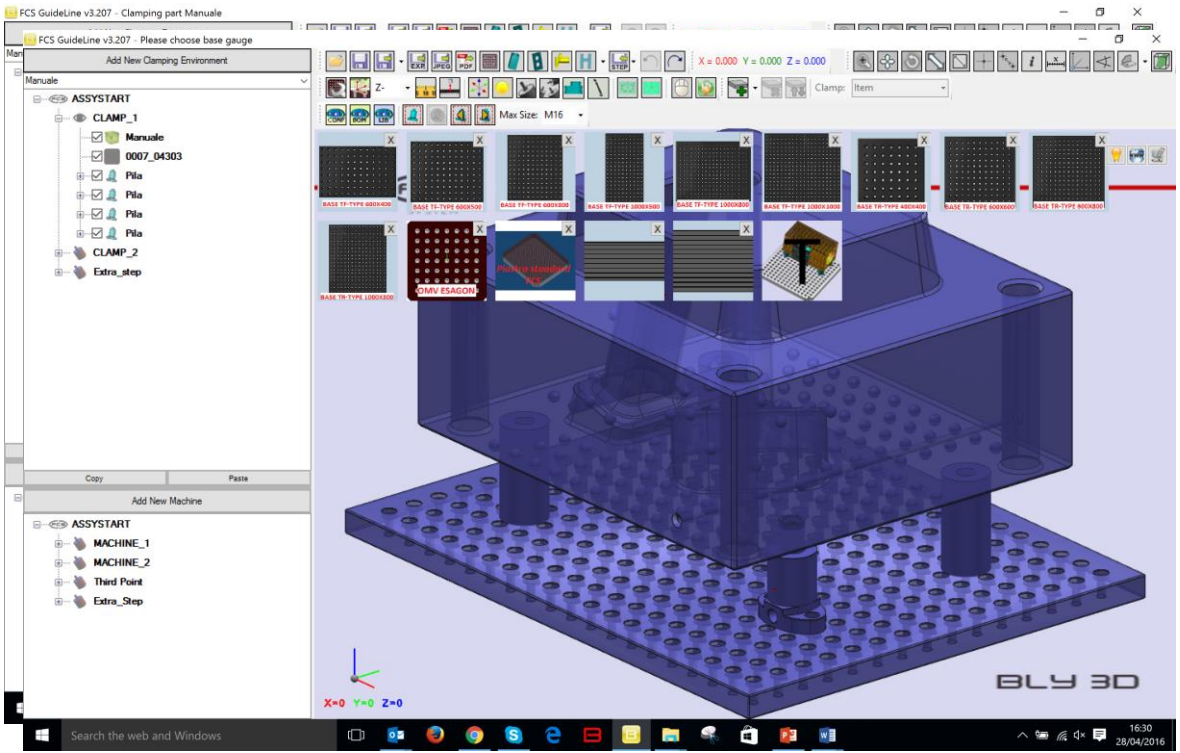
❑ *Replace base gauge*

It is possible to replace the base gauge for an existing clamping environment.
On the clamping tree, right click on the environment's node and choose «Replace Table».



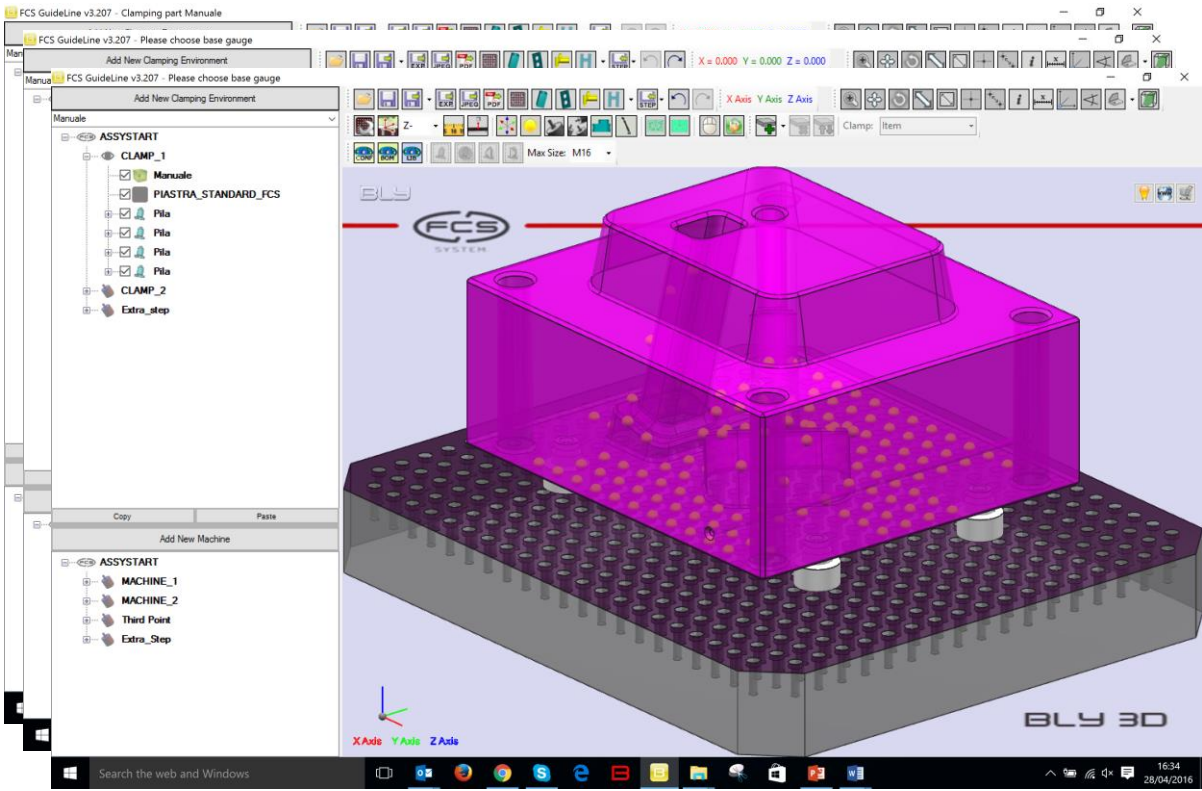
❑ *Replace base gauge (cont'd)*

It is possible to replace the base gauge for an existing clamping environment.
On the clamping tree, right click on the environment's node and choose «Replace Table».
The bases' library will be then displayed, so you can select the new gauge to be used.



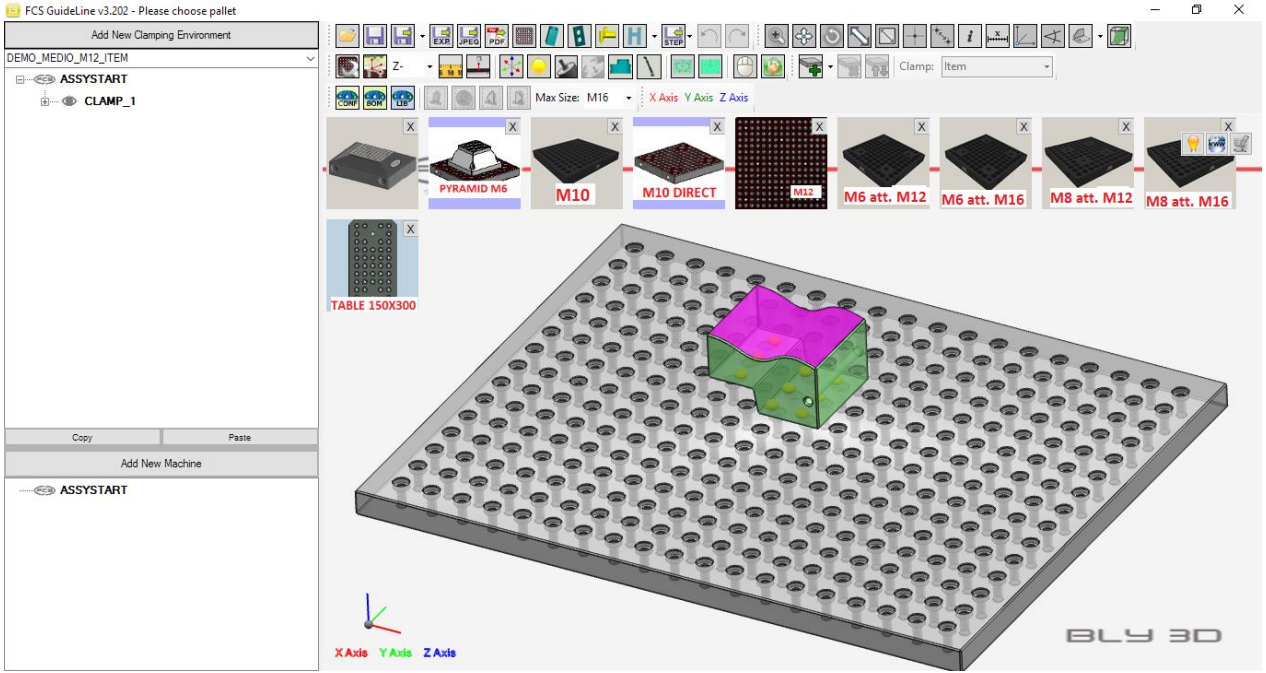
❑ *Replace base gauge (cont'd)*

It is possible to replace the base gauge for an existing clamping environment.
On the clamping tree, right click on the environment's node and choose «Replace Table».
The bases' library will be then displayed, so you can select the new gauge to be used.
Guideline will then replace it, trying to keep the stacks in the original position if possible.
N.B: stacks will be recalculated based on the minimum distance setting, therefore you will loose the height you could have positioned the piece, eventually.
However, Guideline will keep the orientation.



❑ *Optional: pallet placement*

For small pieces, FCS System propose the usage of additional placement bases, named «pallets».
Guideline detects the dimensions of the workpiece and, in case, propose the selection of the pallet after the selection of the base gauge.
The example in the picture shows you the library of pallet included with the software.



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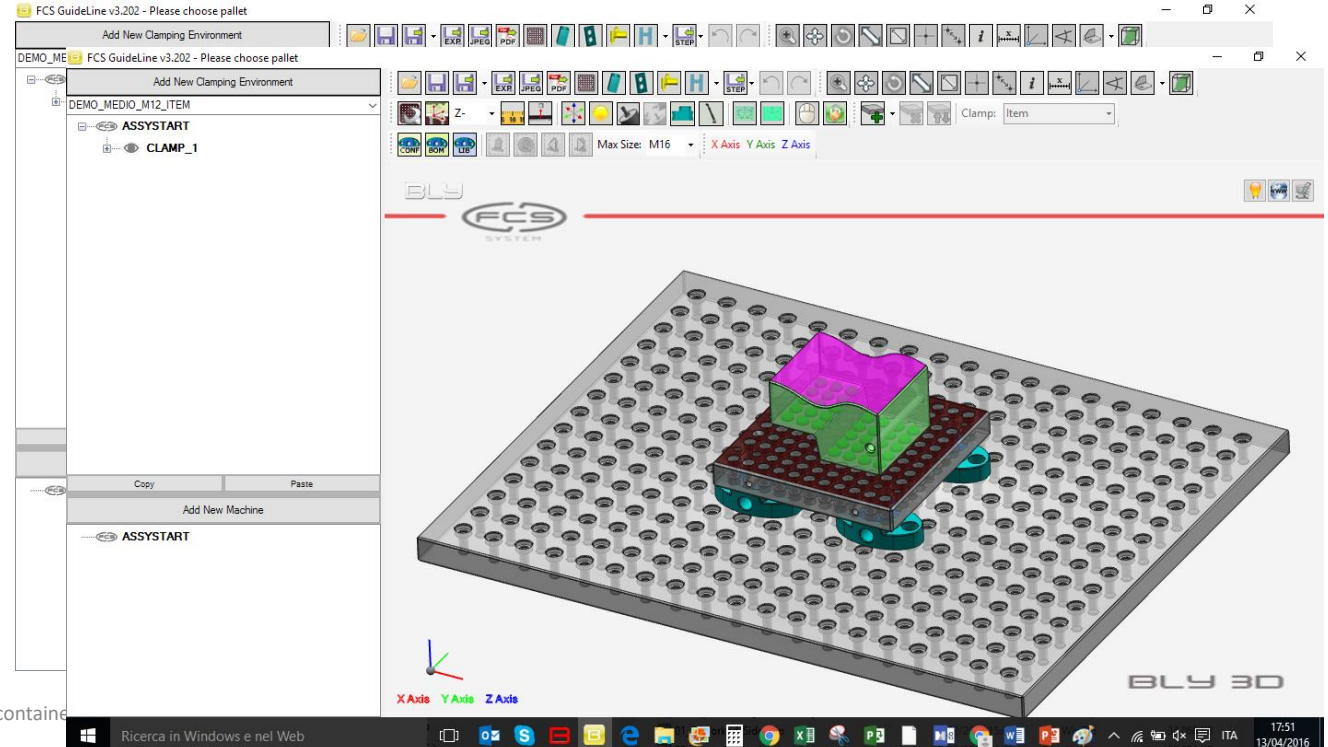
After the selection of the pallet, this one is placed witht the same mechanism of the base gauge, with a predifined set of components.



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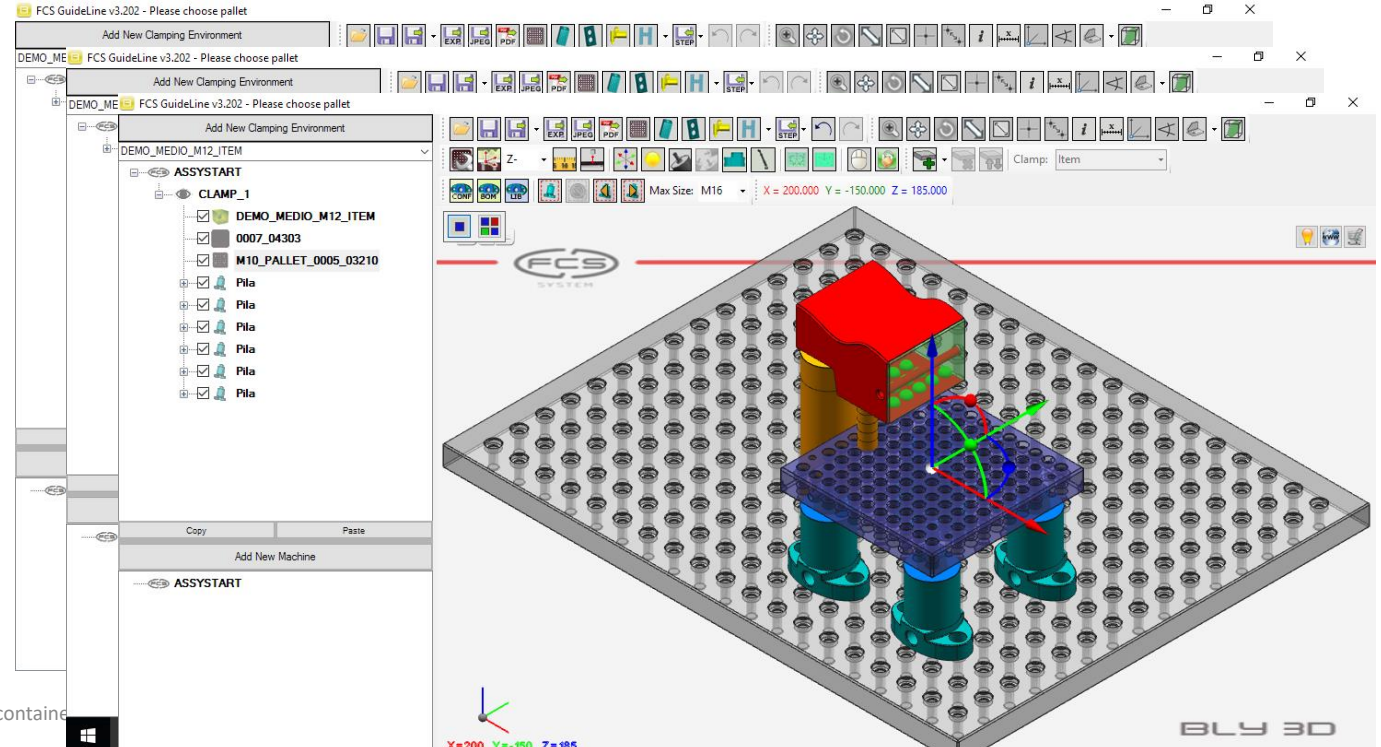
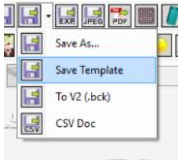
After the selection of the pallet, this one is placed witht the same mechanism of the base gauge, with a predifined set of components.
It is possible to reposition pallet and workpiece indipendently

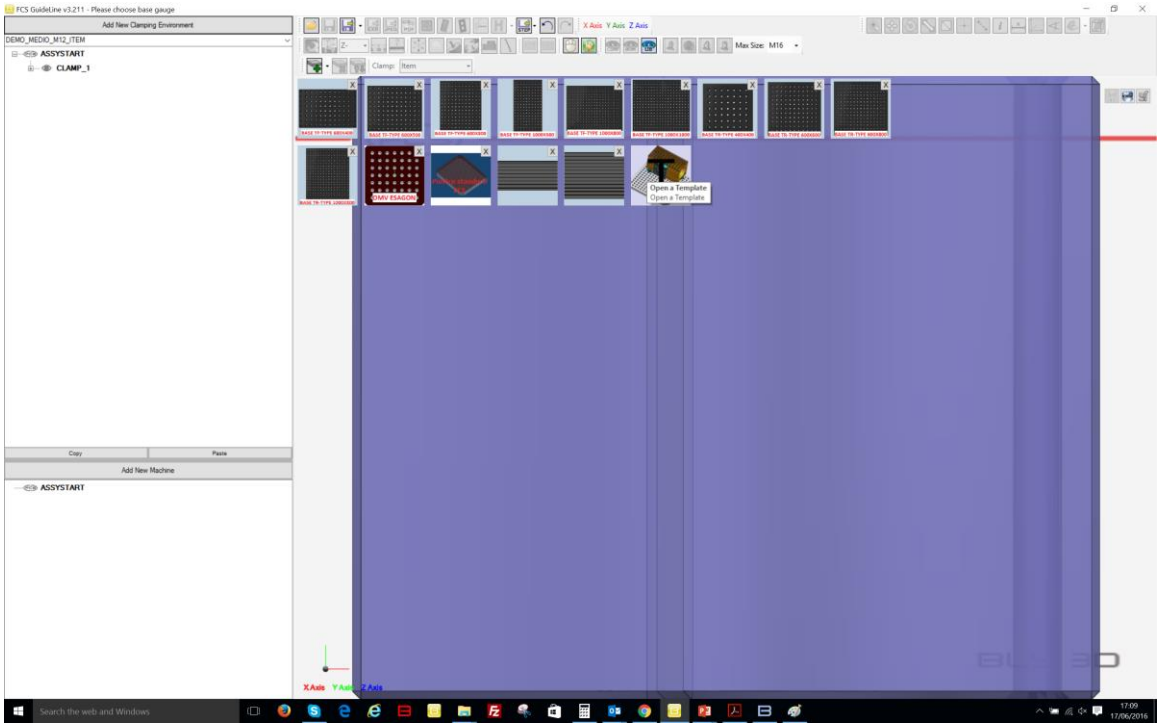


❑ *Optional: Template creation*

It is possible to store a specific clamping configuration in a template, i.e. base gauge + pallet, as a basis for further clampings.

After having defined the configuration you want to store, from the toolbar press 

and store the bly on the hard disk. To reuse it, open a piece, click «Open a Template» from the library and select the bly file.



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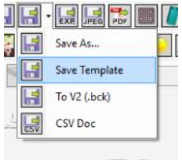
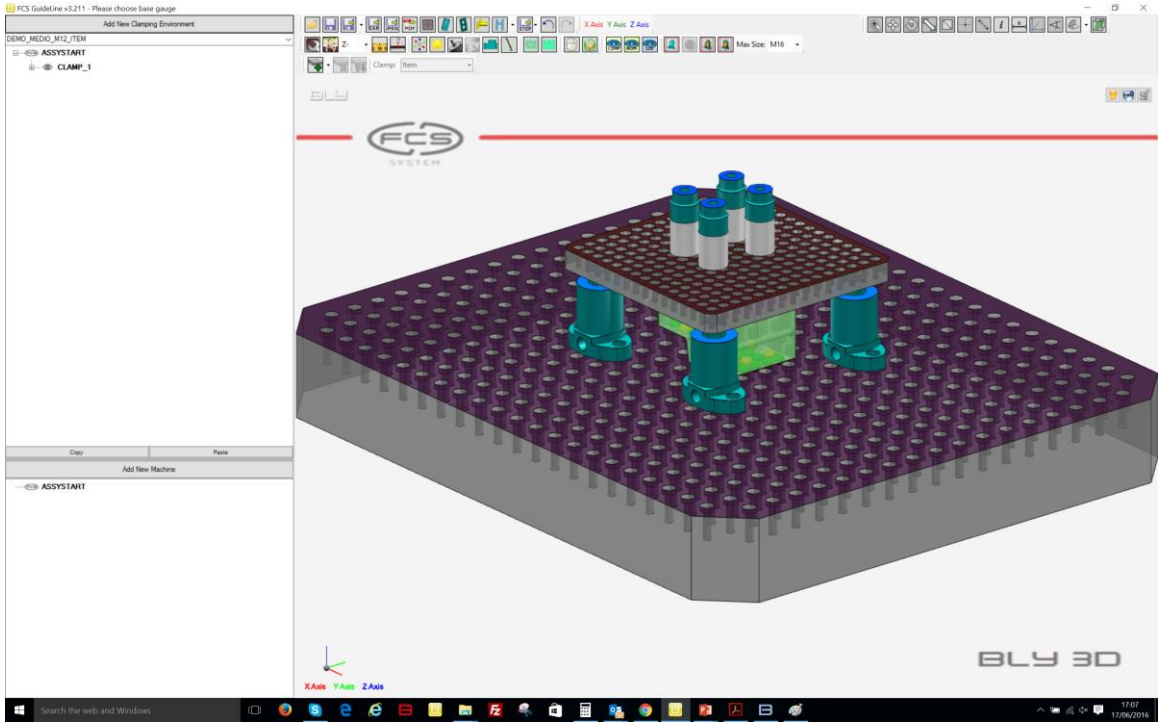
After having defined the configuration you want to store, from the toolbar press 

and store the bly on the hard disk.
To reuse it, open a piece, click «Open a Template» from the library and select the bly file.
Once applied, the piece may be placed in funny positions: simply drag it above the table or pallet and then clamp it as usual



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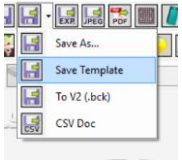

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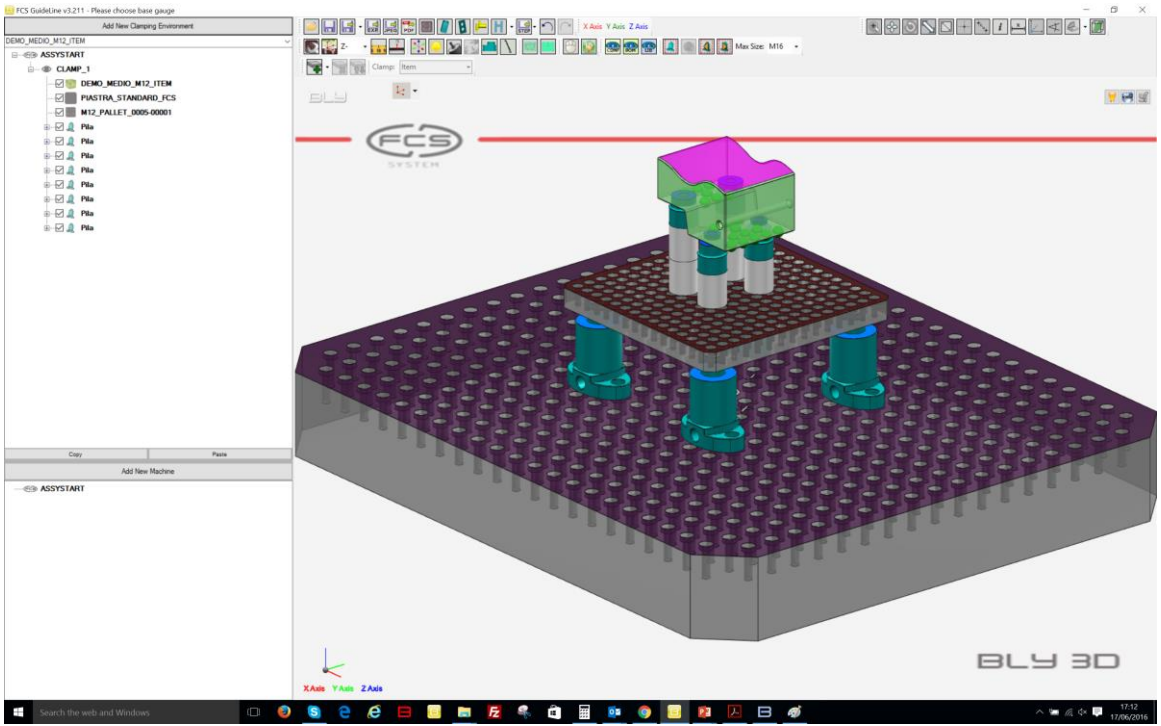
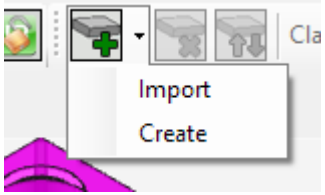


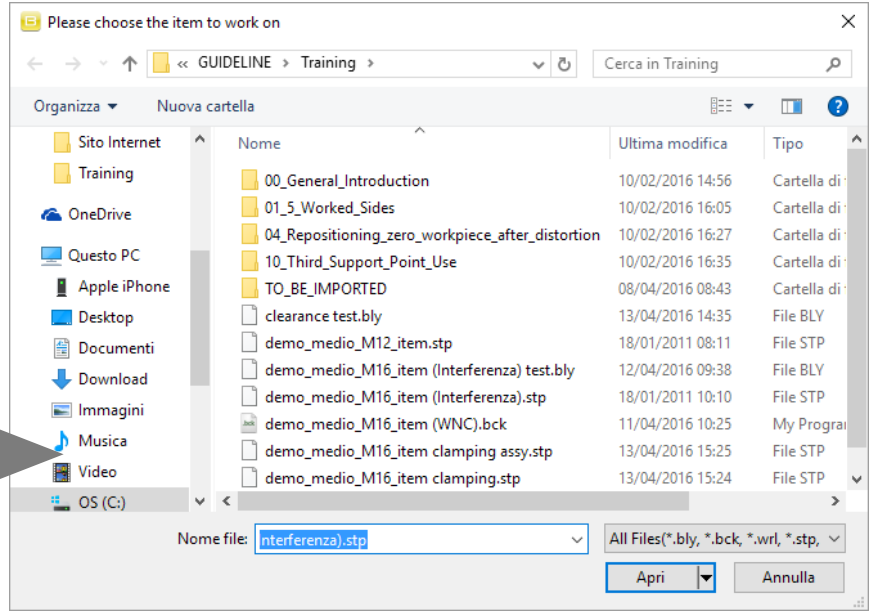
❑ *Optional: Stock creation/import*

It is possible to import or create the stock in Guideline and define its clamping (usually the first clamping in absolute). We reccomend to define the clamping of the stock in a dedicated clamping environement.

To start the import, select «Import» from the Add Stock drop down menu



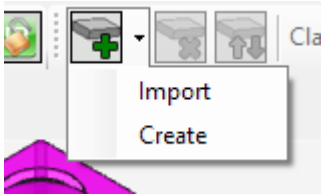
Guideline opens the selection window and let you choose the file



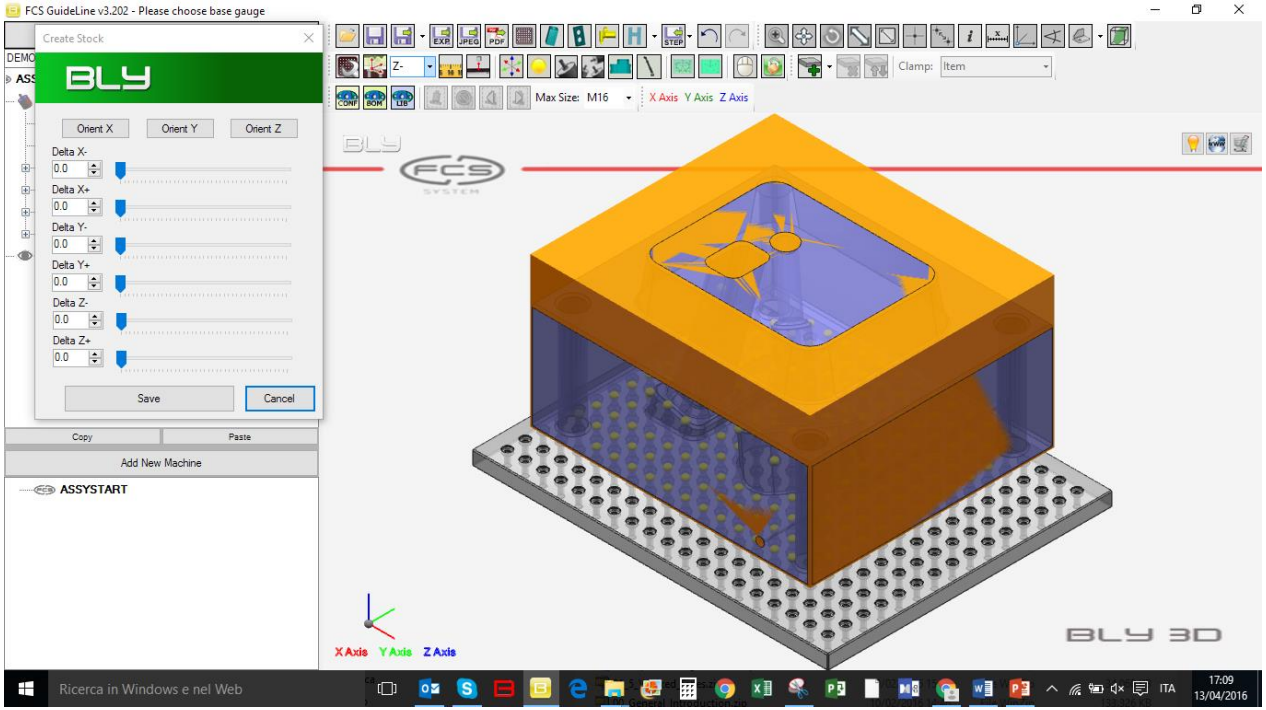
It is important to notice that the model of the stock must be prepared in advance with the desired dimensions and the correct origin (coincident with the workpiece's origin), since Guideline do not let you reposition the stock independently form the workpiece.
This is a designed feature in order to avoid unwanted misaligned, since the aligned is typically defined in the CAD

❑ *Optional: Stock creation/import (cont'd)*

To create the stock, select «Create» from the Add Stock drop down menu

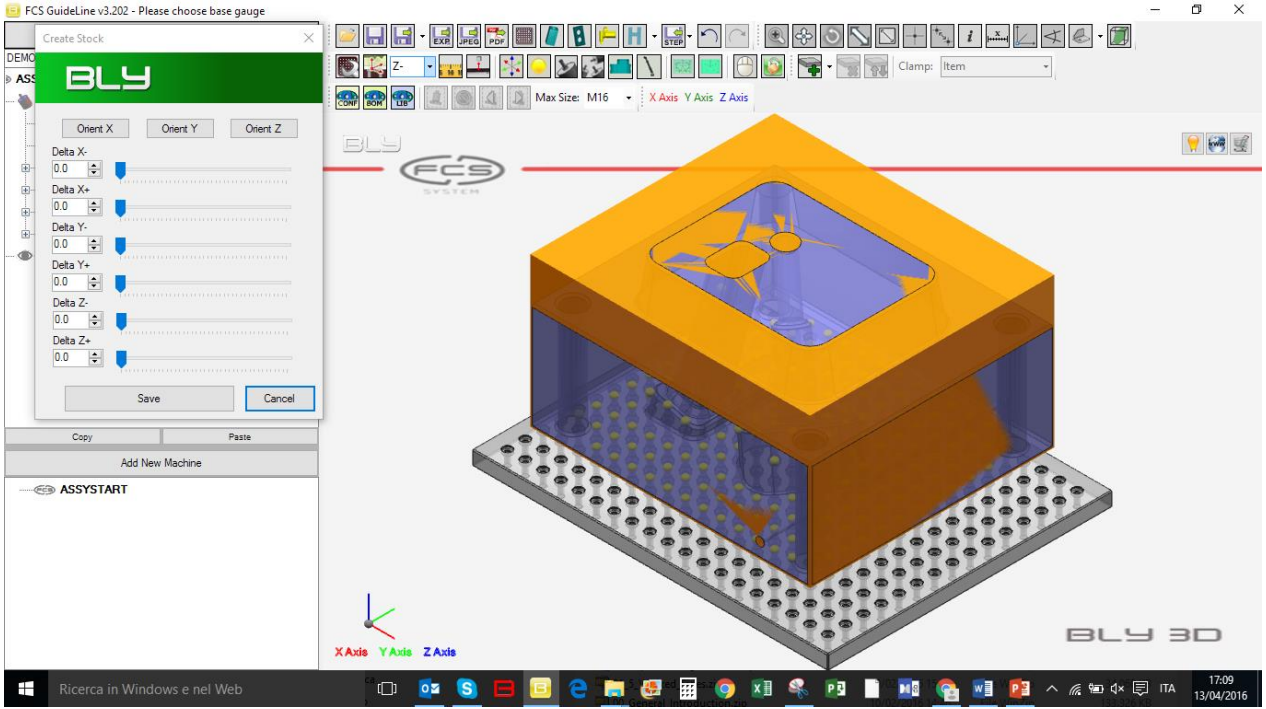


Guideline creates a stock around the workpiece using the bounding box concept (minimum cube containing the workpiece)



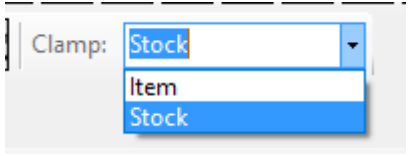
❑ *Optional: Stock creation/import (cont'd)*

You can specify the stock's offset by typing the values into the boxes or using the sliders.
Maximum delta is 40mm.
Once defined, press «Save» to finally create the stock

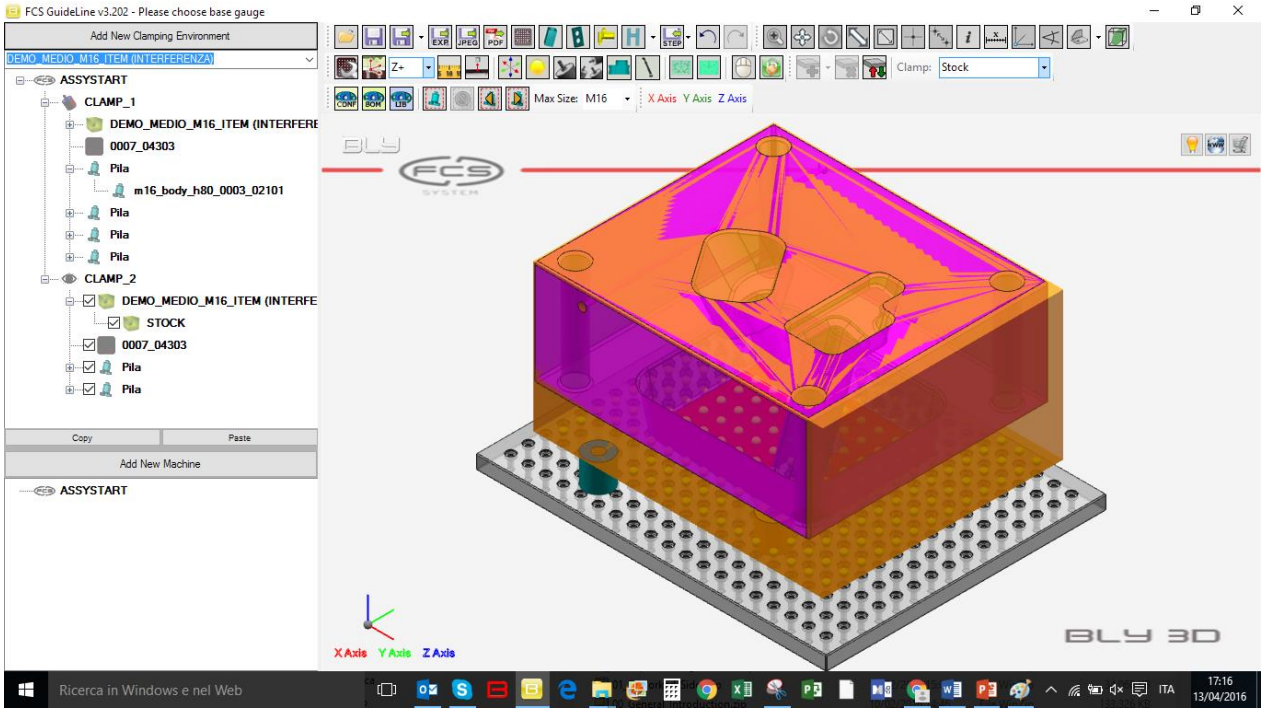


❑ *Optional: Stock creation/import (cont'd)*

It is possible to clamp the stock following the same procedure used for the workpiece: select the option «Stock» from the «Clamp» dropdown menu on the toolbar

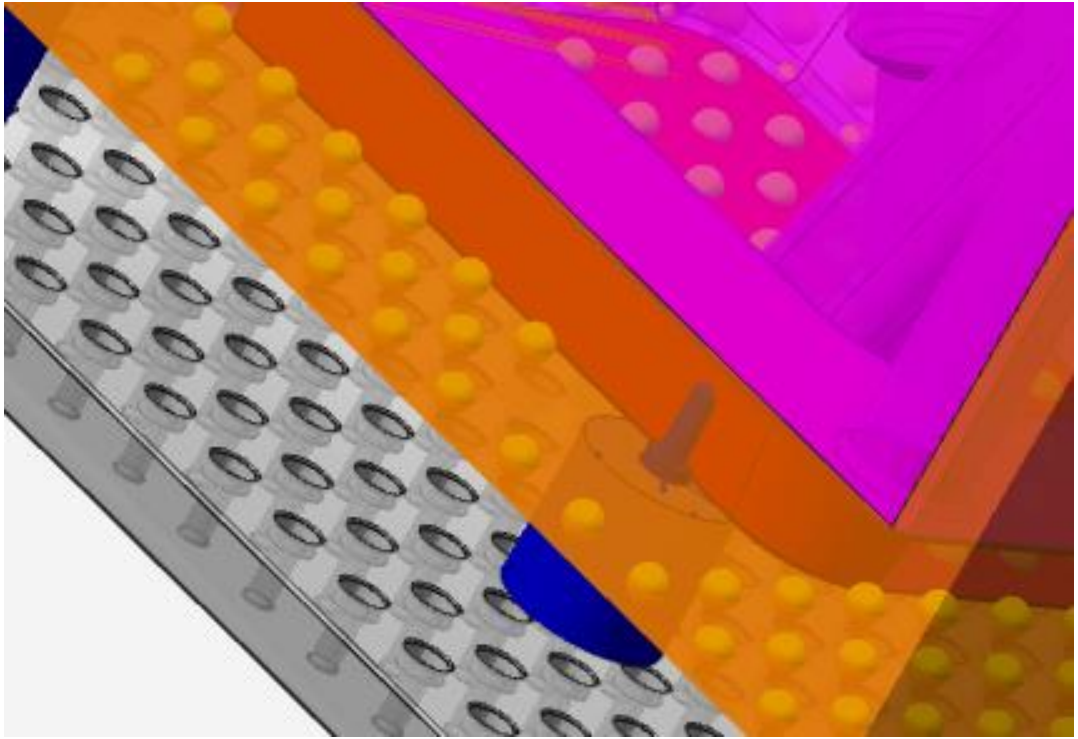


Placement spheres are then projected onto the stock's face



❑ *Optional: Stock creation/import (cont'd)*

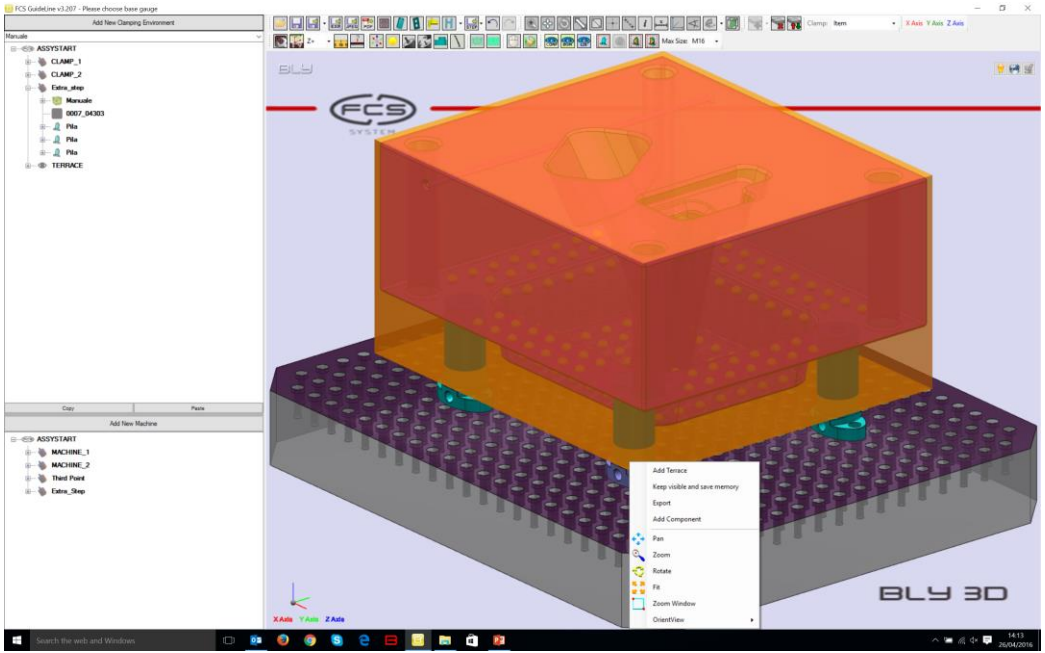
Interference works for the stock placement also: it highlights the interferences between components or tools and workpiece's geometry other than the face overlapping the stock face (if any).
In the picture, the tool interferes with the taper and the radius, which are highlighted as warning: as usual, it is up to the user to analyze the situation and fix it eventually.



❑ *Optional: Terraces*

From this release, it is possible to model terraces, additional material in the stock for placement purposes, which will be removed during workpiece's roughing or finishing.
In order to create them, you need to have a stock created or imported and be in placement mode of the workpiece.

To access the command, right-click onto a stack and choose «Add Terrace».

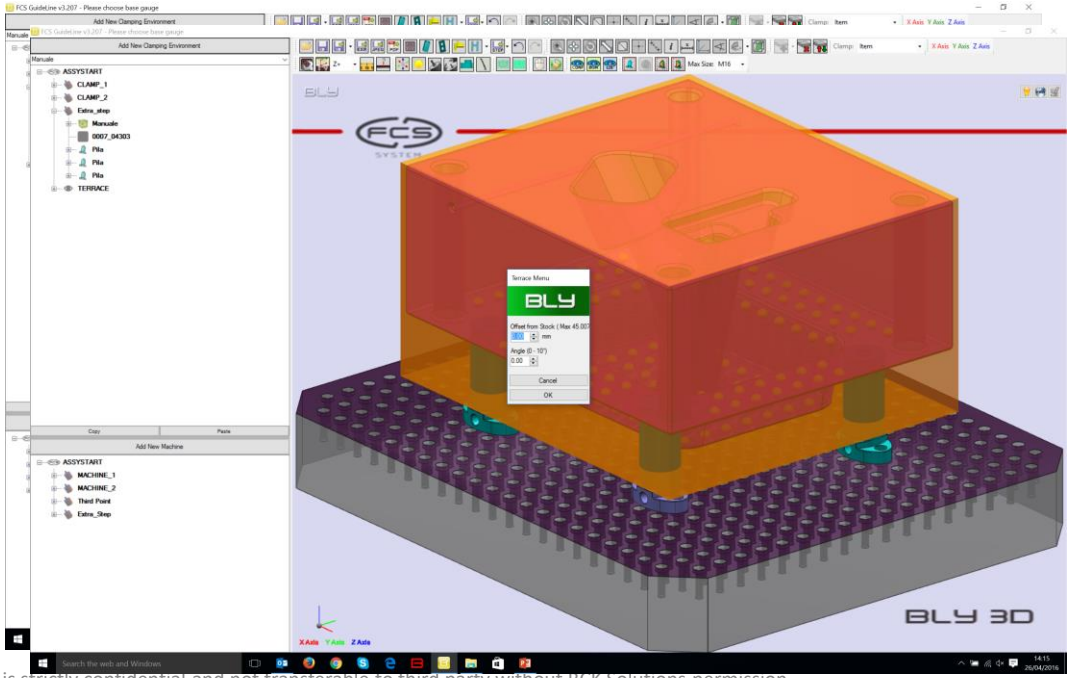


❑ *Optional: Terraces (cont'd)*

From this release, it is possible to model terraces, additional material in the stock for placement purposes, which will be removed during workpiece's roughing or finishing.
In order to create them, you need to have a stock created or imported and be in placement mode of the workpiece.

To access the command, right-click onto a stack and choose «Add Terrace».
In the terrace menu, you can specify the following parameters:

- Offset from stock: creates a terrace having the specified distance from the stock's face
- Angle: add a taper witht the specified angle



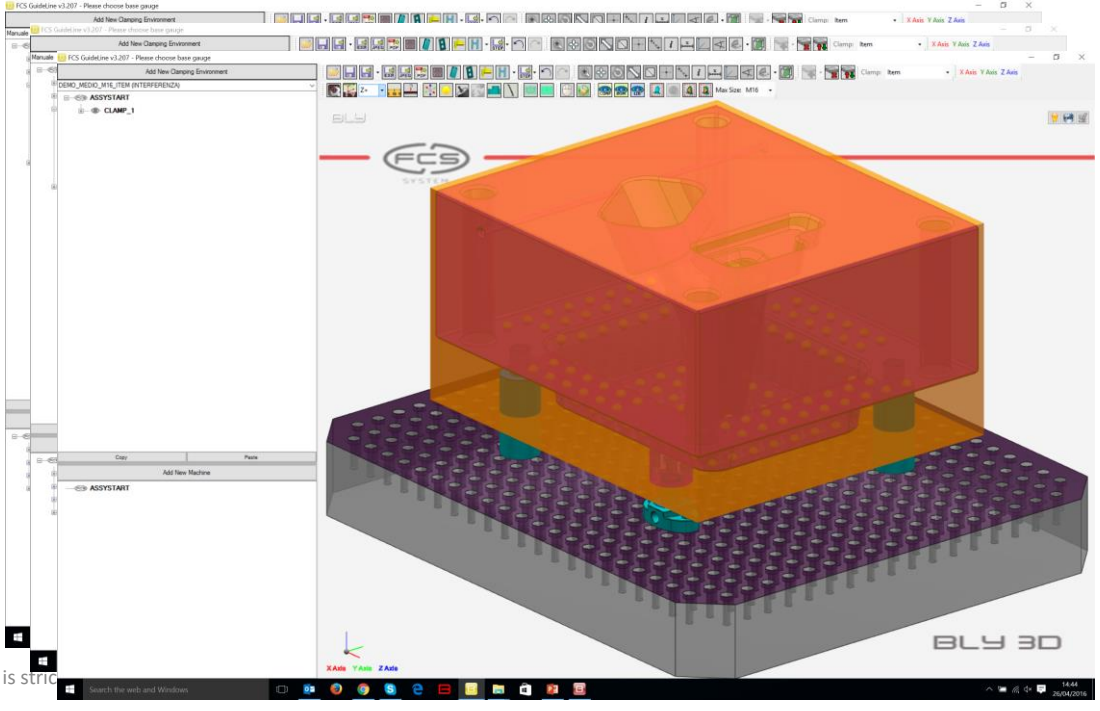
❑ *Optional: Terraces (cont'd)*

From this release, it is possible to model terraces, additional material in the stock for placement purposes, which will be removed during workpiece's roughing or finishing.
In order to create them, you need to have a stock created or imported and be in placement mode of the workpiece.

To access the command, right-click onto a stack and choose «Add Terrace».
In the terrace menu, you can specify the following parameters:

- Offset from stock: creates a terrace having the specified distance from the stock's face
- Angle: add a taper witht the specified angle

Once hit OK, the terrace is created and the stack recalculated accordingly.



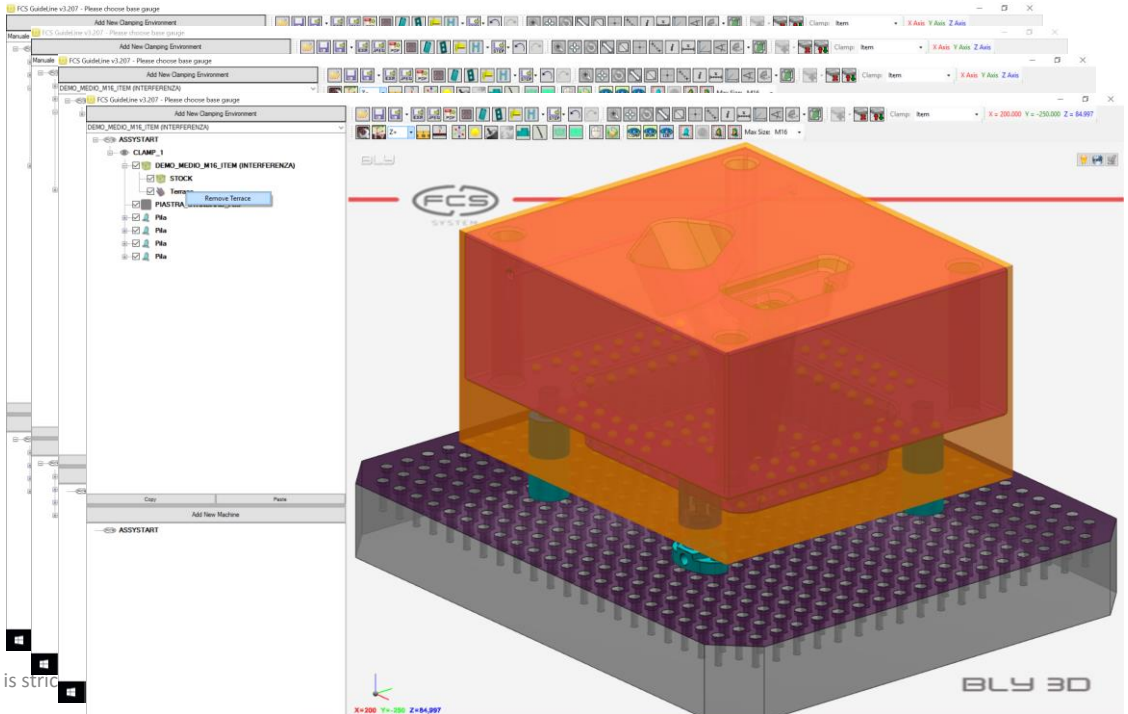
❑ *Optional: Terraces (cont'd)*

From this release, it is possible to model terraces, additional material in the stock for placement purposes, which will be removed during workpiece's roughing or finishing.
In order to create them, you need to have a stock created or imported and be in placement mode of the workpiece.

To access the command, right-click onto a stack and choose «Add Terrace».
In the terrace menu, you can specify the following parameters:

- Offset from stock: creates a terrace having the specified distance from the stock's face
- Angle: add a taper witht the specified angle

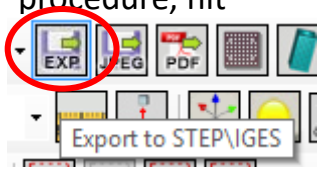
Once hit OK, the terrace is created and the stack recalculated accordingly.
To remove a terrace, right-click on it in the clamping tree and choose «Remove Terrace»



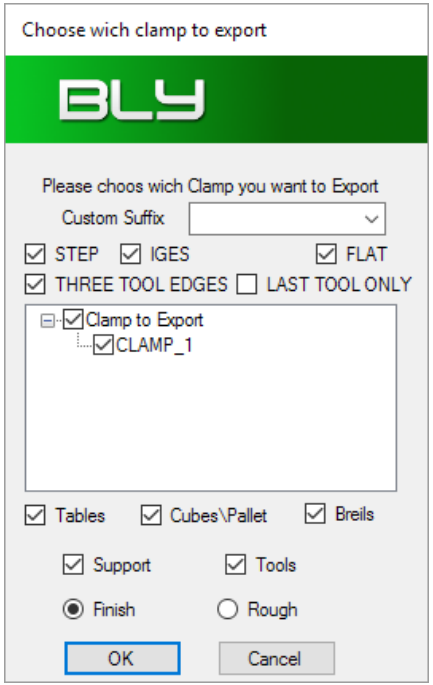
❑ *Export the clamping definition to CAD*

Once the clamping has been defined, it is possible to export the geometry of the FCS' components and the placement seats to a STEP or IGES file, which can then be used as exchange file to a 3D CAD system.
It is important to remember that the command works on the active environment (CAD or CAM) every time.

To start the procedure, hit



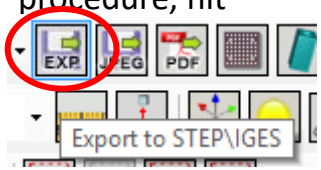
It opens the export options window.



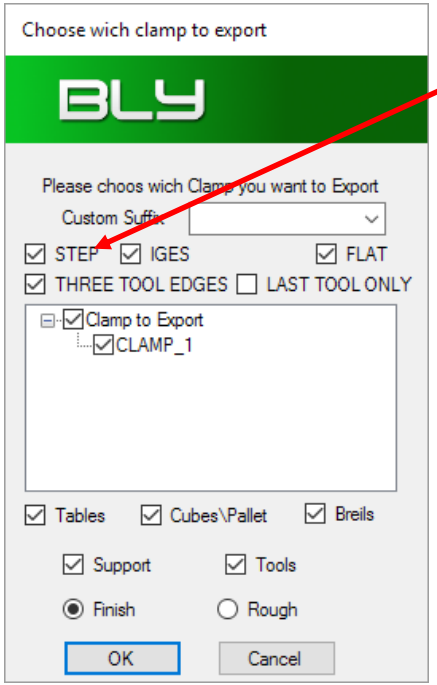
❑ *Export the clamping definition to CAD*

Once the clamping has been defined, it is possible to export the geometry of the FCS' components and the placement seats to a STEP or IGES file, which can then be used as exchange file to a 3D CAD system.
It is important to remember that the command works on the active environment (CAD or CAM) every time.

To start the procedure, hit



It opens the export options window.

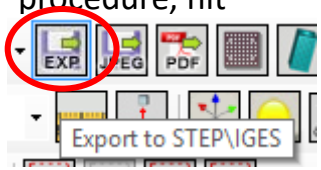


- STEP: creates the STEP file

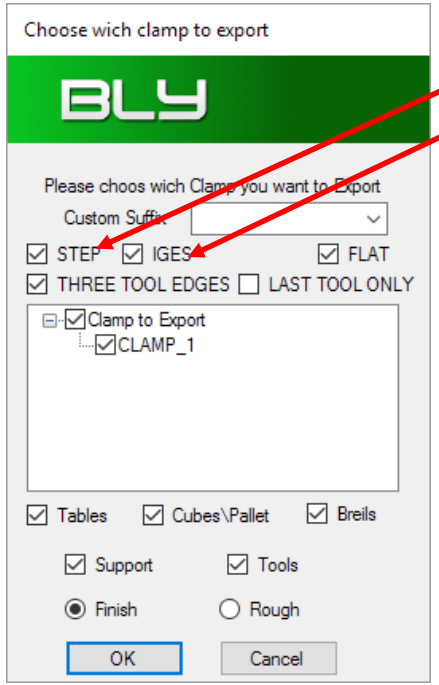
❑ *Export the clamping definition to CAD*

Once the clamping has been defined, it is possible to export the geometry of the FCS' components and the placement seats to a STEP or IGES file, which can then be used as exchange file to a 3D CAD system.
It is important to remember that the command works on the active environment (CAD or CAM) every time.

To start the procedure, hit



It opens the export options window.

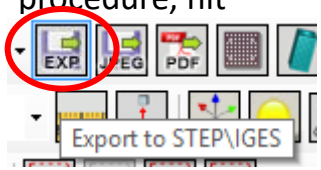


- STEP: creates the STEP file
- IGES: creates the IGES file

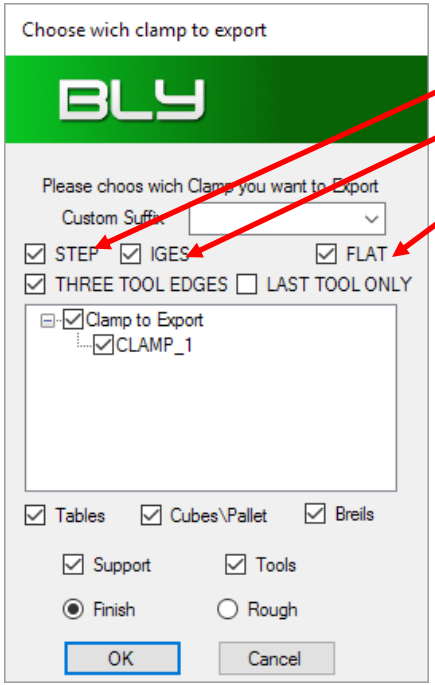
❑ *Export the clamping definition to CAD*

Once the clamping has been defined, it is possible to export the geometry of the FCS' components and the placement seats to a STEP or IGES file, which can then be used as exchange file to a 3D CAD system.
It is important to remember that the command works on the active environment each time.

To start the procedure, hit



It opens the export options window.

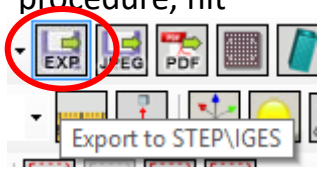


- STEP: creates the STEP file
- IGES: creates the IGES file
- FLAT: places all geometry as single component, without creating assembly structure

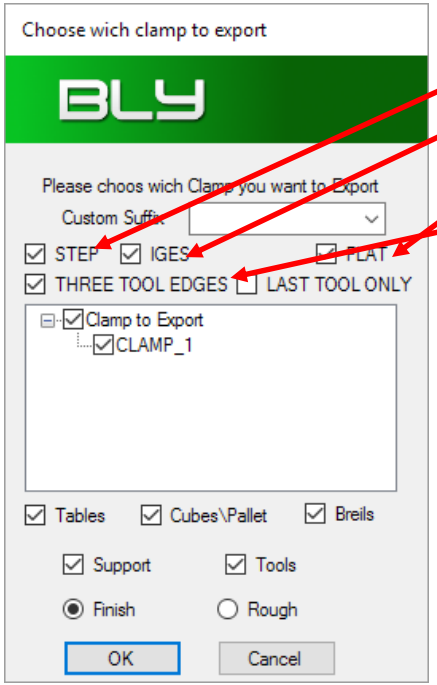
❑ *Export the clamping definition to CAD*

Once the clamping has been defined, it is possible to export the geometry of the FCS' components and the placement seats to a STEP or IGES file, which can then be used as exchange file to a 3D CAD system.
It is important to remember that the command works on the active environment (CAD or CAM) every time.

To start the procedure, hit



It opens the export options window.

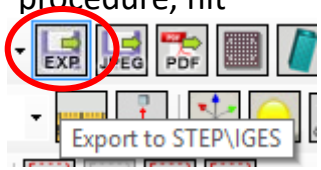


- STEP: creates the STEP file
- IGES: creates the IGES file
- FLAT: places all geometry as single component, without creating assembly structure
- THREE TOOL EDGES: keeps the faces of the tools' pin separated. Unthincking it will merge them (depending on CAD preferences)

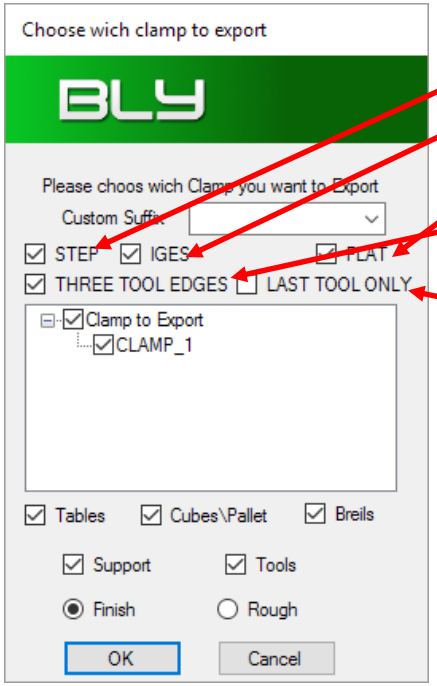
❑ *Export the clamping definition to CAD*

Once the clamping has been defined, it is possible to export the geometry of the FCS' components and the placement seats to a STEP or IGES file, which can then be used as exchange file to a 3D CAD system.
It is important to remember that the command works on the active environment (CAD or CAM) every time.

To start the procedure, hit



It opens the export options window.

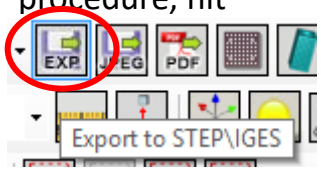


- STEP: creates the STEP file
- IGES: creates the IGES file
- FLAT: places all geometry as single component, without creating assembly structure
- THREE TOOL EDGES: keeps the faces of the tools' pin separated. Unthincking it will merge them (depending on CAD preferences)
- LAST TOOL ONLY: export the tool of the component placed at the top of the stack

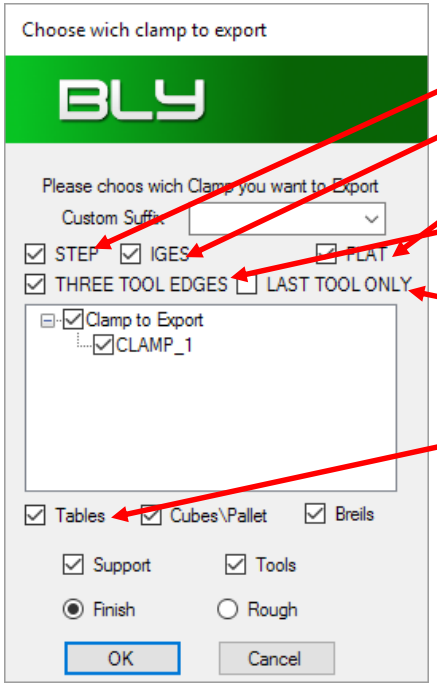
❑ *Export the clamping definition to CAD*

Once the clamping has been defined, it is possible to export the geometry of the FCS' components and the placement seats to a STEP or IGES file, which can then be used as exchange file to a 3D CAD system.
It is important to remember that the command works on the active environment (CAD or CAM) every time.

To start the procedure, hit



It opens the export options window.

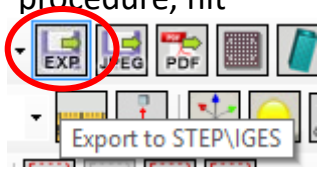


- STEP: creates the STEP file
- IGES: creates the IGES file
- FLAT: places all geometry as single component, without creating assembly structure
- THREE TOOL EDGES: keeps the faces of the tools' pin separated. Unthincking it will merge them (depending on CAD preferences)
- LAST TOOL ONLY: export the tool of the component placed at the top of the stack
- TABLES: includes base gauge's geometry

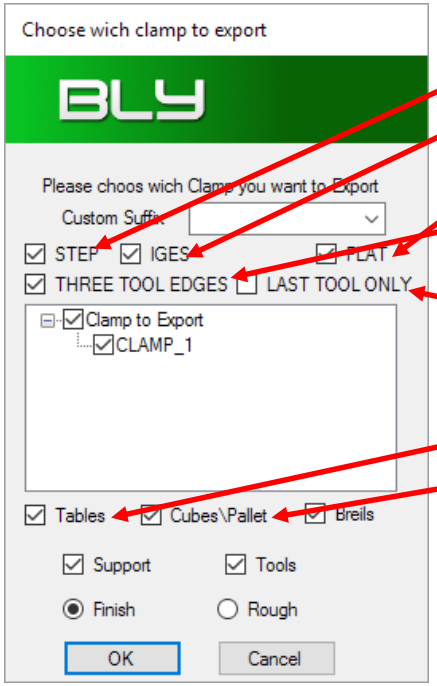
❑ *Export the clamping definition to CAD*

Once the clamping has been defined, it is possible to export the geometry of the FCS' components and the placement seats to a STEP or IGES file, which can then be used as exchange file to a 3D CAD system.
It is important to remember that the command works on the active environment (CAD or CAM) every time.

To start the procedure, hit



It opens the export options window.

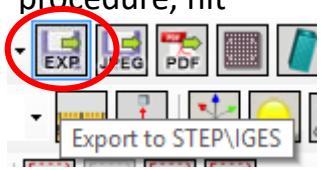


- STEP: creates the STEP file
- IGES: creates the IGES file
- FLAT: places all geometry as single component, without creating assembly structure
- THREE TOOL EDGES: keeps the faces of the tools' pin separated. Unthincking it will merge them (depending on CAD preferences)
- LAST TOOL ONLY: export the tool of the component placed at the top of the stack
- TABLES: includes base gauge's geometry
- CUBES/PALLET: includes pallet, shoulders and cubes geometry

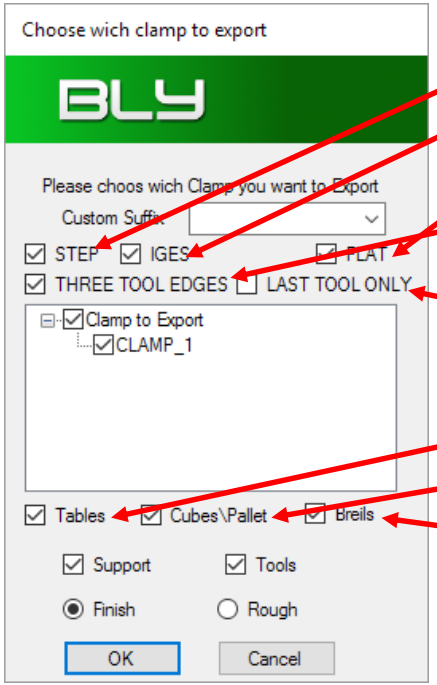
❑ *Export the clamping definition to CAD*

Once the clamping has been defined, it is possible to export the geometry of the FCS' components and the placement seats to a STEP or IGES file, which can then be used as exchange file to a 3D CAD system.
It is important to remember that the command works on the active environment (CAD or CAM) every time.

To start the procedure, hit



It opens the export options window.

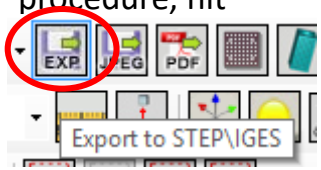


- STEP: creates the STEP file
- IGES: creates the IGES file
- FLAT: places all geometry as single component, without creating assembly structure
- THREE TOOL EDGES: keeps the faces of the tools' pin separated. Unthincking it will merge them (depending on CAD preferences)
- LAST TOOL ONLY: export the tool of the component placed at the top of the stack
- TABLES: includes base gauge's geometry
- CUBES/PALLET: includes pallet, shoulders and cubes geometry
- BREYLS: includes breyls' geometry. Activates also SUPPORT and TOOLS options.

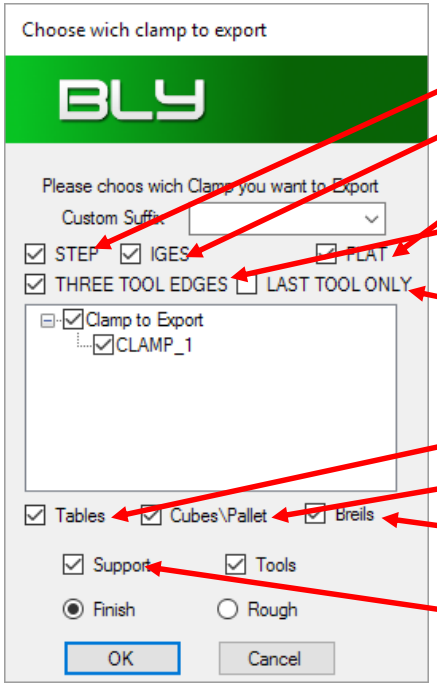
❑ *Export the clamping definition to CAD*

Once the clamping has been defined, it is possible to export the geometry of the FCS' components and the placement seats to a STEP or IGES file, which can then be used as exchange file to a 3D CAD system.
It is important to remember that the command works on the active environment (CAD or CAM) every time.

To start the procedure, hit



It opens the export options window.

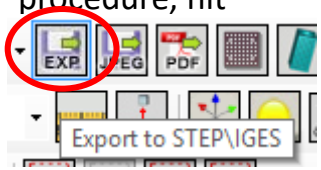


- STEP: creates the STEP file
- IGES: creates the IGES file
- FLAT: places all geometry as single component, without creating assembly structure
- THREE TOOL EDGES: keeps the faces of the tools' pin separated. Unthincking it will merge them (depending on CAD preferences)
- LAST TOOL ONLY: export the tool of the component placed at the top of the stack
- TABLES: includes base gauge's geometry
- CUBES/PALLET: includes pallet, shoulders and cubes geometry
- BREYLS: includes breyls' geometry. Activates also SUPPORT and TOOLS options.
  - SUPPORT: includes supports' geometry

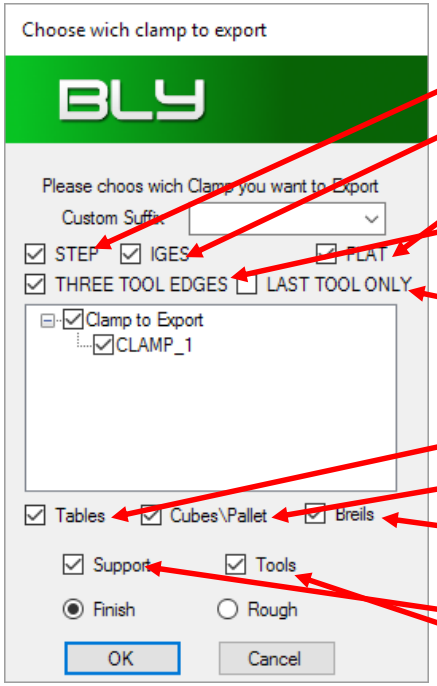
❑ *Export the clamping definition to CAD*

Once the clamping has been defined, it is possible to export the geometry of the FCS' components and the placement seats to a STEP or IGES file, which can then be used as exchange file to a 3D CAD system.
It is important to remember that the command works on the active environment (CAD or CAM) every time.

To start the procedure, hit



It opens the export options window.

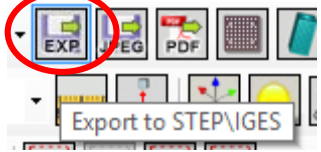


- STEP: creates the STEP file
- IGES: creates the IGES file
- FLAT: places all geometry as single component, without creating assembly structure
- THREE TOOL EDGES: keeps the faces of the tools' pin separated. Unthincking it will merge them (depending on CAD preferences)
- LAST TOOL ONLY: export the tool of the component placed at the top of the stack
- TABLES: includes base gauge's geometry
- CUBES/PALLET: includes pallet, shoulders and cubes geometry
- BREYLS: includes breyls' geometry. Activates also SUPPORT and TOOLS options.
  - SUPPORT: includes supports' geometry
  - TOOLS: includes supports' geometry

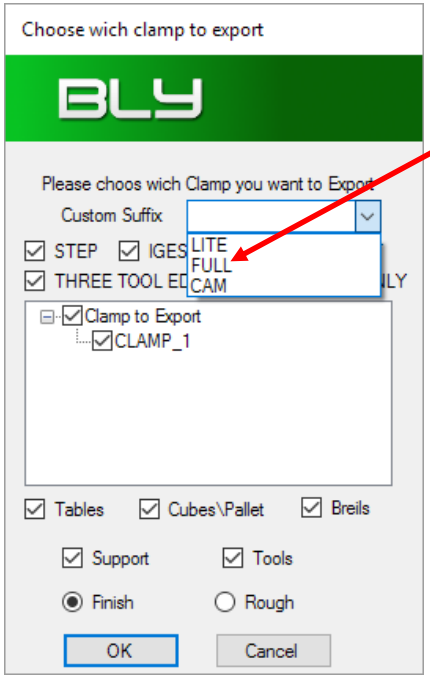
❑ *Export the clamping definition to CAD (cont'd)*

Once the clamping has been defined, the geometry of the FCS' components and the placement seats can be exported to a STEP or IGES file, which can then be used as exchange file for a 3D CAD system.

To start the procedure, hit



It opens the export options window



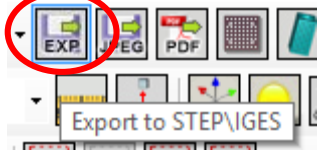
On the drop-down menu:

- LITE, FULL, CAM: export geometry contained in the library, if existing (see page 87).

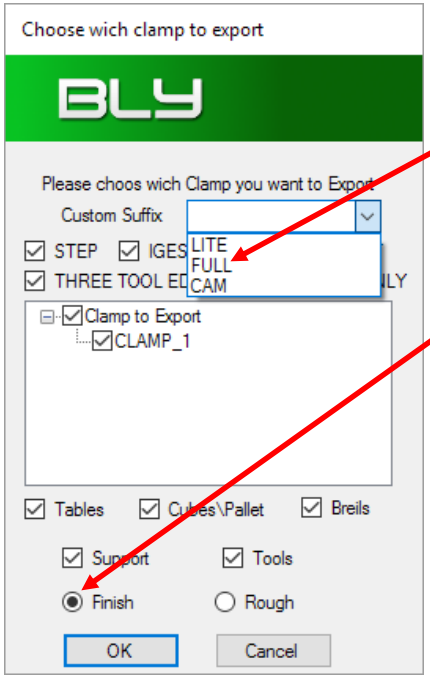
❑ *Export the clamping definition to CAD (cont'd)*

Once the clamping has been defined, the geometry of the FCS' components and the placement seats can be exported to a STEP or IGES file, which can then be used as exchange file for a 3D CAD system.

To start the procedure, hit



It opens the export options window



On the drop-down menu:

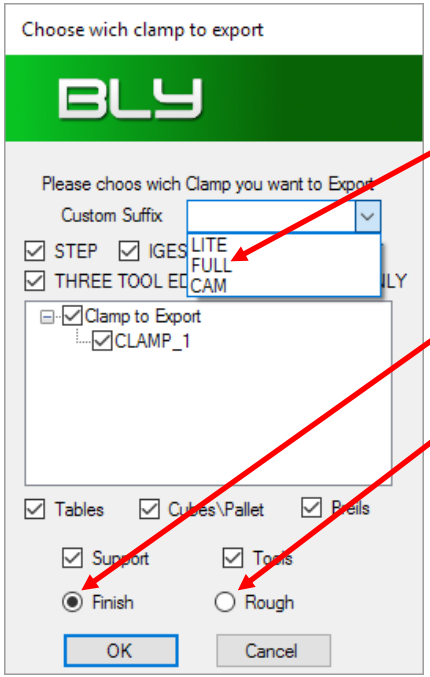
- LITE, FULL, CAM: export geometry contained in the library, if existing (see page 87).
- FINISH: export a larger tool, in order to create the seats for the finish milling

❑ *Export the clamping definition to CAD (cont'd)*

Once the clamping has been defined, the geometry of the FCS' components and the placement seats can be exported to a STEP or IGES file, which can then be used as exchange file for a 3D CAD system.

To start the procedure, hit

It opens the export options window

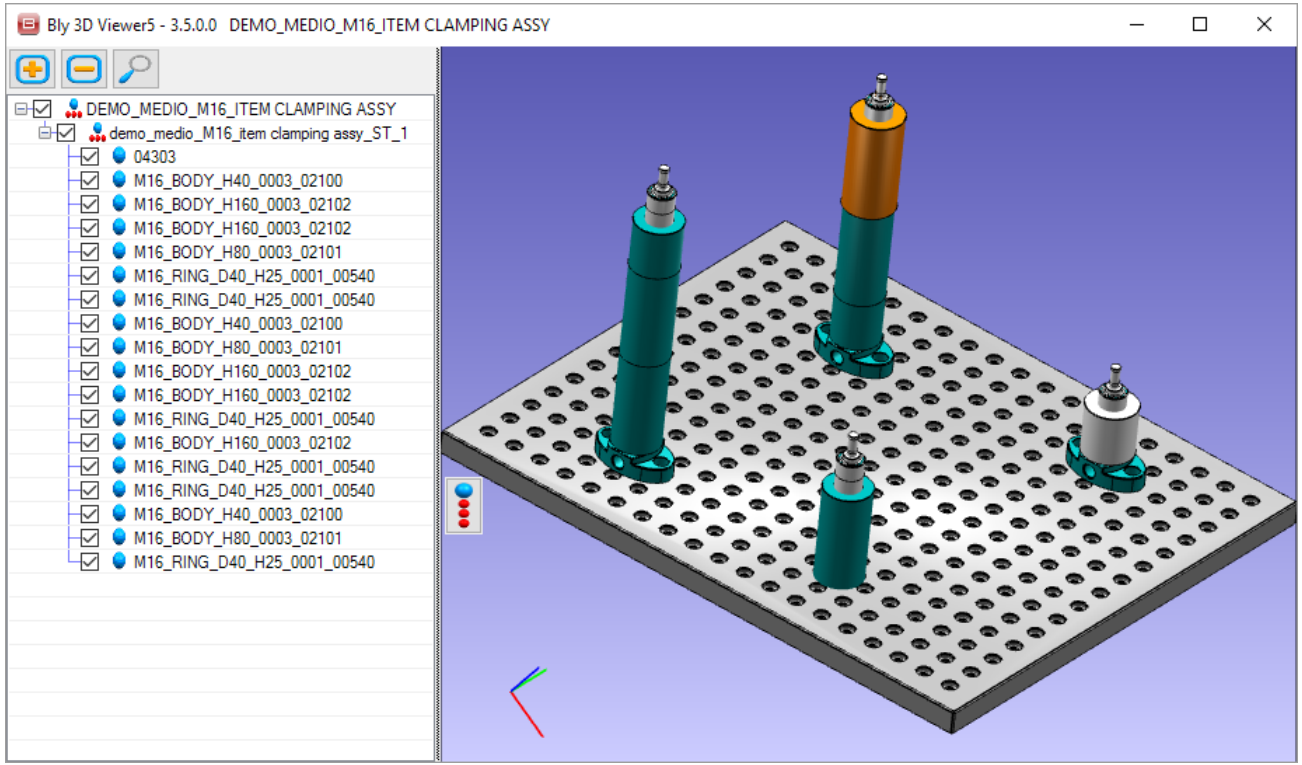


On the drop-down menu:

- LITE, FULL, CAM: export geometry contained in the library, if existing (see page 86)
- FINISH: export a larger tool, in order to create the seats for the finish milling
- ROUGH: exports a smaller tool, in order to create the seats for the rough milling

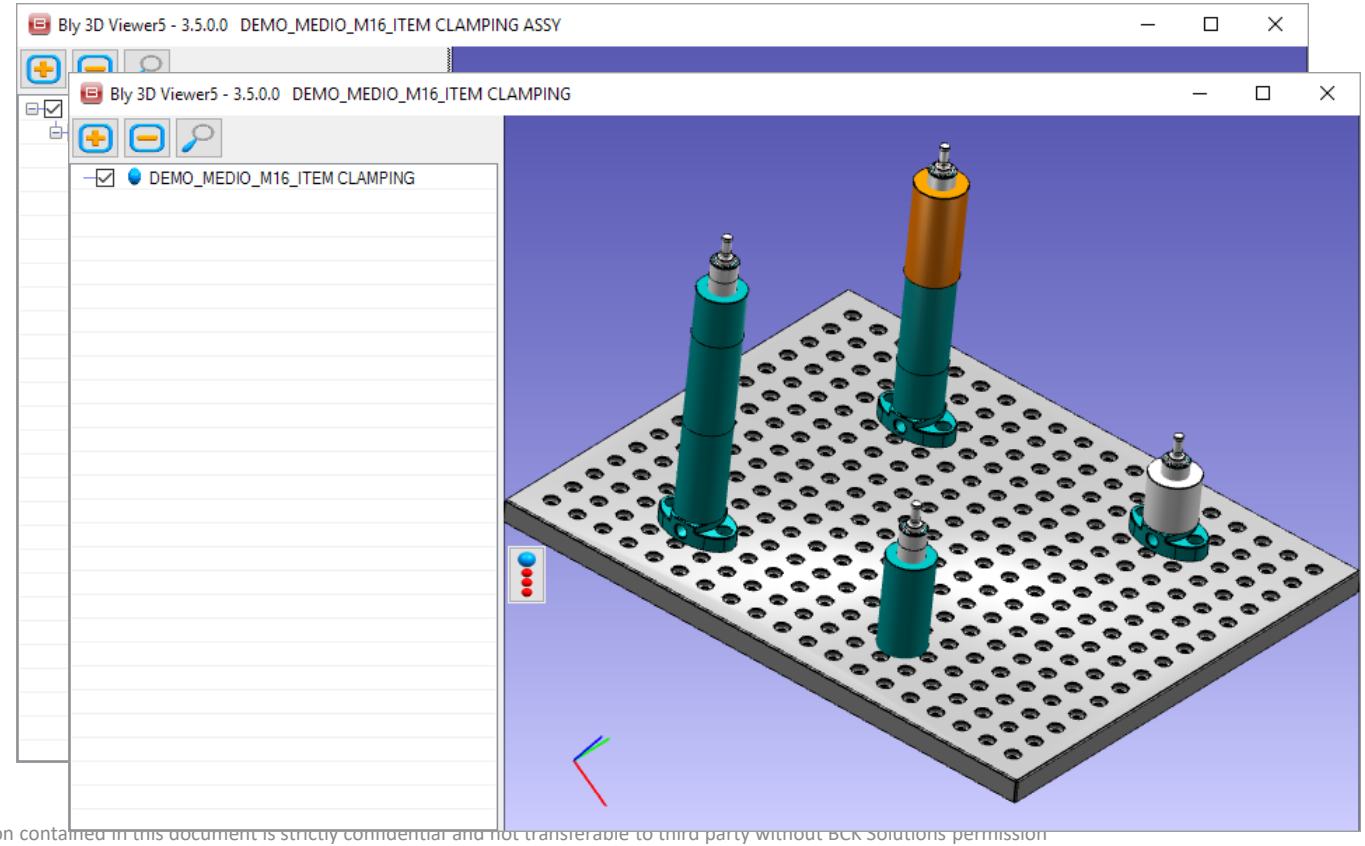
❑ *Export the clamping definition to CAD (cont'd)*

Example of exported assembly



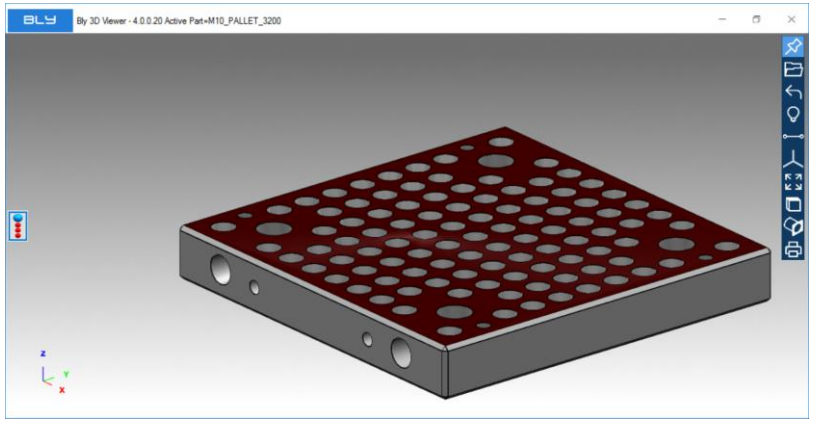
❑ *Export the clamping definition to CAD (cont'd)*

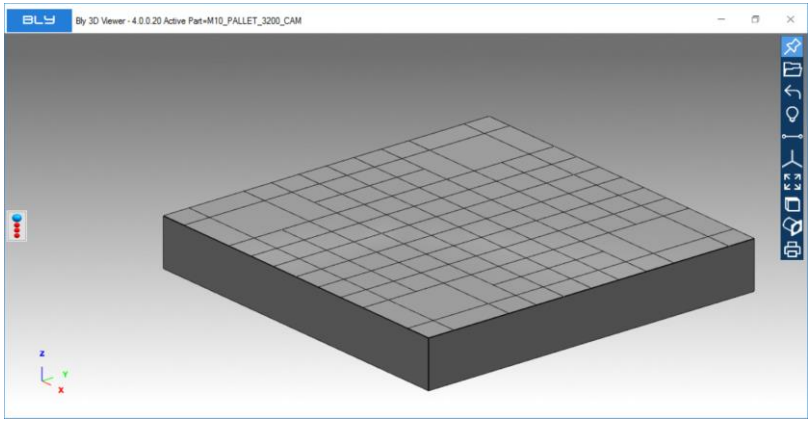
Example of exported FLAT



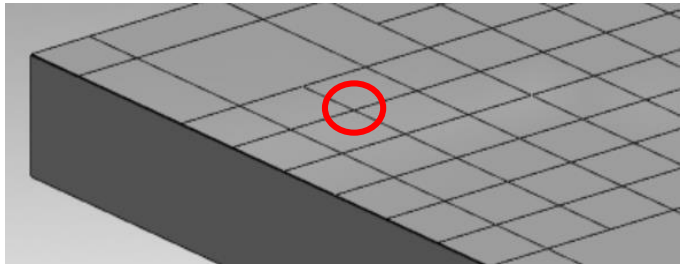
❑ *Export the clamping definition to CAD – Simplified libraries*

Guideline includes in its library simplified models of components library, particularly light, with graphical representation of the seats. Those files are named like the detailed part plus a «_CAM» suffix



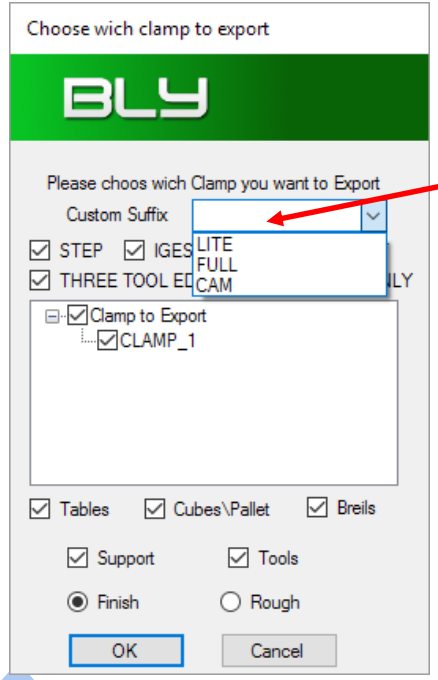


The pictures aboves show you an example of the detailed model (left) compared with the simplified pallet M10 (right). The grid of placement holes is represented by a net of lines, crossing in correspondance of holes' centers (see red circle in the picture on the right): this model has the advantage of being ready for CAMpath design, without any preparation work by the user to close the holes with surfaces.



❑ *Export the clamping definition to CAD – Simplified libraries*

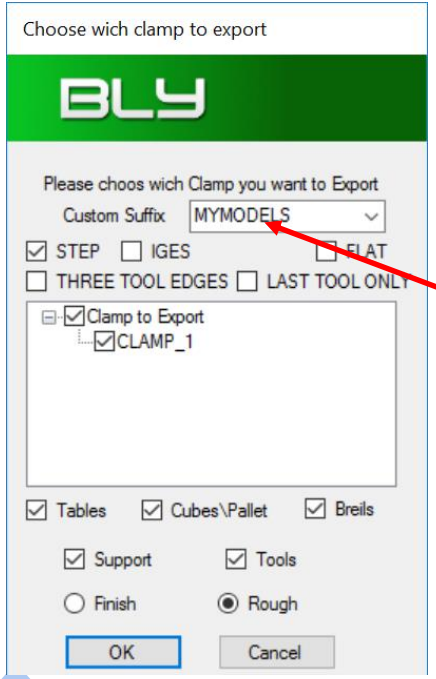
It is possible to create as many different versions of library as desired, to be used in different situation (i.e.: simplified models to be imported in CAD to increase translation performance, CAM design ect.)



Guideline has three pre-set options in the export menu (LITE, FULL, CAM) and the simplified models for CAM option: it is possible to create the models for the other 2 options simply generating the step files with the desired geometry and name them with the following convention:
[detailedmodelname]_LITE.stp (or _FULL or _CAM).
Guideline will then seek for the selected options' models and use them, if present, otherwise it will use the detailed ones.

❑ *Export the clamping definition to CAD – Simplified libraries*

It is possible to create as many different versions of library as desired, to be used in different situation (i.e.: simplified models to be imported in CAD to increase translation performance, CAM design ect.)



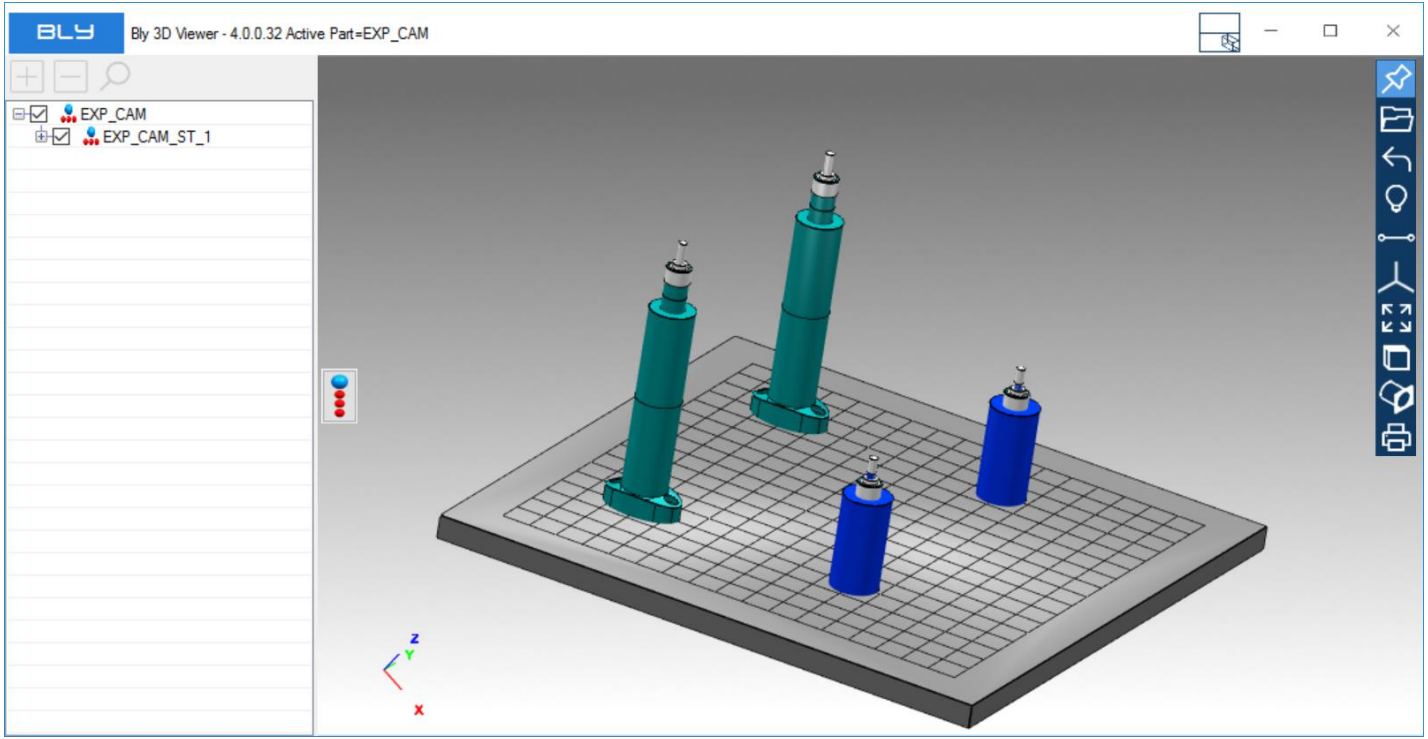
Guideline has three pre-set options in the export menu (LITE, FULL, CAM) and the simplified models for CAM option: it is possible to create the models for the other 2 options simply generating the step files with the desired geometry and name them with the following convention:
[detailedmodelname]_LITE.stp (or _FULL or _CAM).
Guideline will then seek for the selected options' models and use them, if present, otherwise it will use the detailed ones.

It is also possible to use a different option «on fly»: you can create components named «[detailedmodelname]_[optionname].stp», place them in the the library and type [optionname] in the Custom Suffix field in the export window.

In the example in the picture, [optionname]=MYMODELS

❑ *Export the clamping definition to CAD – Simplified libraries*

Example of simplified geometry exported using «CAM» library



PHASE 2: CLAMPING DESIGN

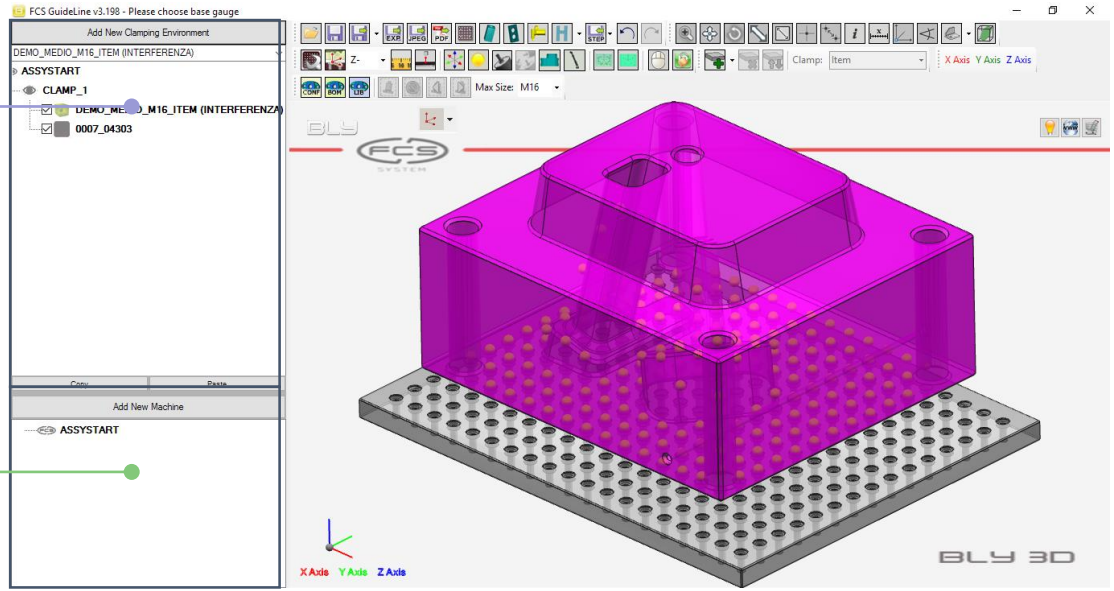
The two working environments:

- *Clamping environment (CAD)*

  Target:
  make the clamping holes on the workpiece. A first clamping, named by default «CLAMP_1», is created as soon as the base gauge is selected and made working environment.

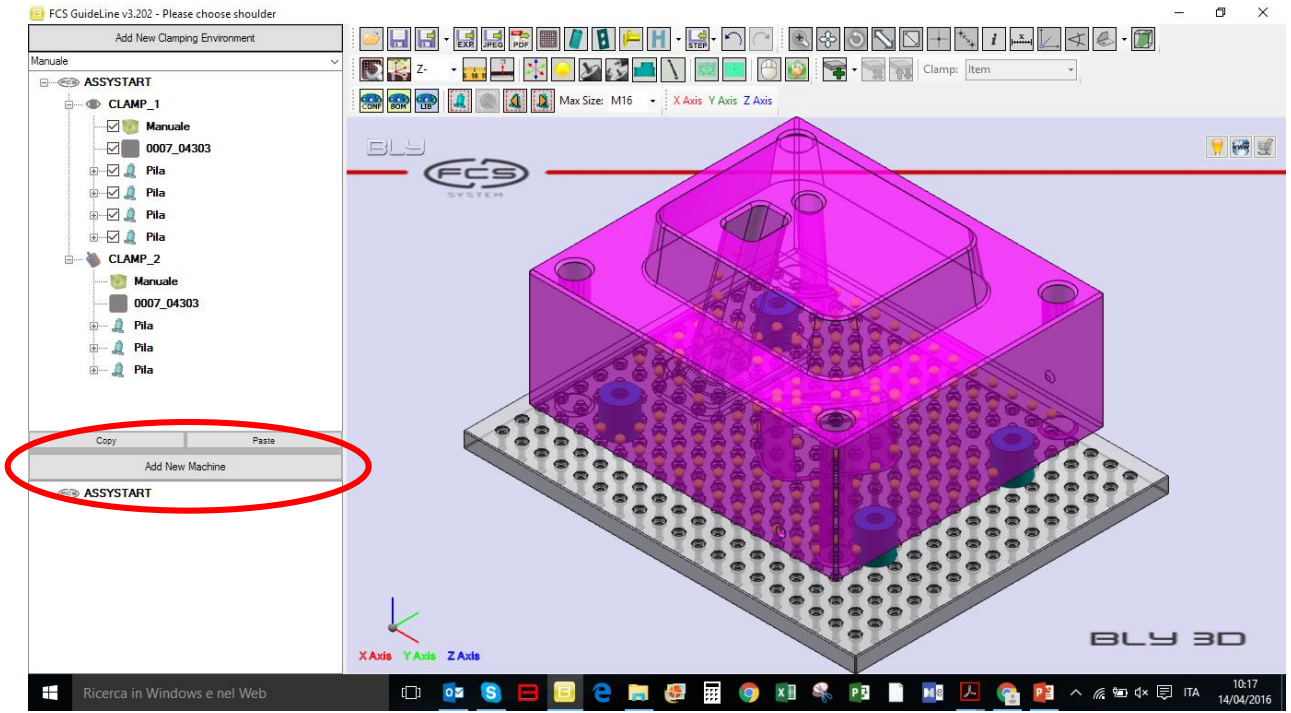
- *Machining environment (CAM)*

  Target:
  make the complete clamping, starting from the clamping holes already defined in the previous environment (or in any case already included in the workpiece geometry).



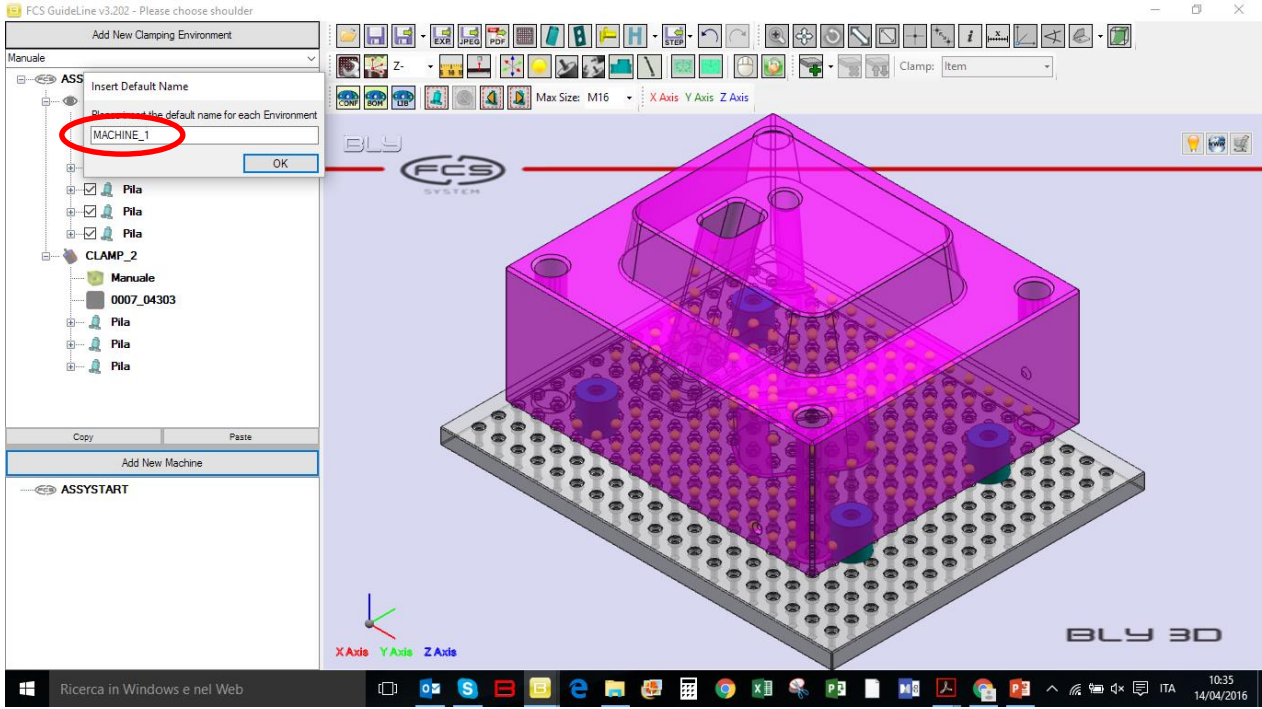
❑ *Creating the machining (CAM) environment*

Guideline can use, as starting point for the CAM environment, workpieces whose placement seats have been defined in CAD environment or, alternatively, imported 3d models having placement seats already created in a 3D CAD system: in this last case, the software recognizes them automatically, based on their geometry.
To create the environment, press «Add New Machine» button in the CAM window.



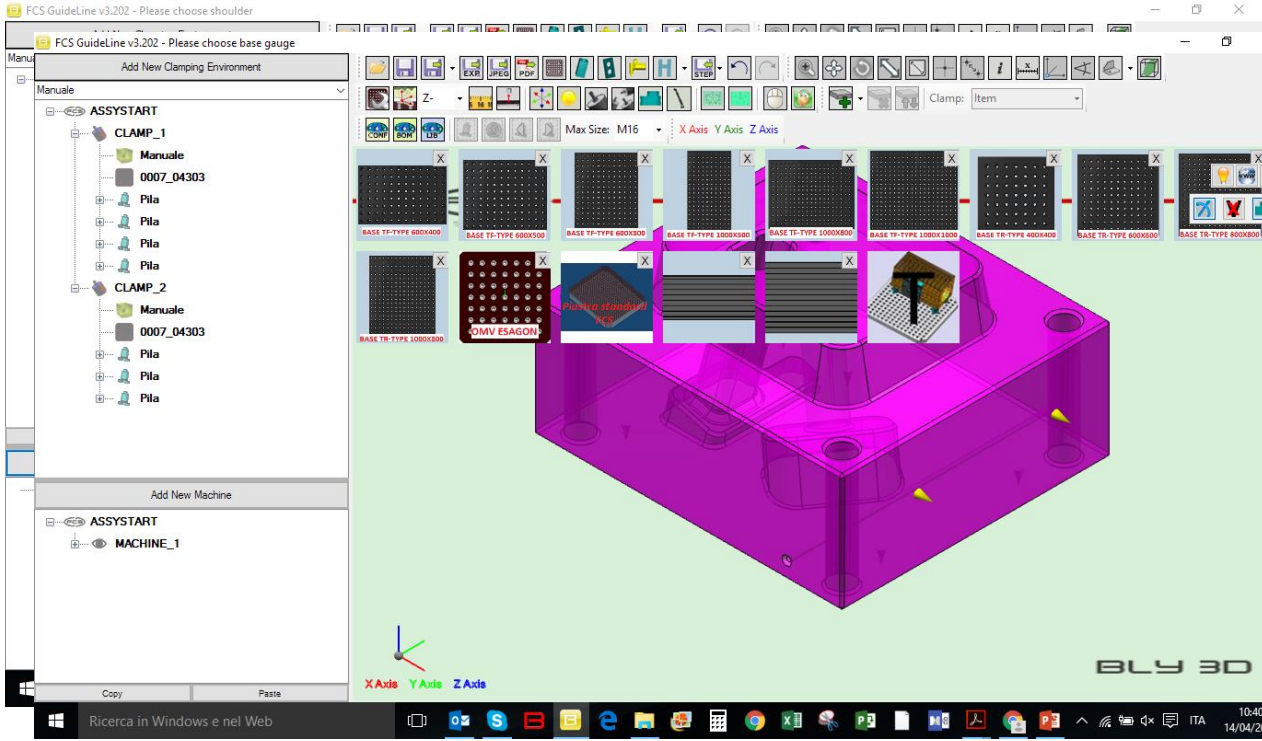
❑ *Creating the machining (CAM) environment (cont'd)*

Type the name of the Environment in the displayed window and hit «OK».



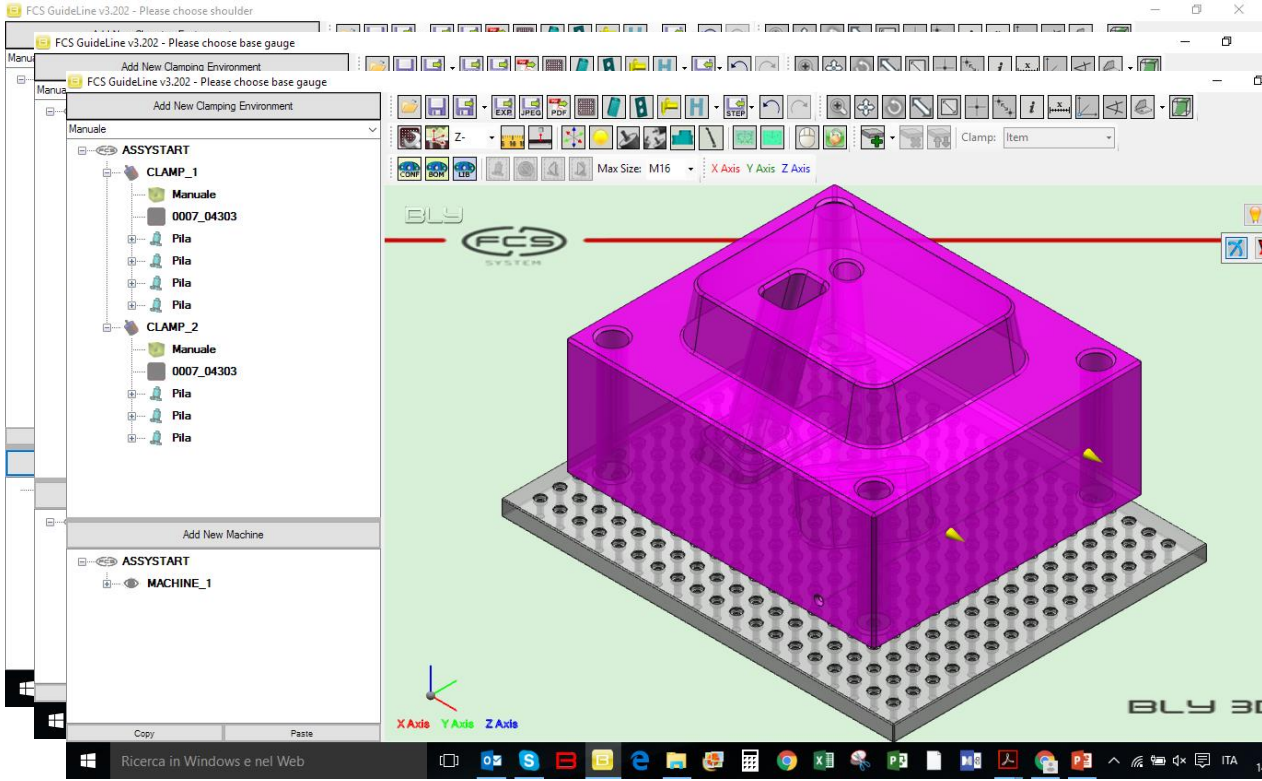
❑ *Creating the machining (CAM) environment (cont'd)*

Type the name of the Environment in the displayed window and hit «OK».
Select the base gauge. Please, note that it can be different from the one used in CAD environment, eventually.
Also, the background changes to light green, in order to help you identify quickly the environment you are operating.



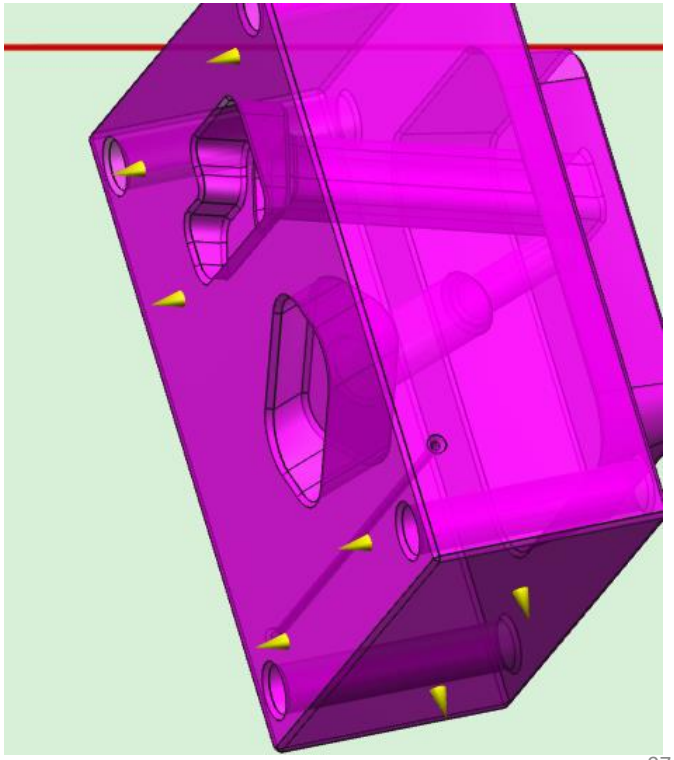
❑ *Creating the machining (CAM) environment (cont'd)*

Type the name of the Environment in the displayed window and hit «OK».
Select the base gauge. Please, note that it can be different from the one used in CAD environment, eventually.
Also, the background changes to light green, in order to help you identify quickly the environment you are operating.
After the selection, the base gauge is placed using the origins, as in CAD environment.



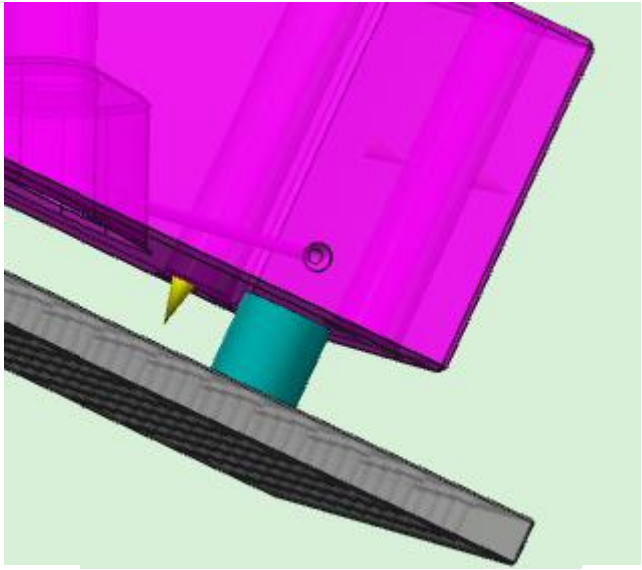
❑ *Orient the workpiece*

As opposite as CAD environment, CAM does not give you the possibility to freely orient the workpiece, but forces you to select the orientations defined by the placement seats already present in the workpiece and indicated by the small yellow cones (see picture).
Those cones are the acceptable positions for the placement seats and, consequentely, for the stacks of FCS Systems. Any other placement is not acceptable, since has not been evaluated and approved.
The reason is, by definition, because CAM environment is made to place the workpiece, while CAD is the environment where the designers study the placement seats creation and evaluate possible modification to the piece, if needed.
Splitting the two environments gives the flexibility to capture inputs and knowledge from both designers and machine operators, without interfering one another.



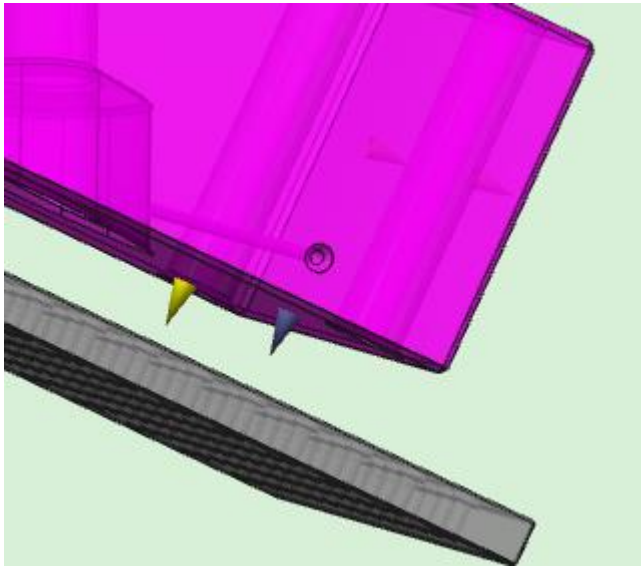
❑ *Place the workpiece*

To place the workpiece, select the cone you need and Guideline will place the default stack.



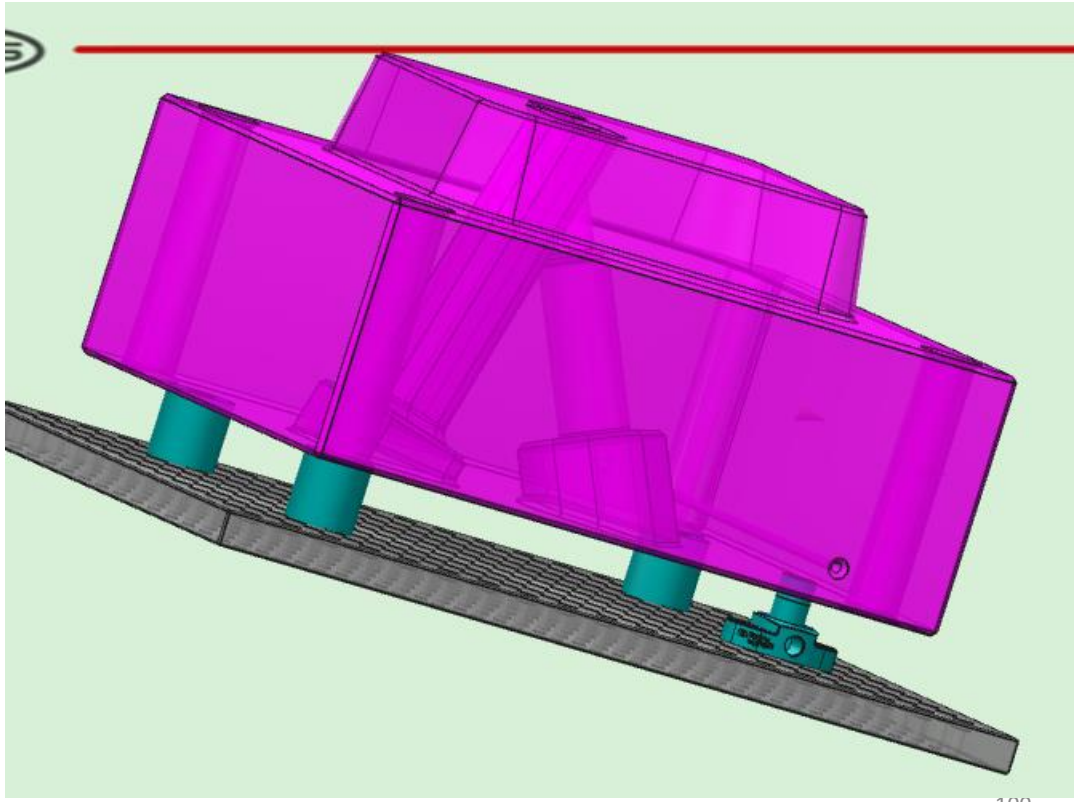
❑ *Place the workpiece (cont'd)*

The software is able to recognize automatically all the placement cones with the same orientation, therefore it can place all the related stacks at once: to do so, click on one of the cones while pressing SHIFT on the keyboard.



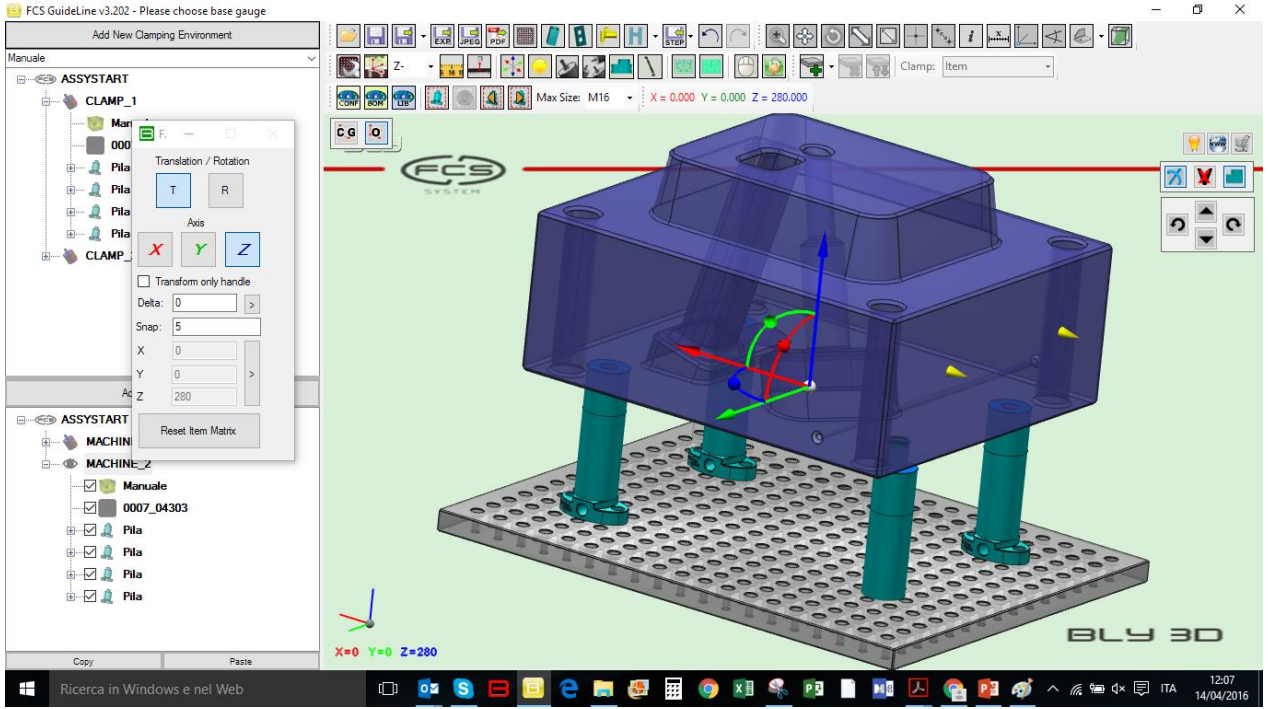
❑ *Place the workpiece (cont'd)*

The software is able to recognize automatically all the placement cones with the same orientation, therefore it can place all the related stacks at once: to do so, click on one of the cones while pressing SHIFT on the keyboard. As you notice on the picture, Guideline calculates automatically the stacks based on the depth of the placement holes on the workpiece, avoiding impossible combinations



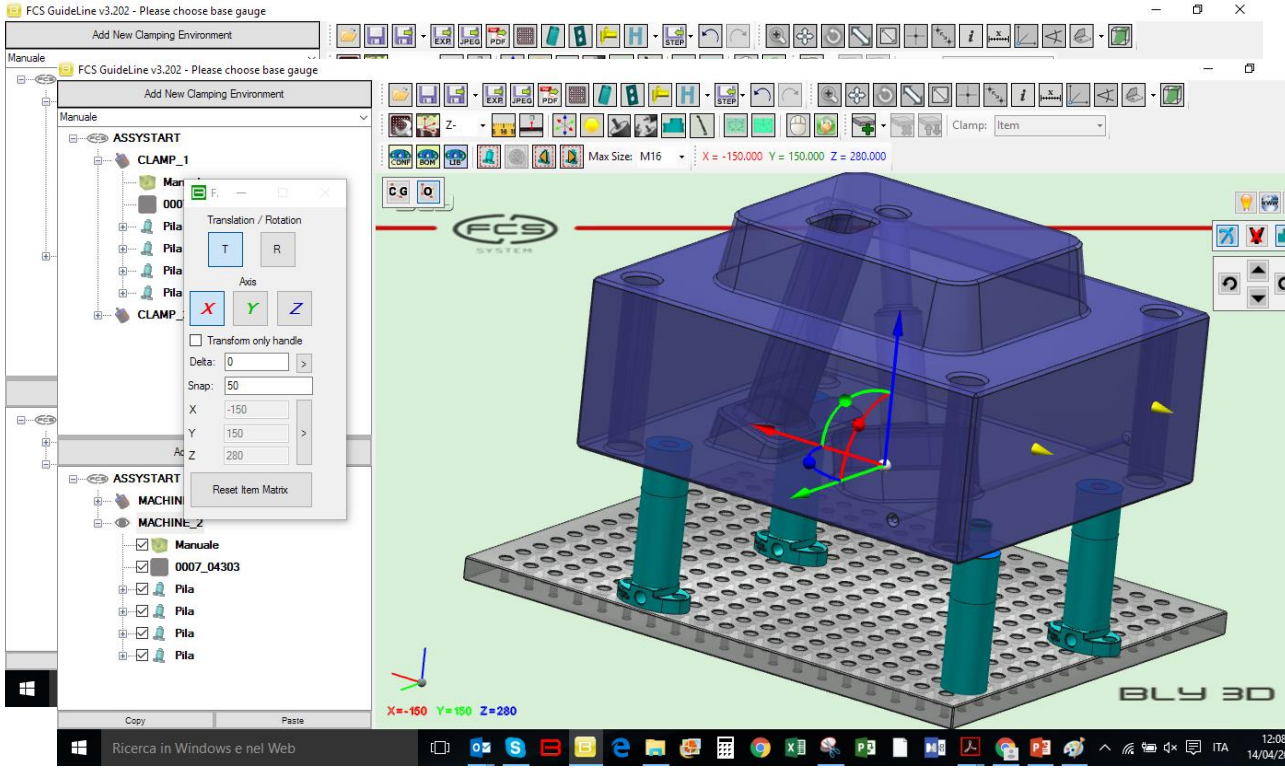
❑ *Define the stacks*

To define the stacks, first action is normally to decide the height (distance from the base gauge) of the workpiece: Guideline will the propose a stack based on the table settings.



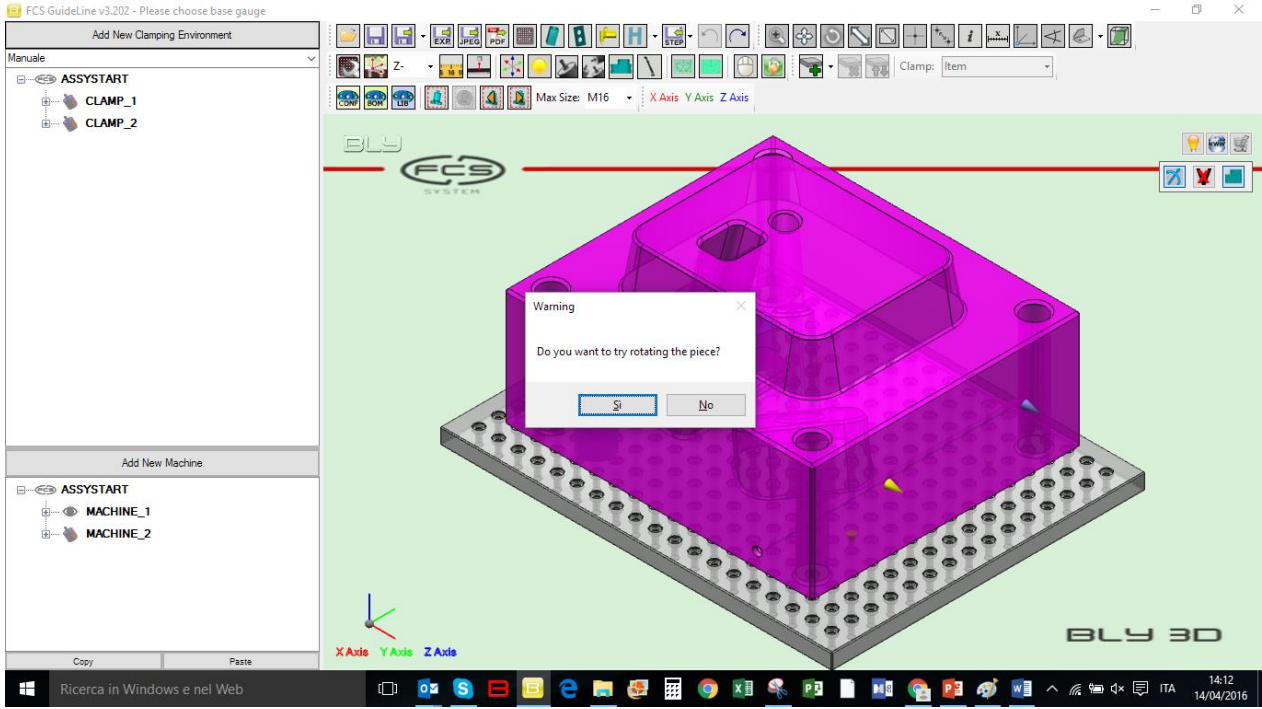
❑ *Define the stacks (cont'd)*

To define the stacks, first action is normally to decide the height (distance from the base gauge) of the workpiece: Guideline will the propose a stack based on the table settings.
N.B: in spite of the display, in CAM is possible to drag the piece along the cones direction only.
However, transalation in the other directions is allowed and will result in translating the piece and the stacks together on the base gauge.



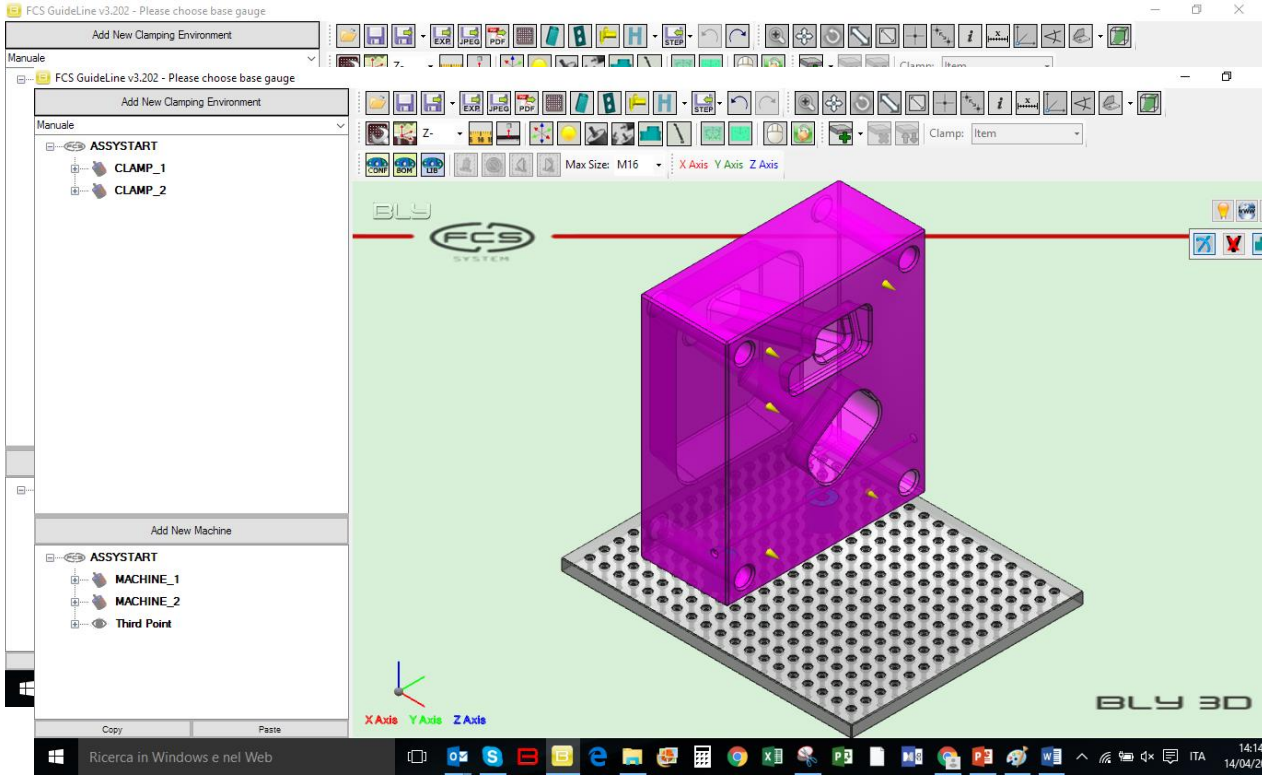
❑ *Third Point Clamp*

Third Point Clamp is managed in CAM environment too: similarly to the «normal» clamp, placements seats must be defined in advance and be compatible with the hole on the gauge/pallet to clamp the piece. It is not mandatory to orient the piece before selecting the cones: Guideline will check the situation and ask you if the piece must be rotated, eventually.



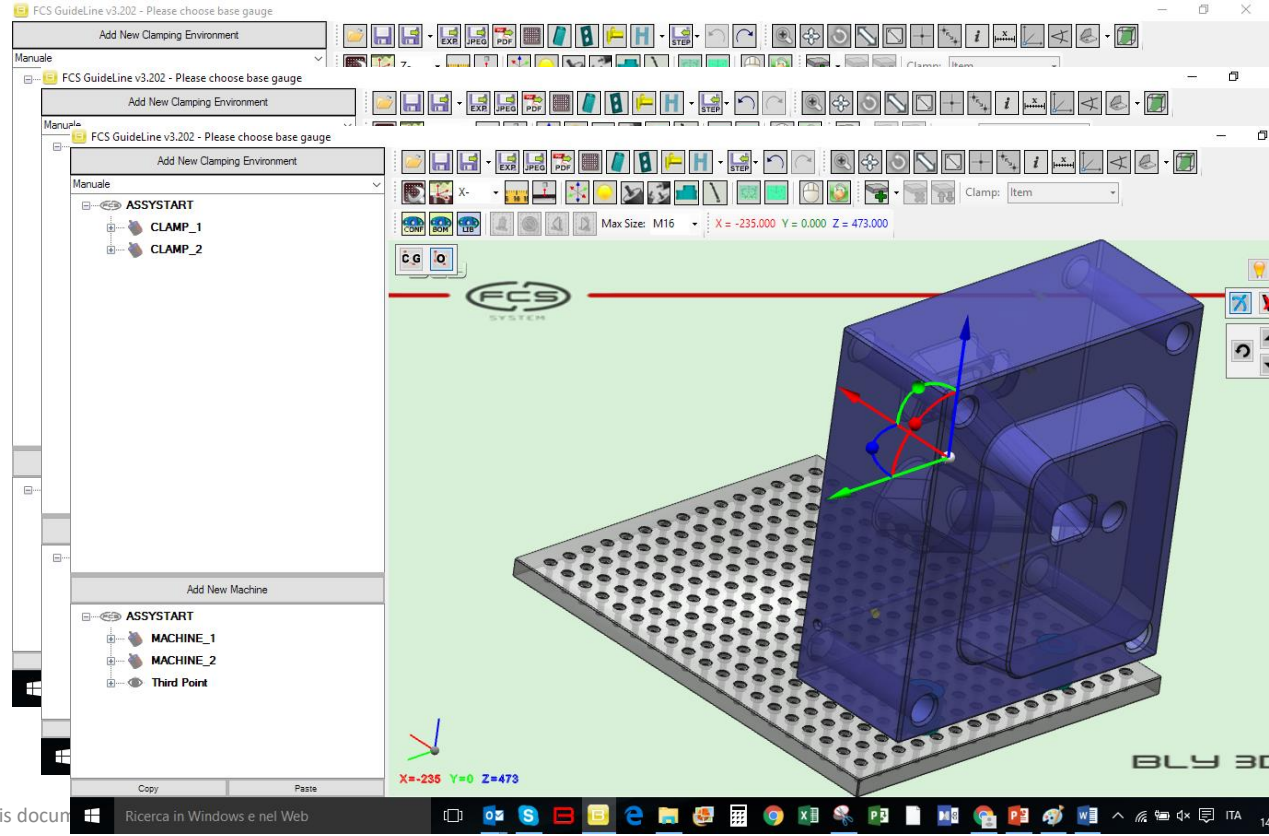
❑ *Third Point Clamp (cont'd)*

Third Point Clamp is managed in CAM environment too: similarly to the «normal» clamp, placements seats must be defined in advance and be compatible with the hole on the gauge/pallet to clamp the piece. It is not mandatory to orient the piece before selecting the cones: Guideline will check the situation and ask you if the piece must be rotated, eventually.



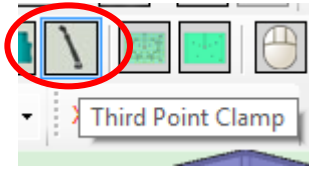
❑ *Third Point Clamp (cont'd)*

Third Point Clamp is managed in CAM environment too: similarly to the «normal» clamp, placements seats must be defined in advance and be compatible with the hole on the gauge/pallet to clamp the piece.
It is not mandatory to orient the piece before selecting the cones: Guideline will check the situation and ask you if the piece must be rotated, eventually.
After the first placement, the piece can be moved as usual.

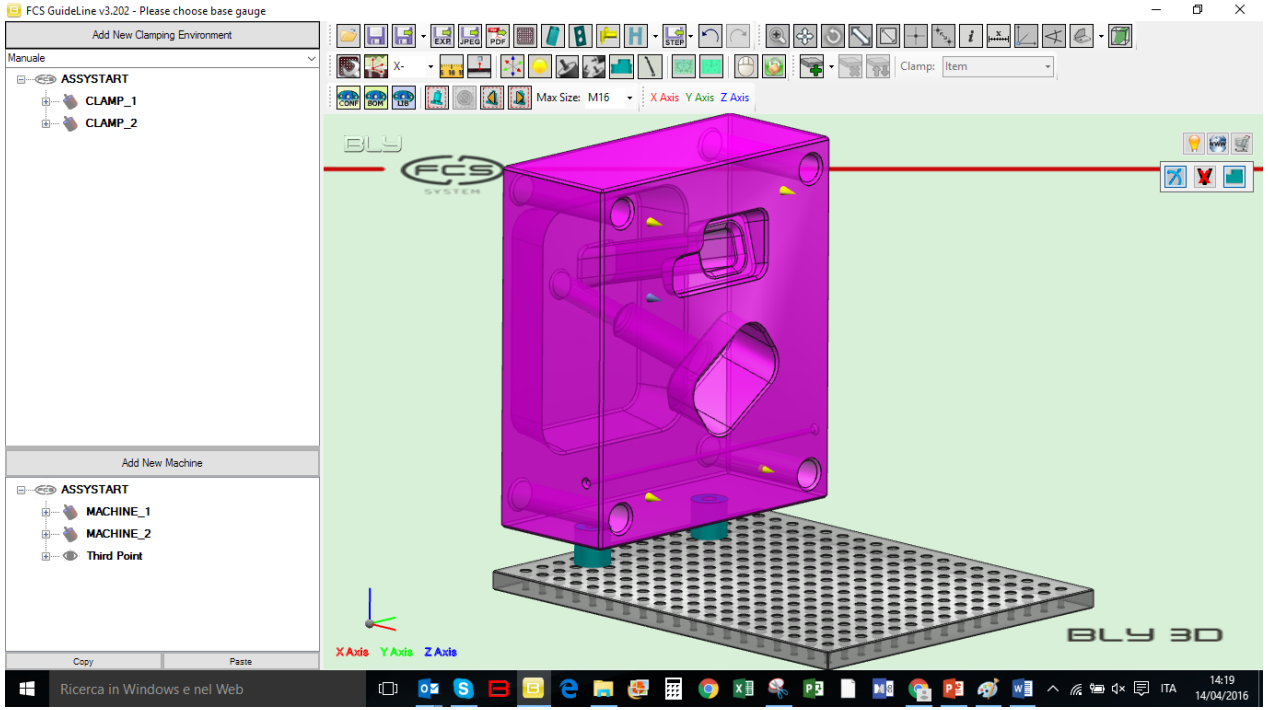


❑ *Third Point Clamp (cont'd)*

To activate the Third Point placement, press

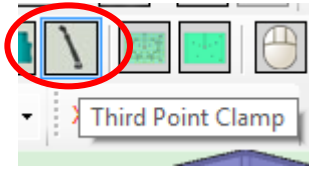


on the toolbar: since the position cannot be defined freely, you can select the placement cone only.

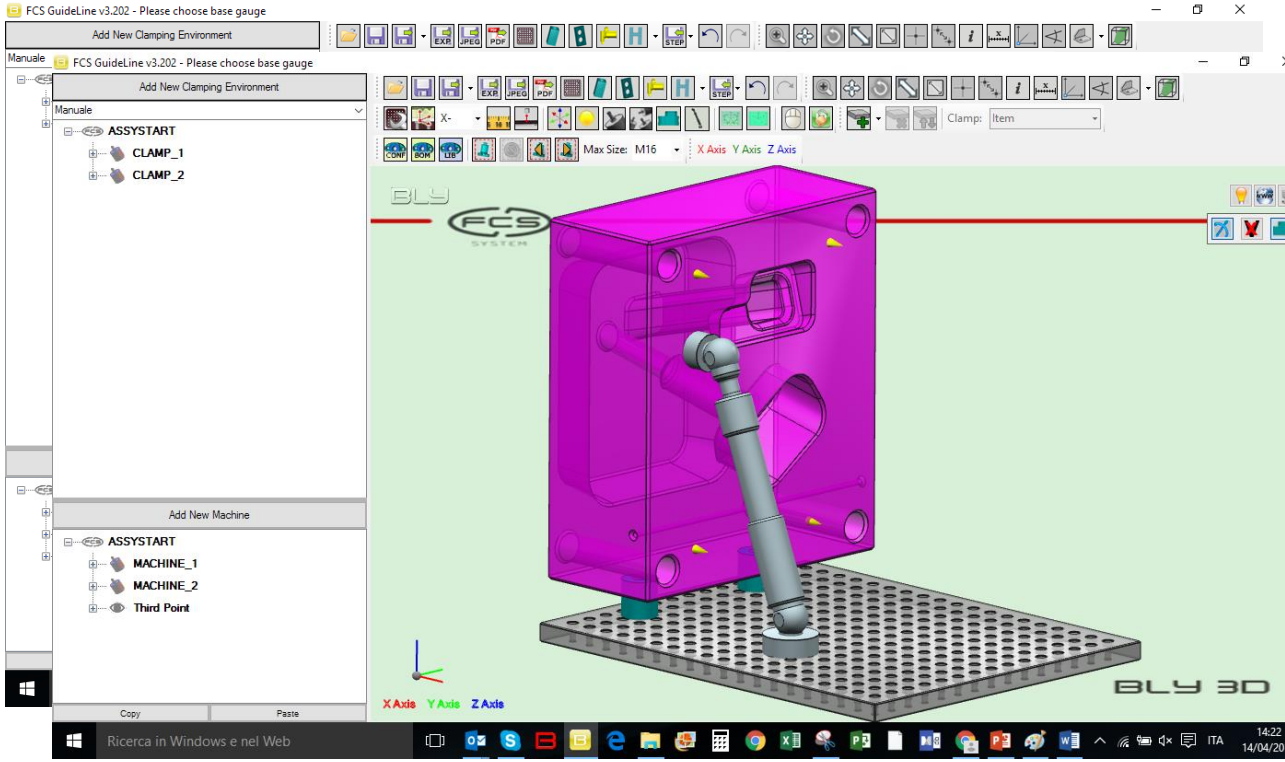


❑ *Third Point Clamp (cont'd)*

To activate the Third Point placement, press

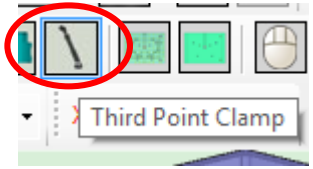


on the toolbar: since the position cannot be defined freely, you can select the placement cone only. After that, Guideline will place the arm in a proposed position.



❑ *Third Point Clamp (cont'd)*

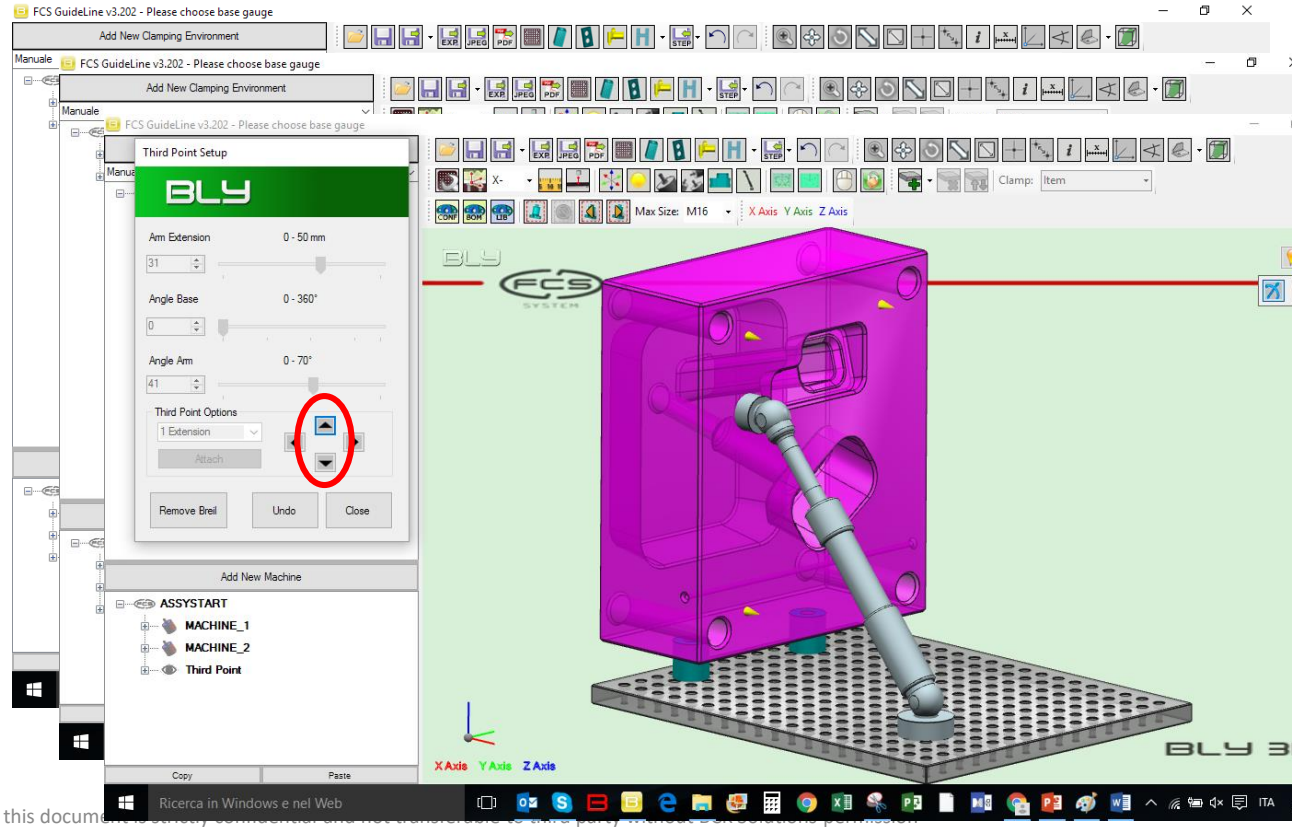
To activate the Third Point placement, press



on the toolbar: since the position cannot be defined freely, you can select the placement cone only. After that, Guideline will place the arm in a proposed position.

You can modify the attachment hole on the base gauge by clicking the arm itself and using the vertical arrows on the placement window: the software will calculate the parameters as required by the position

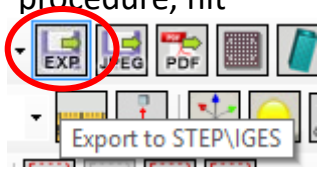
The button Remove Breil will delete the arm.



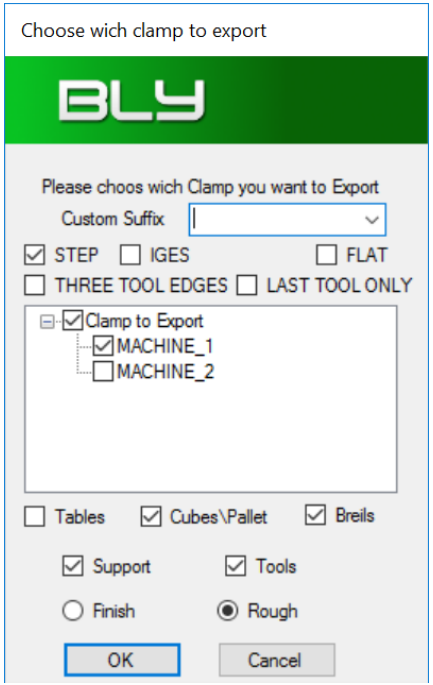
❑ *Export the clamping definition to CAM*

Once the clamping has been defined, it is possible to export the geometry of the FCS' components and the placement seats to a STEP or IGES file, which can then be used as exchange file to a 3D CAD system.
It is important to remember that the command works on the active environment (CAD or CAM) every time.

To start the procedure, hit



It opens the export options window.



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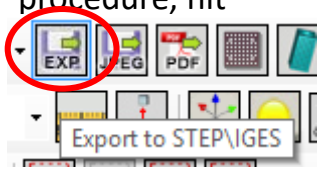
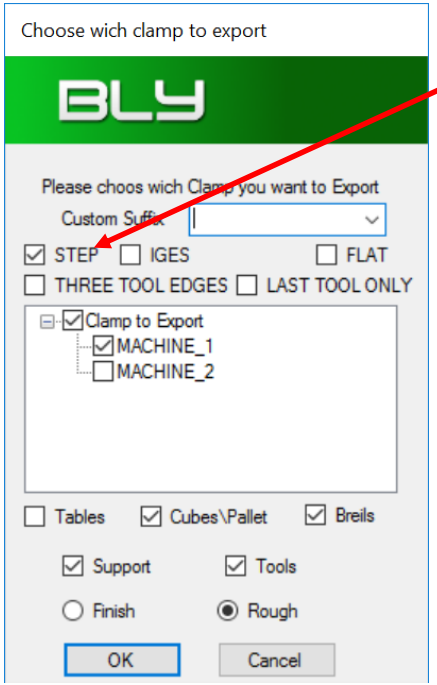


It opens the export options window.



- STEP: creates the STEP file

❑ *Export the clamping definition to CAM*

Once the clamping has been defined, it is possible to export the geometry of the FCS' components and the placement seats to a STEP or IGES file, which can then be used as exchange file to a 3D CAD system.
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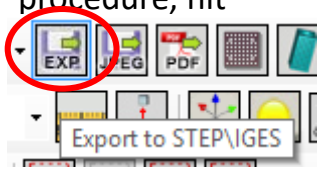
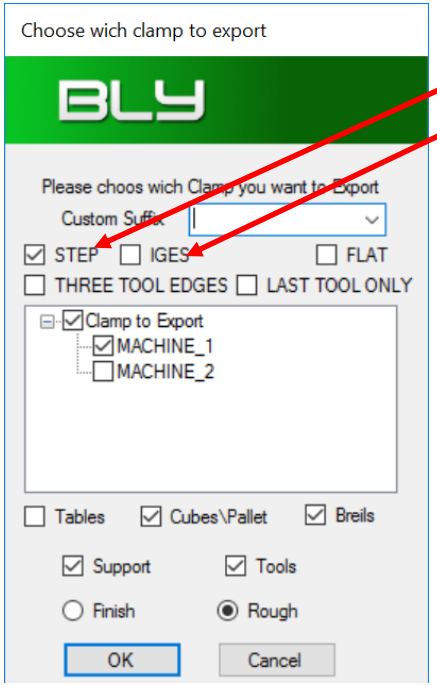


It opens the export options window.

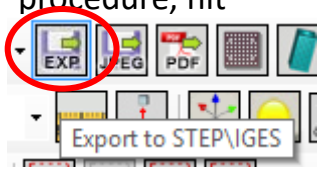


- STEP: creates the STEP file
- IGES: creates the IGES file

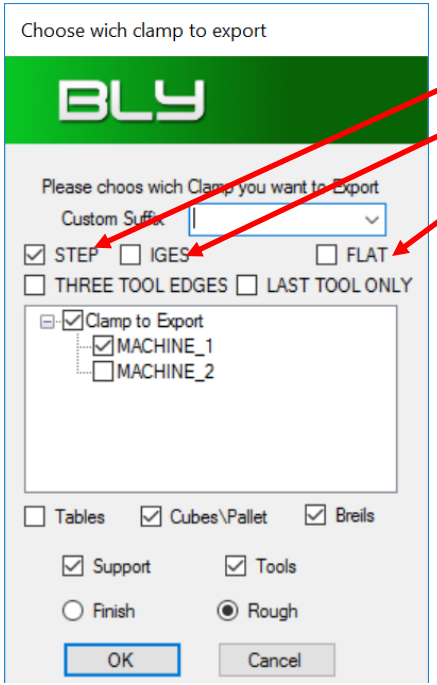
❑ *Export the clamping definition to CAM*

Once the clamping has been defined, it is possible to export the geometry of the FCS' components and the placement seats to a STEP or IGES file, which can then be used as exchange file to a 3D CAD system.
It is important to remember that the command works on the active environment (CAD or CAM) every time.

To start the procedure, hit



It opens the export options window.

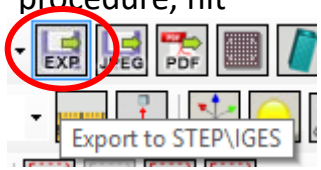


- STEP: creates the STEP file
- IGES: creates the IGES file
- FLAT: places all geometry as single component, without creating assembly structure

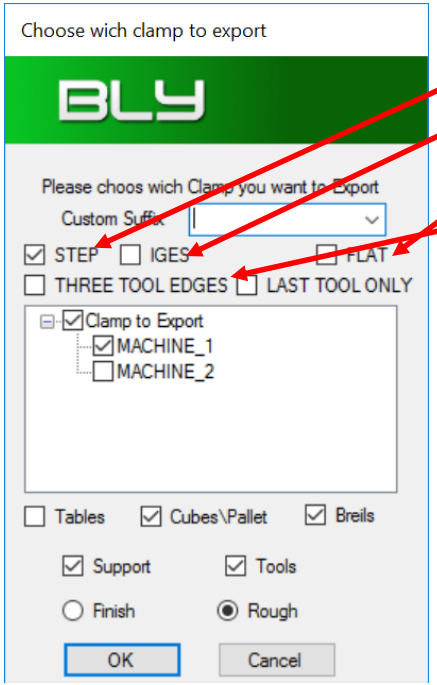
❑ *Export the clamping definition to CAM*

Once the clamping has been defined, it is possible to export the geometry of the FCS' components and the placement seats to a STEP or IGES file, which can then be used as exchange file to a 3D CAD system.
It is important to remember that the command works on the active environment (CAD or CAM) every time.

To start the procedure, hit



It opens the export options window.



- STEP: creates the STEP file
- IGES: creates the IGES file
- FLAT: places all geometry as single component, without creating assembly structure
- THREE TOOL EDGES: keeps the faces of the tools' pin separated. Unthincking it will merge them (depending on CAD preferences)

❑ *Export the clamping definition to CAM*

Once the clamping has been defined, it is possible to export the geometry of the FCS' components and the placement seats to a STEP or IGES file, which can then be used as exchange file to a 3D CAD system.
It is important to remember that the command works on the active environment (CAD or CAM) every time.

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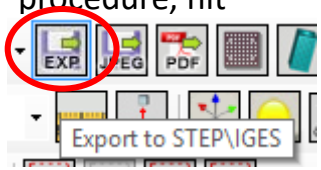
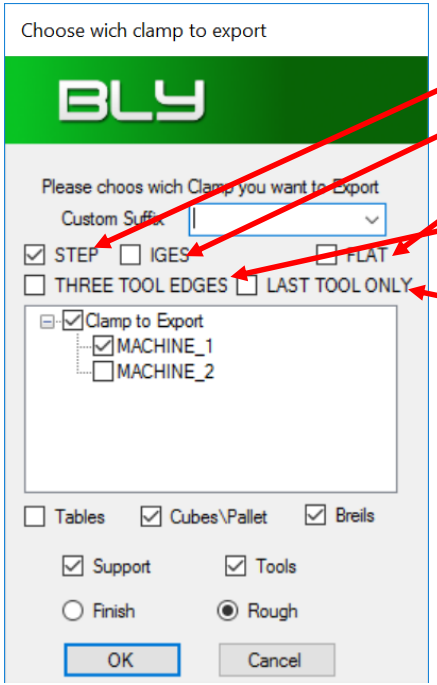


It opens the export options window.

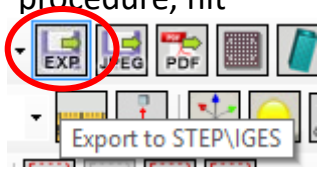


- STEP: creates the STEP file
- IGES: creates the IGES file
- FLAT: places all geometry as single component, without creating assembly structure
- THREE TOOL EDGES: keeps the faces of the tools' pin separated. Unthincking it will merge them (depending on CAD preferences)
- LAST TOOL ONLY: export the tool of the component placed at the top of the stack

❑ *Export the clamping definition to CAM*

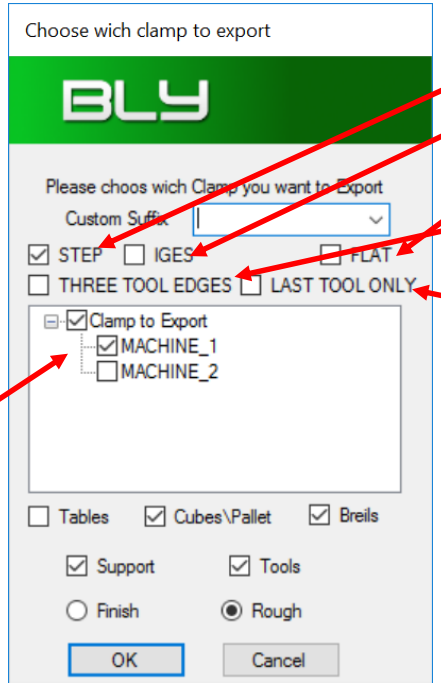
Once the clamping has been defined, it is possible to export the geometry of the FCS' components and the placement seats to a STEP or IGES file, which can then be used as exchange file to a 3D CAD system.
It is important to remember that the command works on the active environment (CAD or CAM) every time.

To start the procedure, hit



It opens the export options window.

Selectable Machines to export

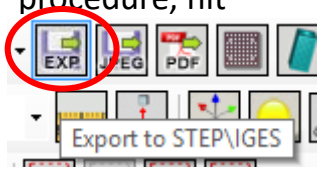


- STEP: creates the STEP file
- IGES: creates the IGES file
- FLAT: places all geometry as single component, without creating assembly structure
- THREE TOOL EDGES: keeps the faces of the tools' pin separated. Unthincking it will merge them (depending on CAD preferences)
- LAST TOOL ONLY: export the tool of the component placed at the top of the stack

❑ *Export the clamping definition to CAM*

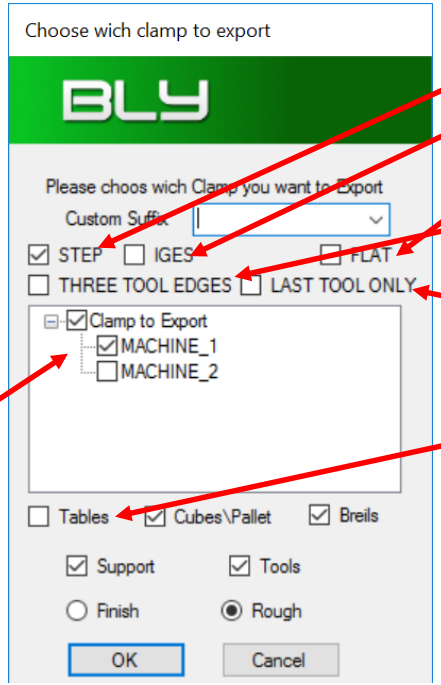
Once the clamping has been defined, it is possible to export the geometry of the FCS' components and the placement seats to a STEP or IGES file, which can then be used as exchange file to a 3D CAD system.
It is important to remember that the command works on the active environment (CAD or CAM) every time.

To start the procedure, hit



It opens the export options window.

Selectable Machines to export

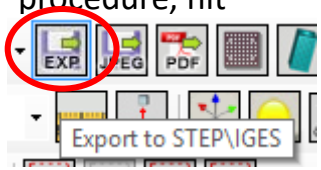


- STEP: creates the STEP file
- IGES: creates the IGES file
- FLAT: places all geometry as single component, without creating assembly structure
- THREE TOOL EDGES: keeps the faces of the tools' pin separated. Unthincking it will merge them (depending on CAD preferences)
- LAST TOOL ONLY: export the tool of the component placed at the top of the stack
- TABLES: includes base gauge's geometry

❑ *Export the clamping definition to CAM*

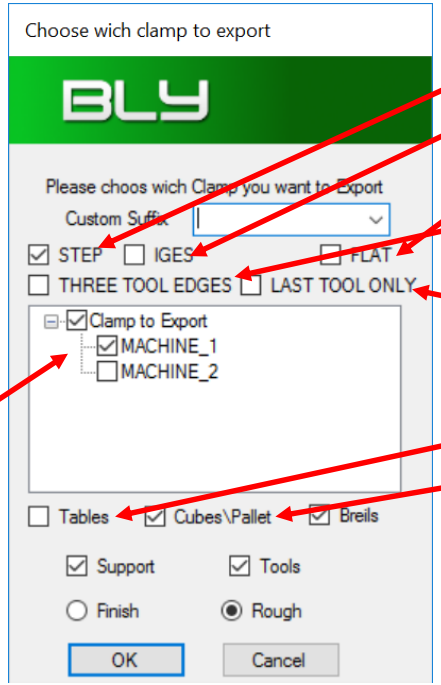
Once the clamping has been defined, it is possible to export the geometry of the FCS' components and the placement seats to a STEP or IGES file, which can then be used as exchange file to a 3D CAD system.
It is important to remember that the command works on the active environment (CAD or CAM) every time.

To start the procedure, hit



It opens the export options window.

Selectable Machines to export

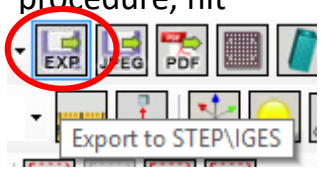


- STEP: creates the STEP file
- IGES: creates the IGES file
- FLAT: places all geometry as single component, without creating assembly structure
- THREE TOOL EDGES: keeps the faces of the tools' pin separated. Unthincking it will merge them (depending on CAD preferences)
- LAST TOOL ONLY: export the tool of the component placed at the top of the stack
- TABLES: includes base gauge's geometry
- CUBES/PALLET: includes pallet, shoulders and cubes geometry

❑ *Export the clamping definition to CAM*

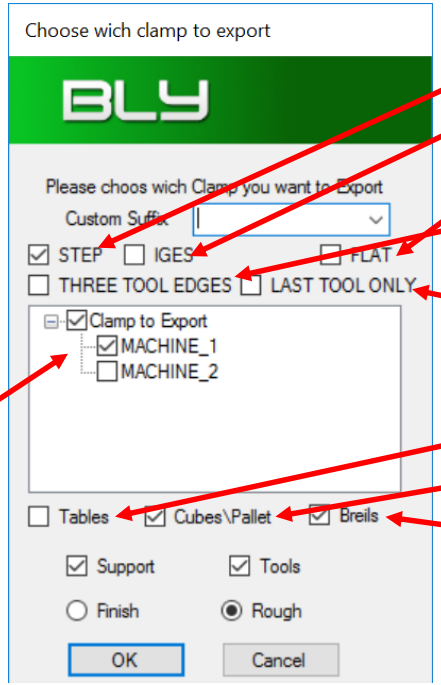
Once the clamping has been defined, it is possible to export the geometry of the FCS' components and the placement seats to a STEP or IGES file, which can then be used as exchange file to a 3D CAD system.
It is important to remember that the command works on the active environment (CAD or CAM) every time.

To start the procedure, hit



It opens the export options window.

Selectable Machines to export

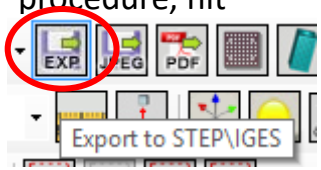


- STEP: creates the STEP file
- IGES: creates the IGES file
- FLAT: places all geometry as single component, without creating assembly structure
- THREE TOOL EDGES: keeps the faces of the tools' pin separated. Unthincking it will merge them (depending on CAD preferences)
- LAST TOOL ONLY: export the tool of the component placed at the top of the stack
- TABLES: includes base gauge's geometry
- CUBES/PALLET: includes pallet, shoulders and cubes geometry
- BREYLS: includes breyls' geometry. Activates also SUPPORT and TOOLS options.

❑ *Export the clamping definition to CAM*

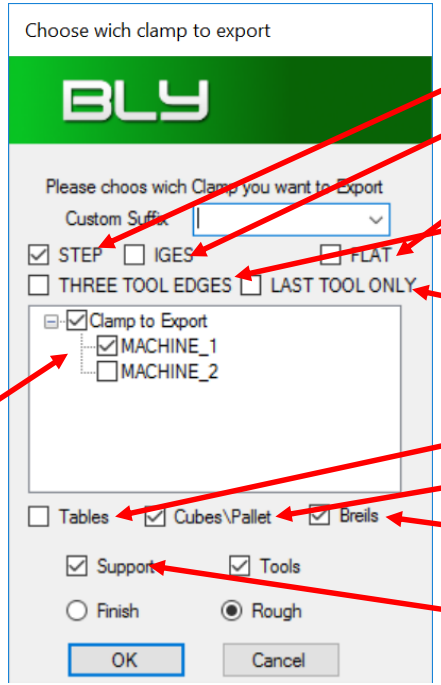
Once the clamping has been defined, it is possible to export the geometry of the FCS' components and the placement seats to a STEP or IGES file, which can then be used as exchange file to a 3D CAD system.
It is important to remember that the command works on the active environment (CAD or CAM) every time.

To start the procedure, hit



It opens the export options window.

Selectable Machines to export

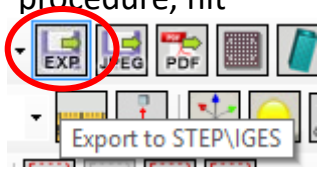


- STEP: creates the STEP file
- IGES: creates the IGES file
- FLAT: places all geometry as single component, without creating assembly structure
- THREE TOOL EDGES: keeps the faces of the tools' pin separated. Unthincking it will merge them (depending on CAD preferences)
- LAST TOOL ONLY: export the tool of the component placed at the top of the stack
- TABLES: includes base gauge's geometry
- CUBES/PALLET: includes pallet, shoulders and cubes geometry
- BREYLS: includes breyls' geometry. Activates also SUPPORT and TOOLS options.
- SUPPORT: includes supports' geometry

❑ *Export the clamping definition to CAM*

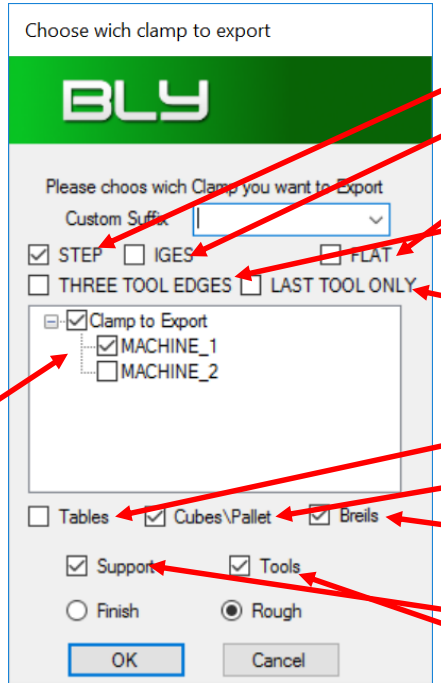
Once the clamping has been defined, it is possible to export the geometry of the FCS' components and the placement seats to a STEP or IGES file, which can then be used as exchange file to a 3D CAD system.
It is important to remember that the command works on the active environment (CAD or CAM) every time.

To start the procedure, hit



It opens the export options window.

Selectable Machines to export

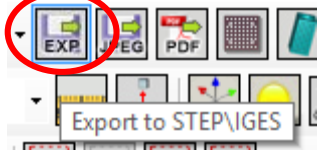


- STEP: creates the STEP file
- IGES: creates the IGES file
- FLAT: places all geometry as single component, without creating assembly structure
- THREE TOOL EDGES: keeps the faces of the tools' pin separated. Unthincking it will merge them (depending on CAD preferences)
- LAST TOOL ONLY: export the tool of the component placed at the top of the stack
- TABLES: includes base gauge's geometry
- CUBES/PALLET: includes pallet, shoulders and cubes geometry
- BREYLS: includes breyls' geometry. Activates also SUPPORT and TOOLS options.
- SUPPORT: includes supports' geometry
- TOOLS: includes supports' geometry

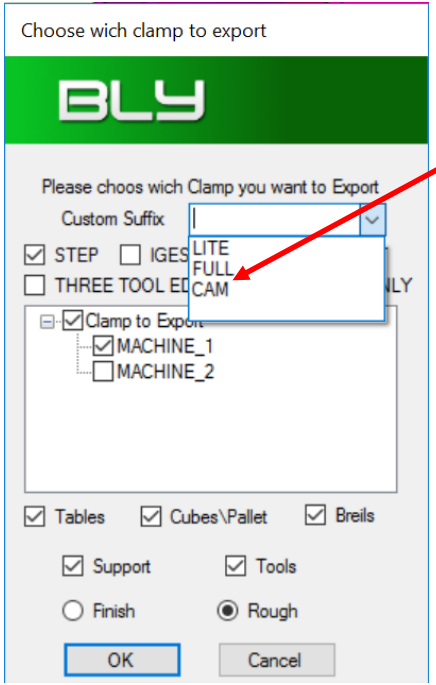
❑ *Export the clamping definition to CAM (cont'd)*

Once the clamping has been defined, the geometry of the FCS' components and the placement seats can be exported to a STEP or IGES file, which can then be used as exchange file for a 3D CAD system.

To start the procedure, hit



It opens the export options window



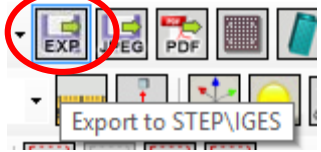
On the drop-down menu:

- LITE, FULL, CAM: export geometry contained in the library, if existing (see page 125)

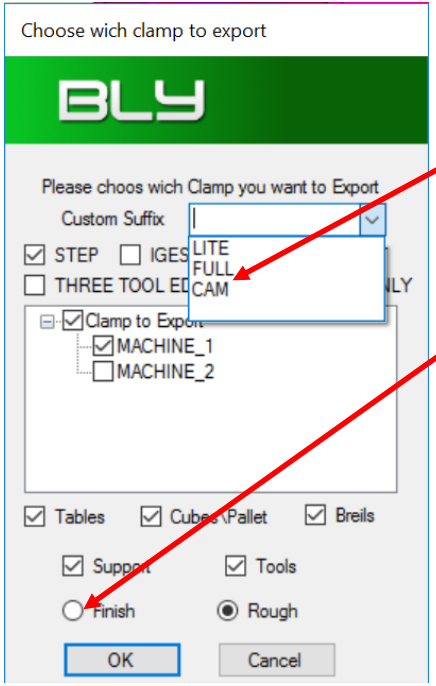
❑ *Export the clamping definition to CAM (cont'd)*

Once the clamping has been defined, the geometry of the FCS' components and the placement seats can be exported to a STEP or IGES file, which can then be used as exchange file for a 3D CAD system.

To start the procedure, hit



It opens the export options window



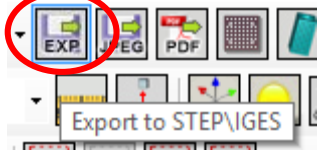
On the drop-down menu:

- LITE, FULL, CAM: export geometry contained in the library, if existing (see page 125)
- FINISH: export a larger tool, in order to create the seats for the finish milling

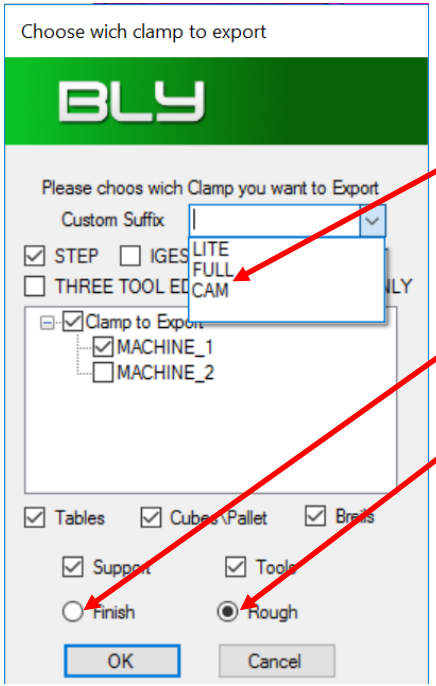
❑ *Export the clamping definition to CAM (cont'd)*

Once the clamping has been defined, the geometry of the FCS' components and the placement seats can be exported to a STEP or IGES file, which can then be used as exchange file for a 3D CAD system.

To start the procedure, hit



It opens the export options window

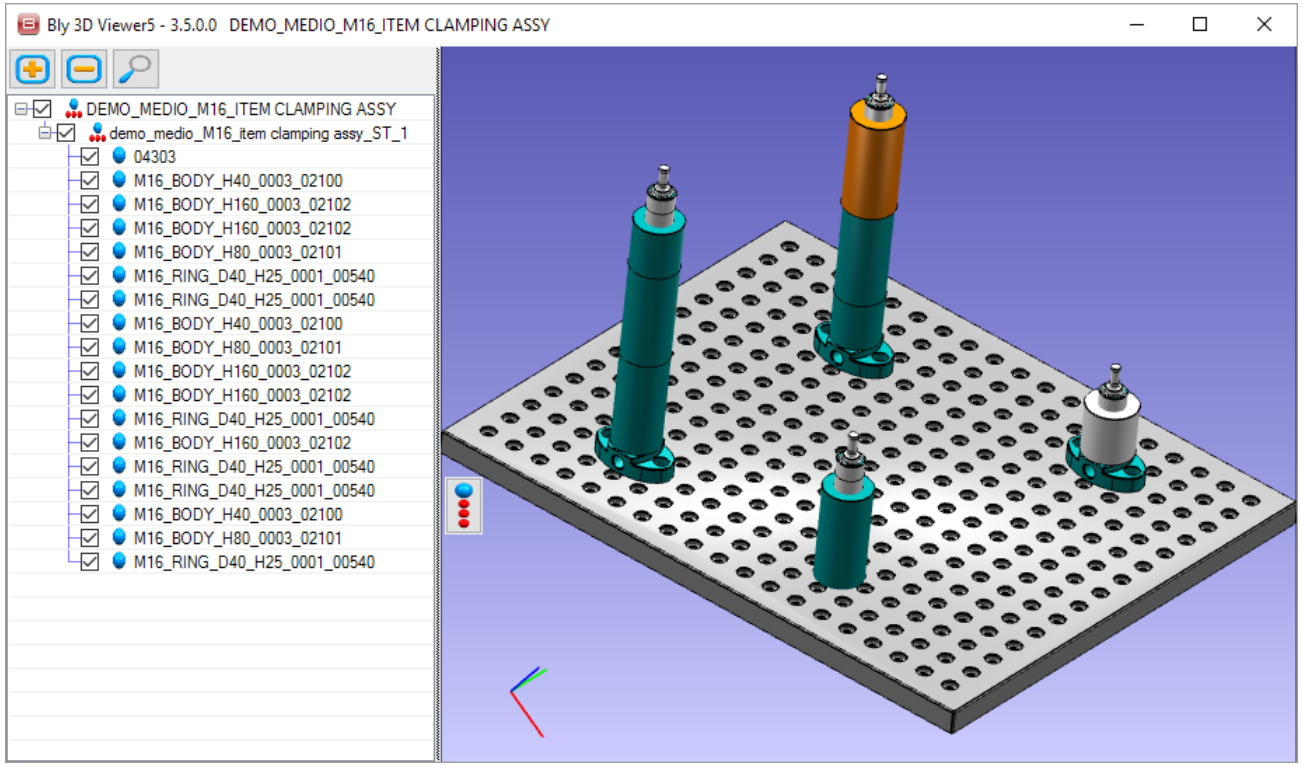


On the drop-down menu:

- LITE, FULL, CAM: export geometry contained in the library, if existing (see page 125)
- FINISH: export a larger tool, in order to create the seats for the finish milling
- ROUGH: exports a smaller tool, in order to create the seats for the rough milling

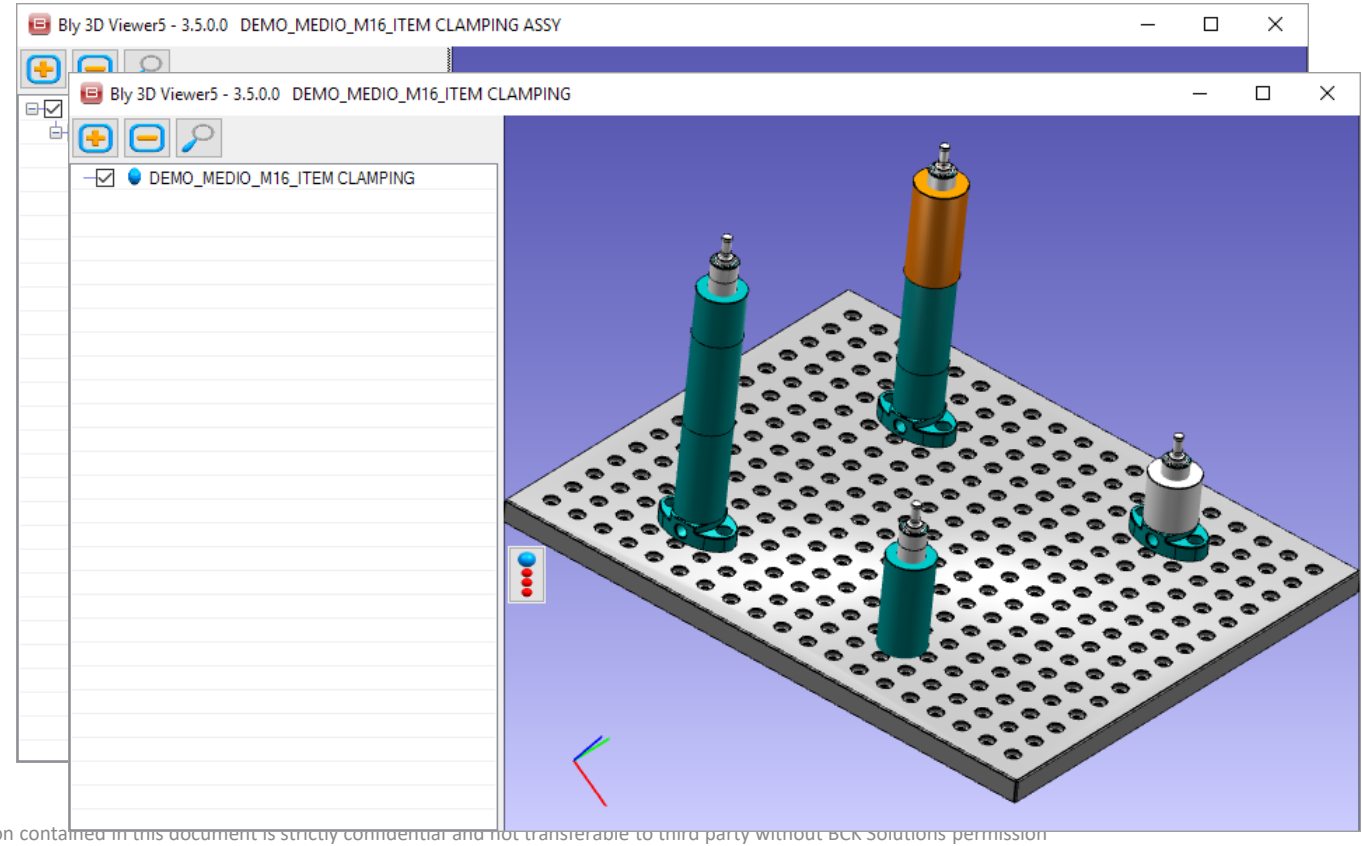
❑ *Export the clamping definition to CAM (cont'd)*

Example of exported assembly



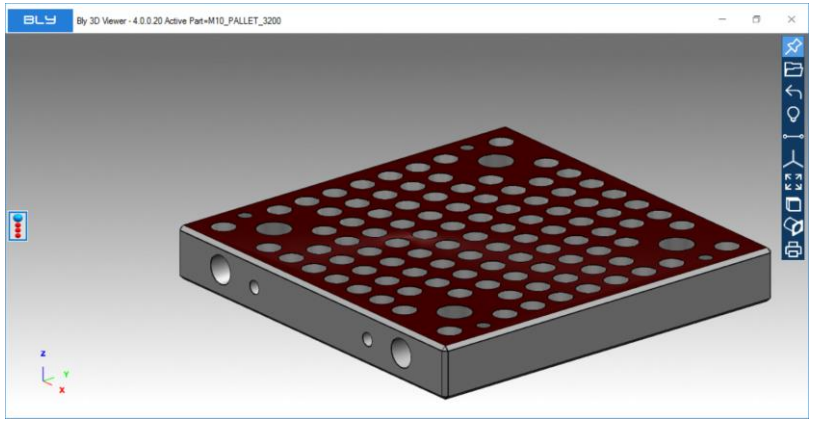
❑ *Export the clamping definition to CAM (cont'd)*

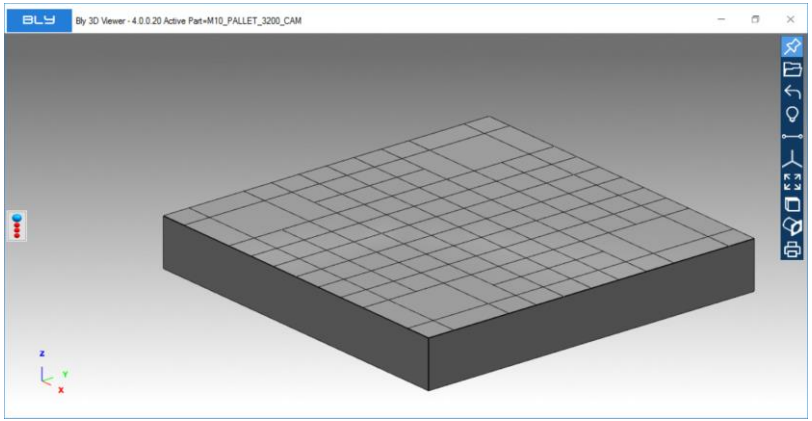
Example of exported FLAT



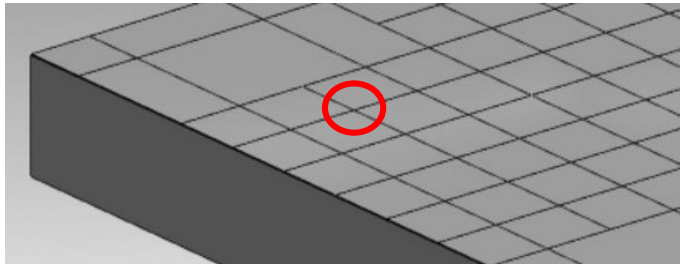
❑ *Export the clamping definition to CAM – Simplified libraries*

Guideline includes in its library simplified models of components library, particularly light, with graphical representation of the seats. Those files are named like the detailed part plus a «_CAM» suffix



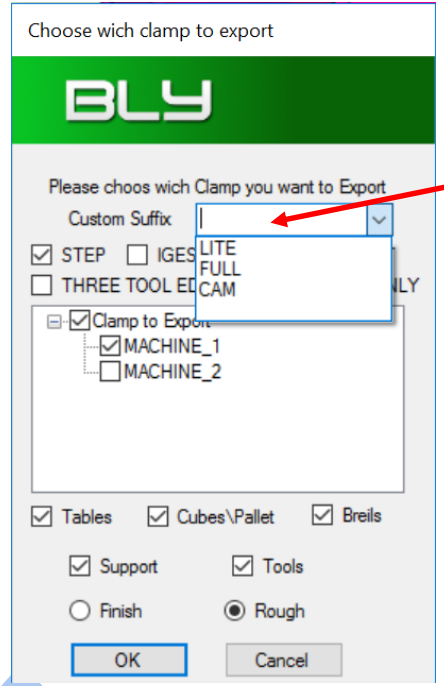


The pictures aboves show you an example of the detailed model (left) compared with the simplified pallet M10 (right). The grid of placement holes is represented by a net of lines, crossing in correspondance of holes' centers (see red circle in the picture on the right): this model has the advantage of being ready for CAMpath design, without any preparation work by the user to close the holes with surfaces.



❑ *Export the clamping definition to CAM – Simplified libraries*

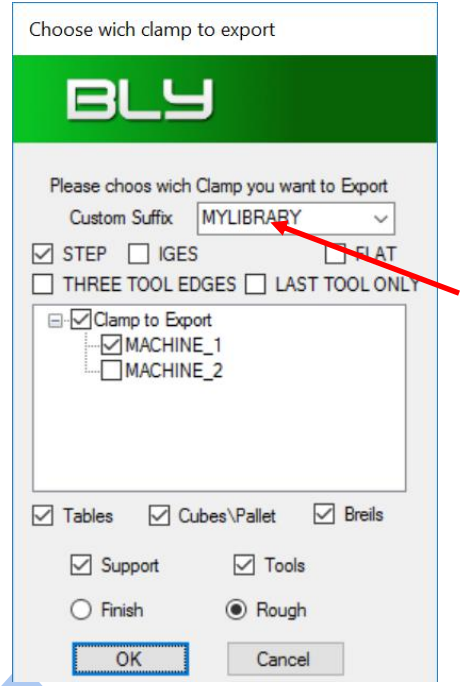
It is possible to create as many different versions of library as desired, to be used in different situation (i.e.: simplified models to be imported in CAD to increase translation performance, CAM design ect.)



Guideline has three pre-set options in the export menu (LITE, FULL, CAM) and the simplified models for CAM option: it is possible to create the models for the other 2 options simply generating the step files with the desired geometry and name them with the following convention:
[detailedmodelname]_LITE.stp (or _FULL or _CAM).
Guideline will then seek for the selected options' models and use them, if present, otherwise it will use the detailed ones.

❑ *Export the clamping definition to CAM – Simplified libraries*

It is possible to create as many different versions of library as desired, to be used in different situation (i.e.: simplified models to be imported in CAD to increase translation performance, CAM design ect.)



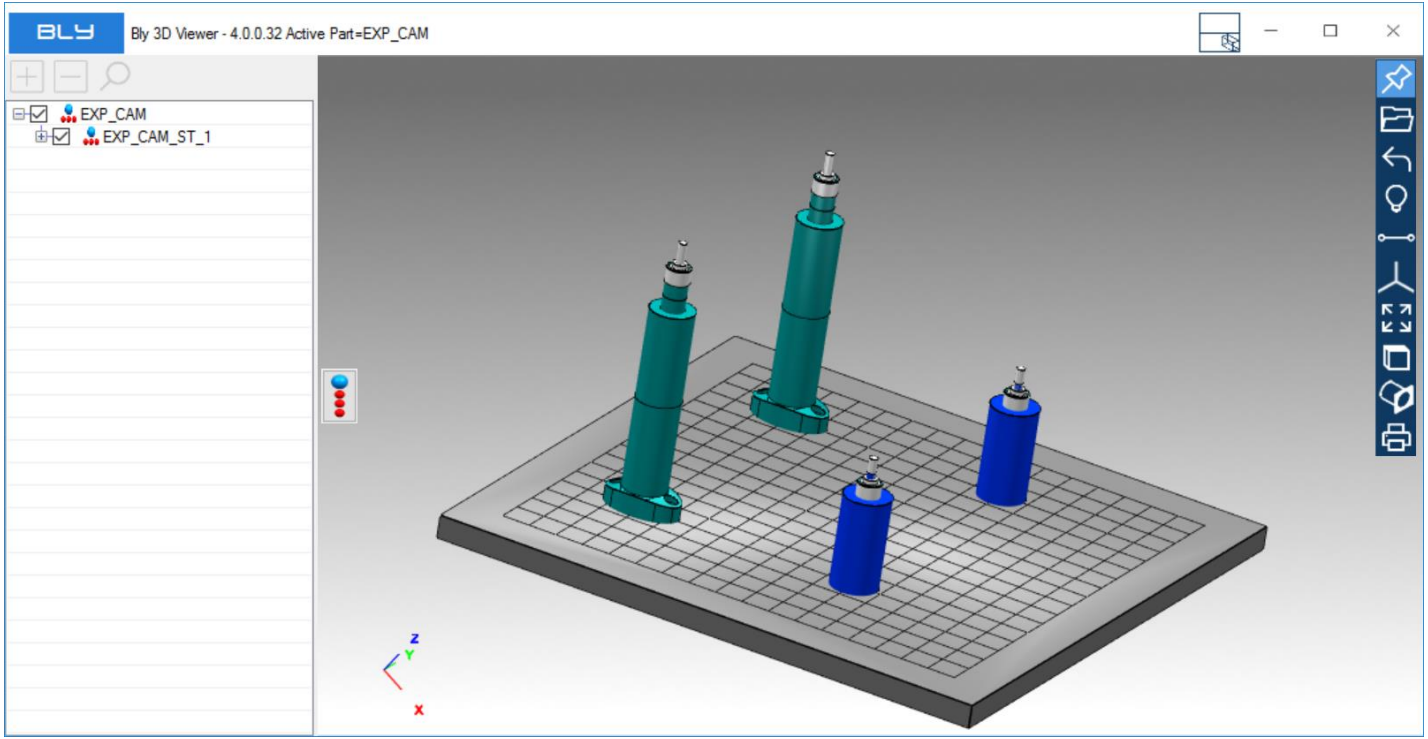
Guideline has three pre-set options in the export menu (LITE, FULL, CAM) and the simplified models for CAM option: it is possible to create the models for the other 2 options simply generating the step files with the desired geometry and name them with the following convention:
[detailedmodelname]_LITE.stp (or _FULL or _CAM).
Guideline will then seek for the selected options' models and use them, if present, otherwise it will use the detailed ones.

It is also possible to use a different option «on fly»: you can create components named «[detailedmodelname]_[optionname].stp», place them in the the library and type [optionname] in the Custom Suffix field in the export window.

In the example in the picture, [optionname]=MYMODELS

❑ *Export the clamping definition to CAM – Simplified libraries*

Example of simplified geometry exported using «CAM» library



❑ *Export clamping instructions to PDF*

It is possible to export the clamping setup(s) from Guideline to a PDF file: this way you can create a document distributable throughout your organization (i.e. as assembly instructions for machines operators). Normally, the generation of this document happens after the clamping design (CAM phase), which represent the actual machine setup, but it can be done from any clamping environment (CAD phase) or machines (CAM phase).

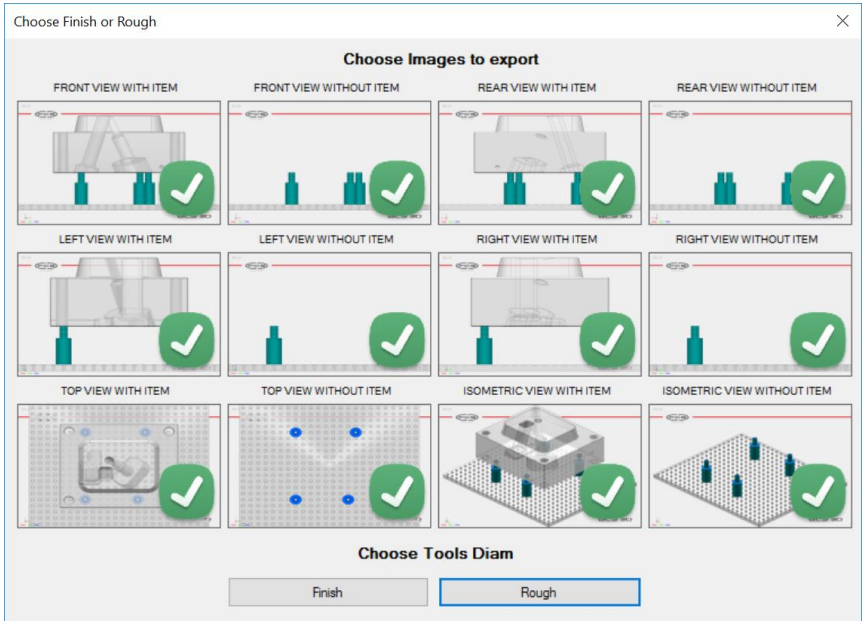
To launch the translation press 

❑ *Export clamping instructions to PDF*

It is possible to export the clamping setup(s) from Guideline to a PDF file: this way you can create a document distributable throughout your organization (i.e. as assembly instructions for machines operators). Normally, the generation of this document happens after the clamping design (CAM phase), which represent the actual machine setup, but it can be done from any clamping environment (CAD phase) or machines (CAM phase).

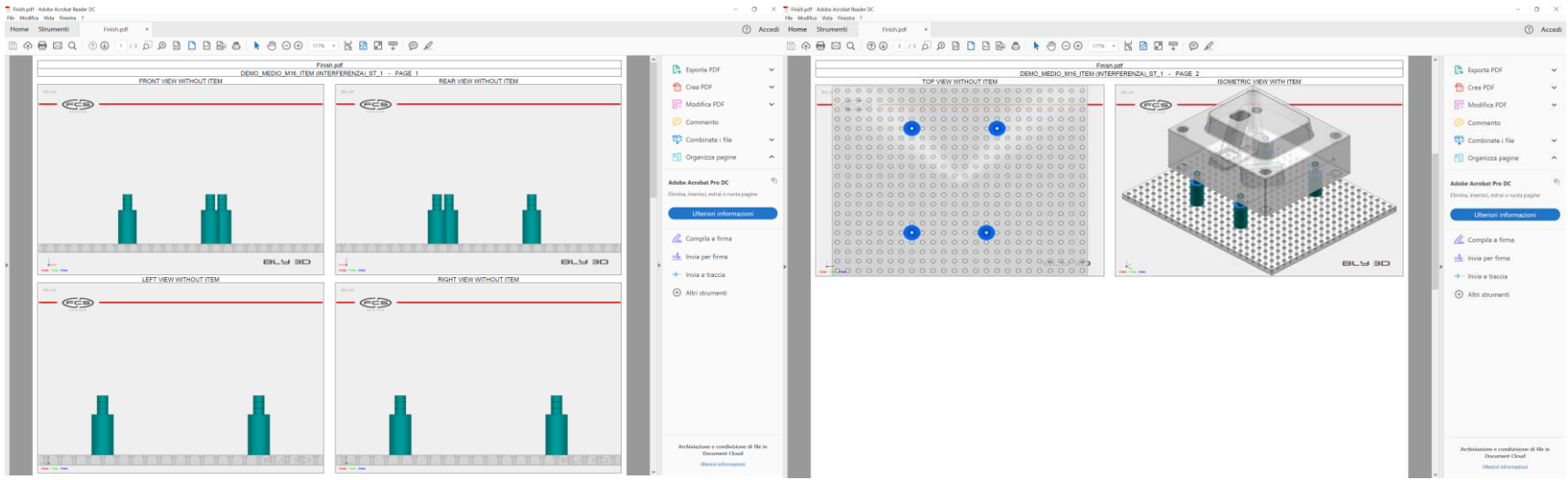
To launch the translation press 

After giving the name, you can choose the views you want to include and which size you want to use in the Bill Of Material (Finish or Rough).
After creation, the document will open automatically.



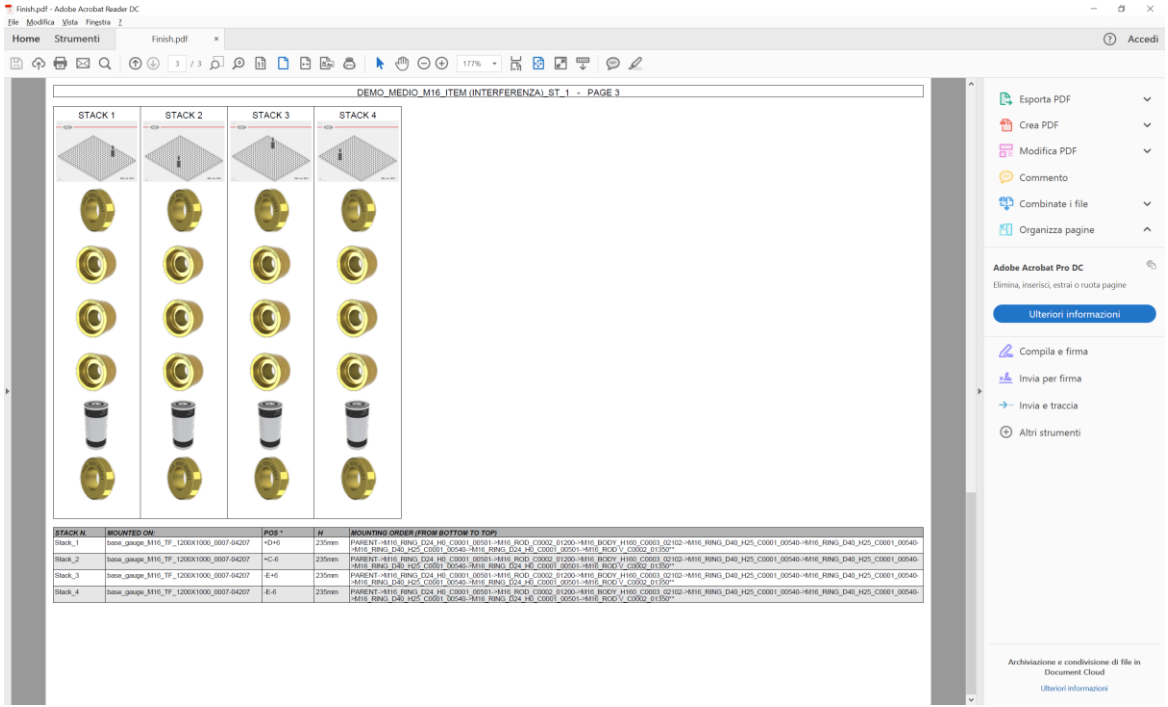
❑ *Export clamping instructions to PDF*

Depending on the number of views selected, the document will have up to 4 views in each page.



❑ *Export clamping instructions to PDF – Bill Of Materials*

The last page reports the BOM for each stack present in the clamping, information related to the position and the lenght of the stacks and the mounting order of each stacks' components.

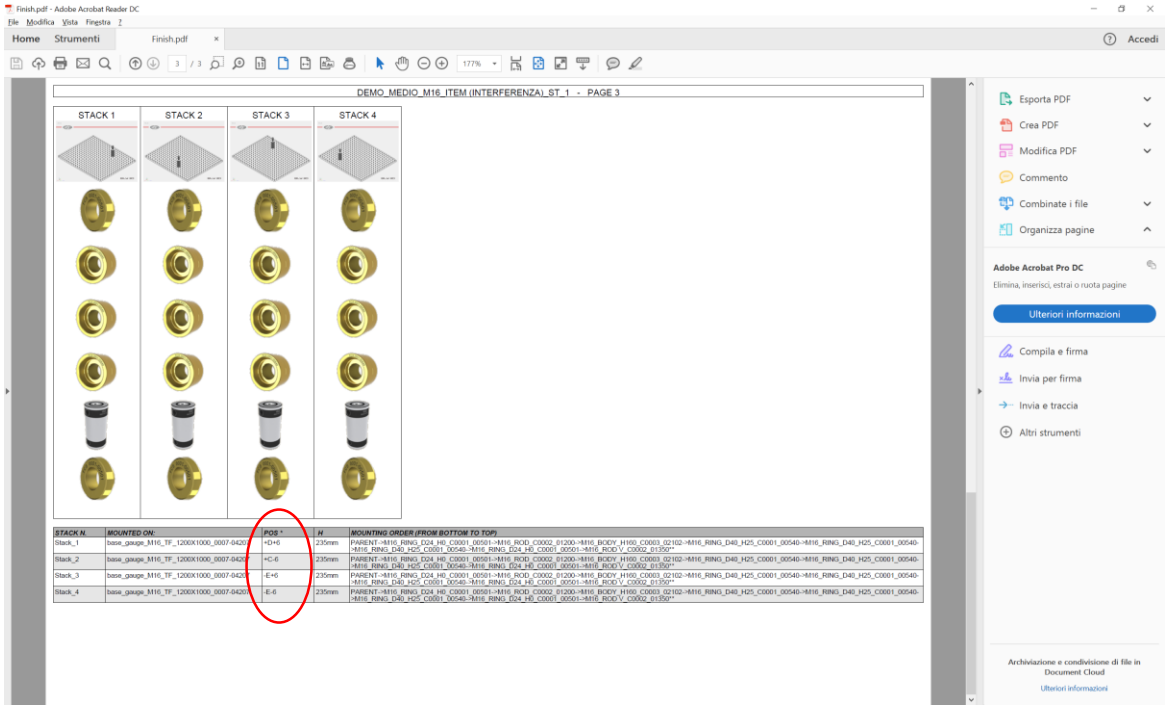


PHASE 2: CLAMPING DESIGN

❑ *Export clamping instructions to PDF – Bill Of Materials*

The last page reports the BOM for each stack present in the clamping, information related to the position and the lenght of the stacks and the mounting order of each stacks' components.

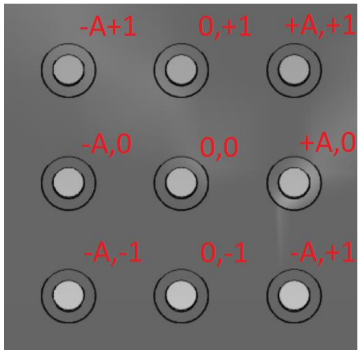
The positions of the stacks are also reported, using the «battleship» coding to identify the holes aligned with the center of the each stack.



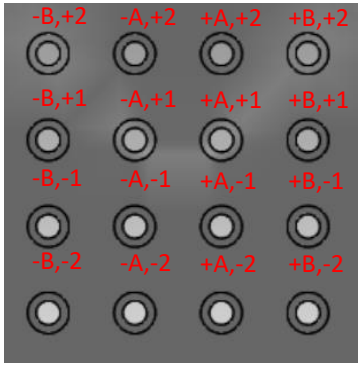
❑ *Export clamping instructions to PDF – Battleship*

The default convention for the battleship is the following:

- Base Gauges type «TR» (with a hole in the center)
  - Central hole is marked «0,0»;
  - Lines are named with letters, + for the upper side, - for the lower side;
  - Columns are named with numbers, + for the right side, - for the left side



- Base Gauges type «TF» (with a hole in the center)
  - Central hole is marked «0,0»;
  - Lines are named with letters, + for the upper side, - for the lower side;
  - Columns are named with numbers, + for the right side, - for the left side



❑ *Export clamping instructions to PDF – Alias*

It order to facilitate communication and parts management, a large number of companies uses internal names to identify FCS components instead of the original codes.
It is possible to set those names in Guideline and make them listed into the clamping instructions.

Open the file «Alias_default.txt» in \GuideLine3\Misc folder and add the alias you want to include in your documentation. Some examples with the right syntax are provided in the file.
Save as «Alias.txt» still in \GuideLine3\Misc folder and restart Guideline: your aliases will be listed into the BOM.

STACK N.	MOUNTED ON:	POS *	H	MOUNTING ORDER (FROM BOTTOM TO TOP)
Stack_1	base_gauge_M16_TR_1000X1000_0007-04304	+G-2	120mm	PARENT->M16_RING_D24_H0_C0001_00501->M16_SCREW TCEI_H40_C0940->M16_SCREW TCEI_H40_C0940->>M16_ROD W_C0002_01600->M16_BODY_H80_C0003_02101->M16_RING_D24_H0_C0001_00501->M16_ROD_C0
Stack_2	base_gauge_M16_TR_1000X1000_0007-04304	A0	40mm	PARENT->M12_M16_BODY_H40_C0003_02000**
Stack_3	base_gauge_M16_TR_1000X1000_0007-04304	+C0	40mm	PARENT->M12_M16_BODY_H40_C0003_02000**
Stack_4	base_gauge_M16_TR_1000X1000_0007-04304	A+2	40mm	PARENT->M12_M16_BODY_H40_C0003_02000**
Stack_5	base_gauge_M16_TR_1000X1000_0007-04304	+C+2	40mm	PARENT->M12_M16_BODY_H40_C0003_02000**
Stack_6	M8_PALLET_0005_03101	A+8	50mm	PARENT->M8_RING_D25_H10_C0001_00210->M8_RING_D25_H10_C0001_00210->M8_RING_D25_H10_C0001_00>M8_RING_D25_H10_C0001_00210->M8_SCREW TCEI_H40_->M8_RING_D14_H0_C0001_00201**

STACK N.	MOUNTED ON:	POS *	H	MOUNTING ORDER (FROM BOTTOM TO TOP)
Stack_1	Machine 1	+G-2	120mm	PARENT->M16_RING_D24_H0_C0001_00501->M16_SCREW TCEI_H40_C0940->M16_SCREW TCEI_H40_C0940->>M16_ROD W_C0002_01600->Body M16 H80->M16_RING_D24_H0_C0001_00501->M16_ROD_C0002_01200**
Stack_2	Machine 1	A0	40mm	PARENT->M12_M16_BODY_H40_C0003_02000**
Stack_3	Machine 1	+C0	40mm	PARENT->M12_M16_BODY_H40_C0003_02000**
Stack_4	Machine 1	A+2	40mm	PARENT->M12_M16_BODY_H40_C0003_02000**
Stack_5	Machine 1	+C+2	40mm	PARENT->M12_M16_BODY_H40_C0003_02000**
Stack_6	Pallet M8	A+8	50mm	PARENT->M8_RING_D25_H10_C0001_00210->M8_RING_D25_H10_C0001_00210->M8_RING_D25_H10_C0001_00>M8_RING_D25_H10_C0001_00210->->M8_RING_D14_H0_C0001_00201**

SIEMENS NX INTEGRATION

❑ *Introduction to Siemens NX integration*

Guideline offers the possibility to integrate with Siemens NX: this speeds up the information exchange with CAD and helps user to gather all aspects related to the clamping in one single environment, increasing the workflow effectivness and saving time. To properly configure the integration, please refer to the installation manual.

The integration supports either single part model or assembly configuration: options will be explained in details in the following pages.

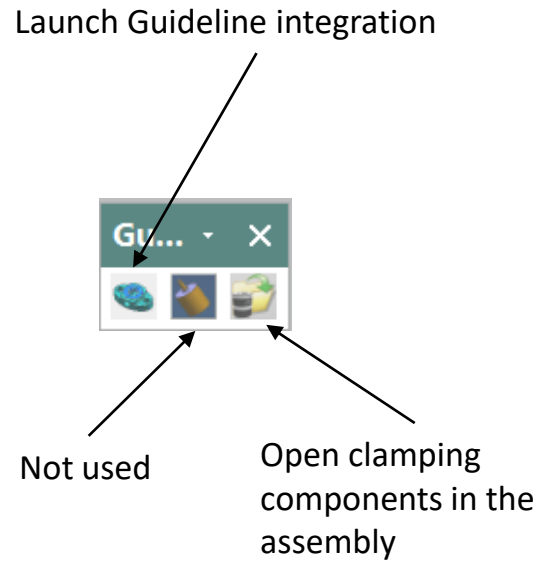
However, option #2 (assembly configuration) is the optimal one, in order to create a dedicated environment and maximize the usage of assembly capabilities in NX.

Guideline imports the geometry of the part directly from NX (avoiding STEP conversion) and makes it ready for clamping definition. Both Desing and Machine environments are accessible, therefore the integration can be effectively used both for seats placement and for clamping configuration.

Once done, Guideline transfers back the clamping components to NX and creates the assembly structure; seats creation is also supported.

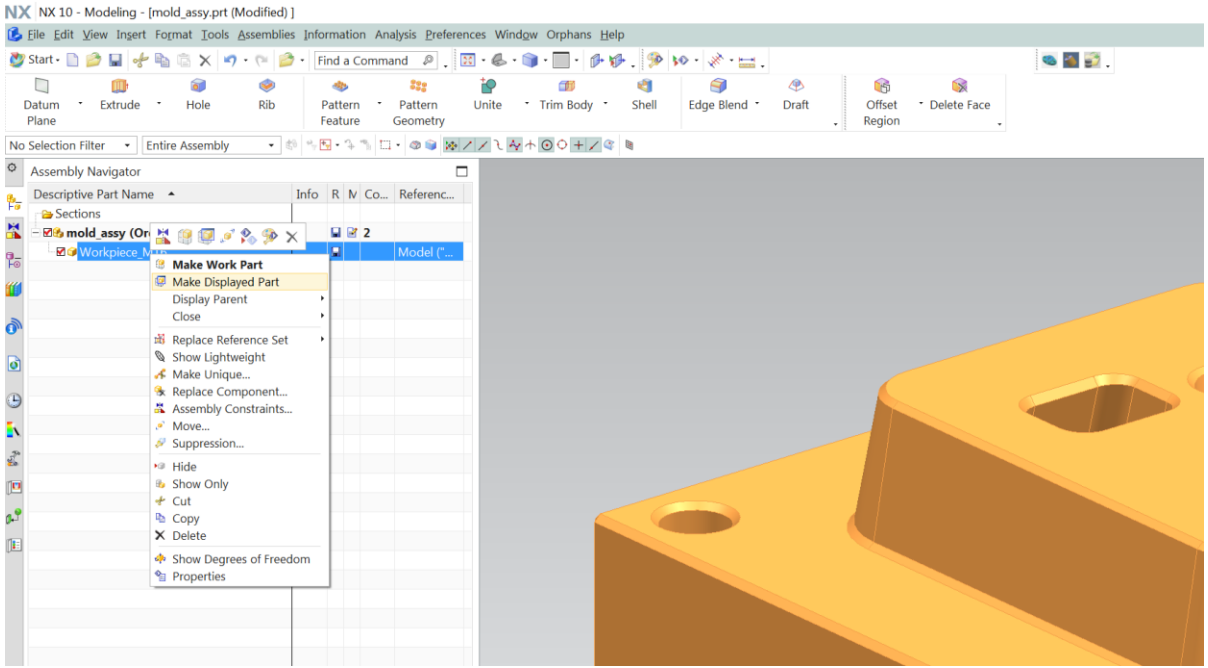
Finally, Guideline manages workpiece's modifications, keeping both environments aligned.

The integration creates an additional menu and toolbar in NX, as shown in the picture



❑ *Creating clamps in NX #1: using the job*

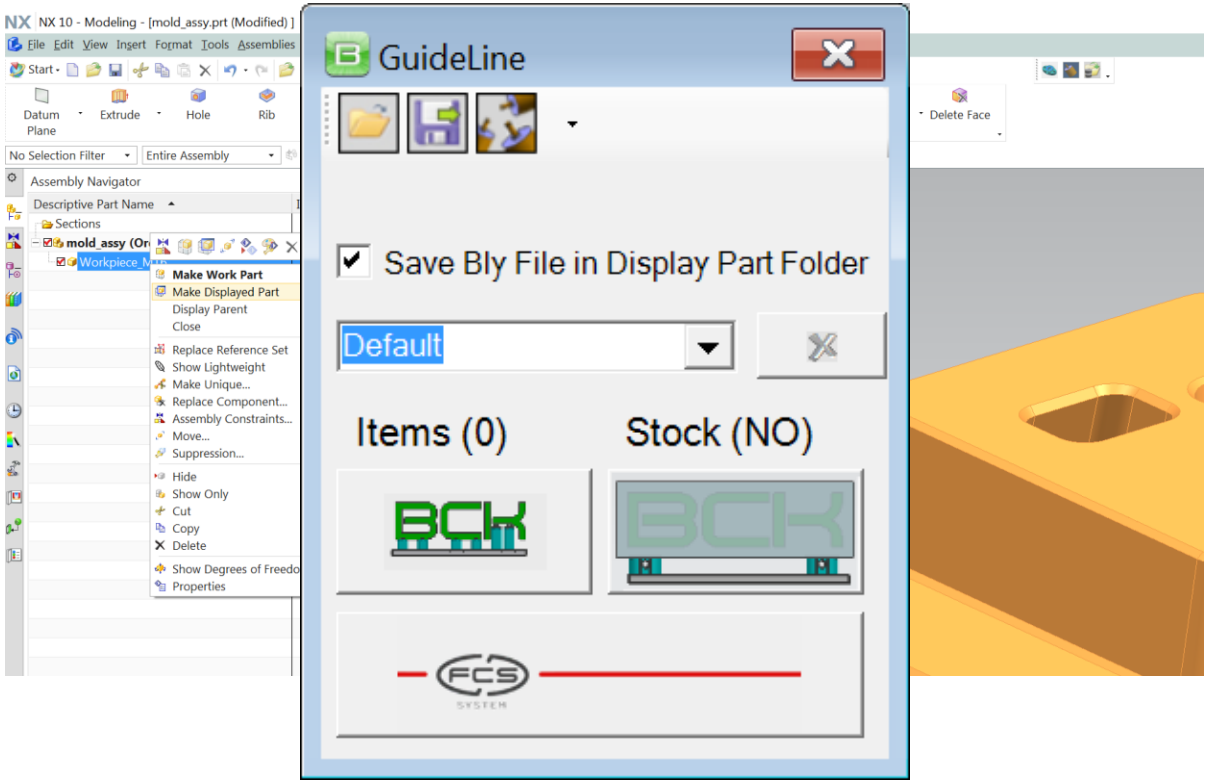
In NX open, make the work part or make displayed part the job.



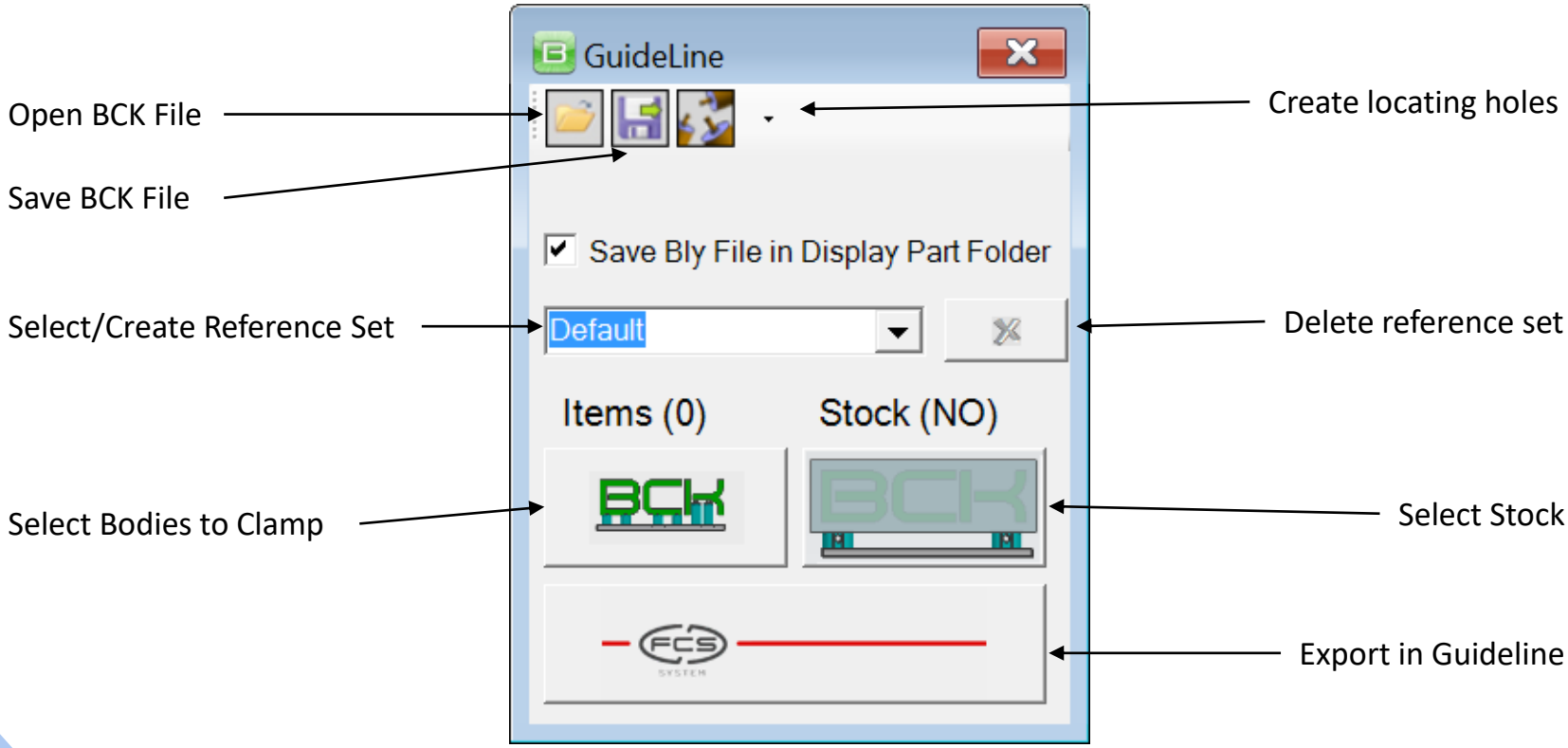
❑ *Creating clamps in NX #1: using the job (con't)*

In NX open, make the work part or make displayed part the job.

Press «Guideline» button  from the additional toolbar: the main Guideline panel is displayed.

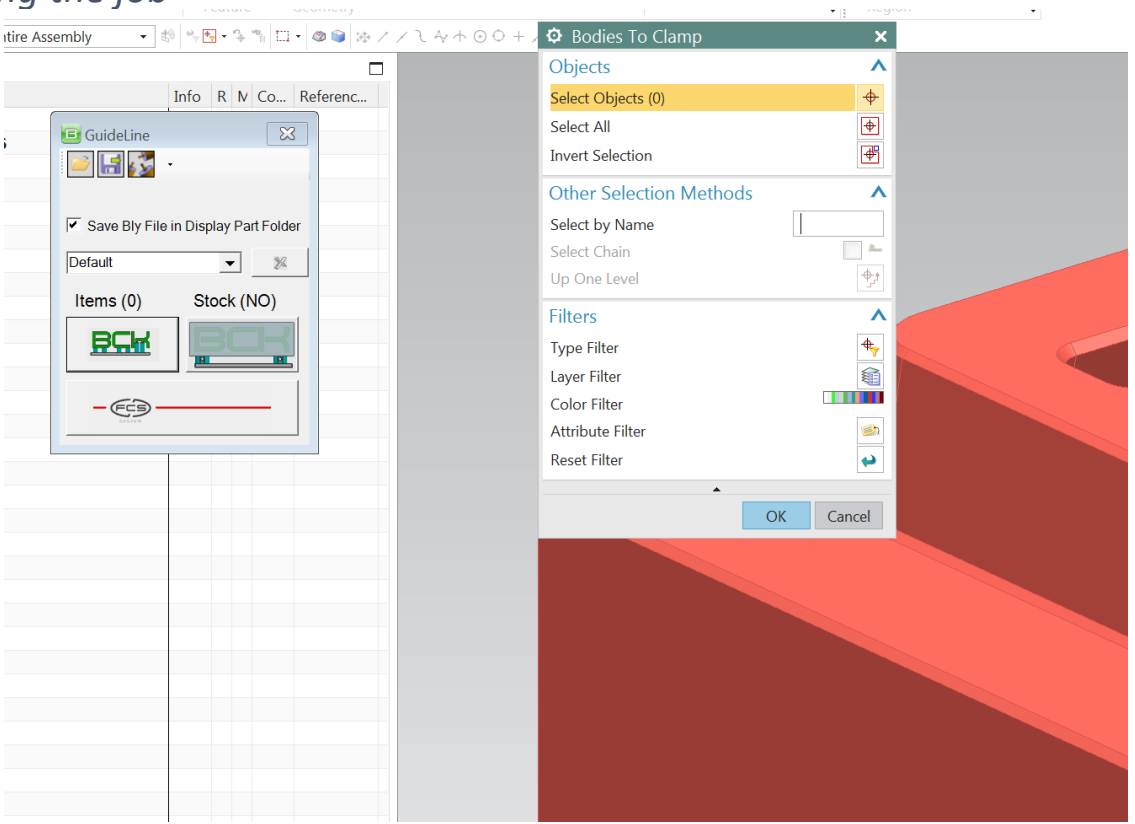


❑ *Creating clamps in NX #1: using the job (con't)*



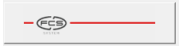
❑ *Creating clamps in NX #1: using the job*

Body to be exported must be selected first: from Guideline window, press «Select Bodies to Clamp»  , click on the job's solid body and confirm with OK.

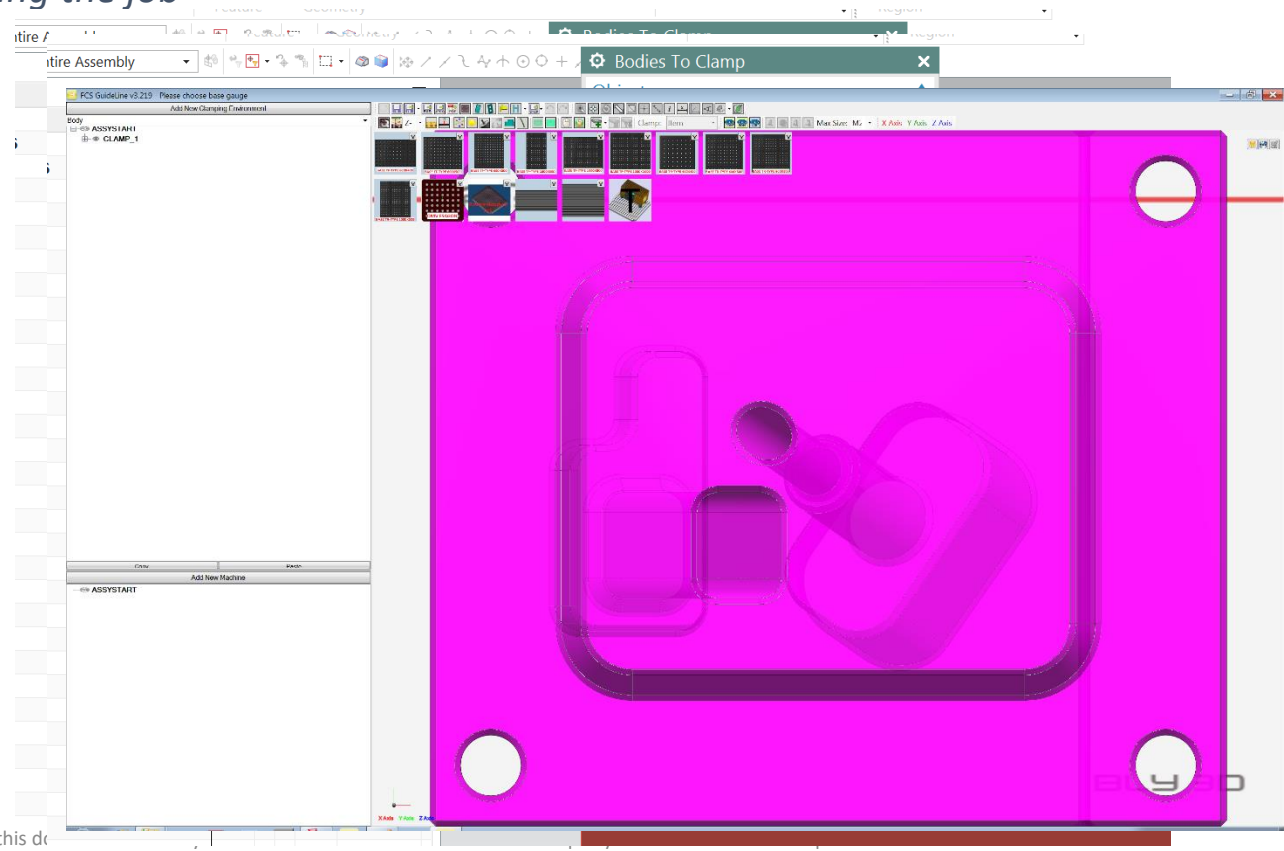


❑ *Creating clamps in NX #1: using the job*

Body to be exported must be selected first: from Guideline window, press «Select Bodies to Clamp»  , click on the job's solid body and confirm with OK.

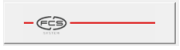
Guideline's window will show that one item has been selected: you can now press «Export in Guideline»  and launch Guideline.

In Guideline you can select the base gauge and clamp as usual.



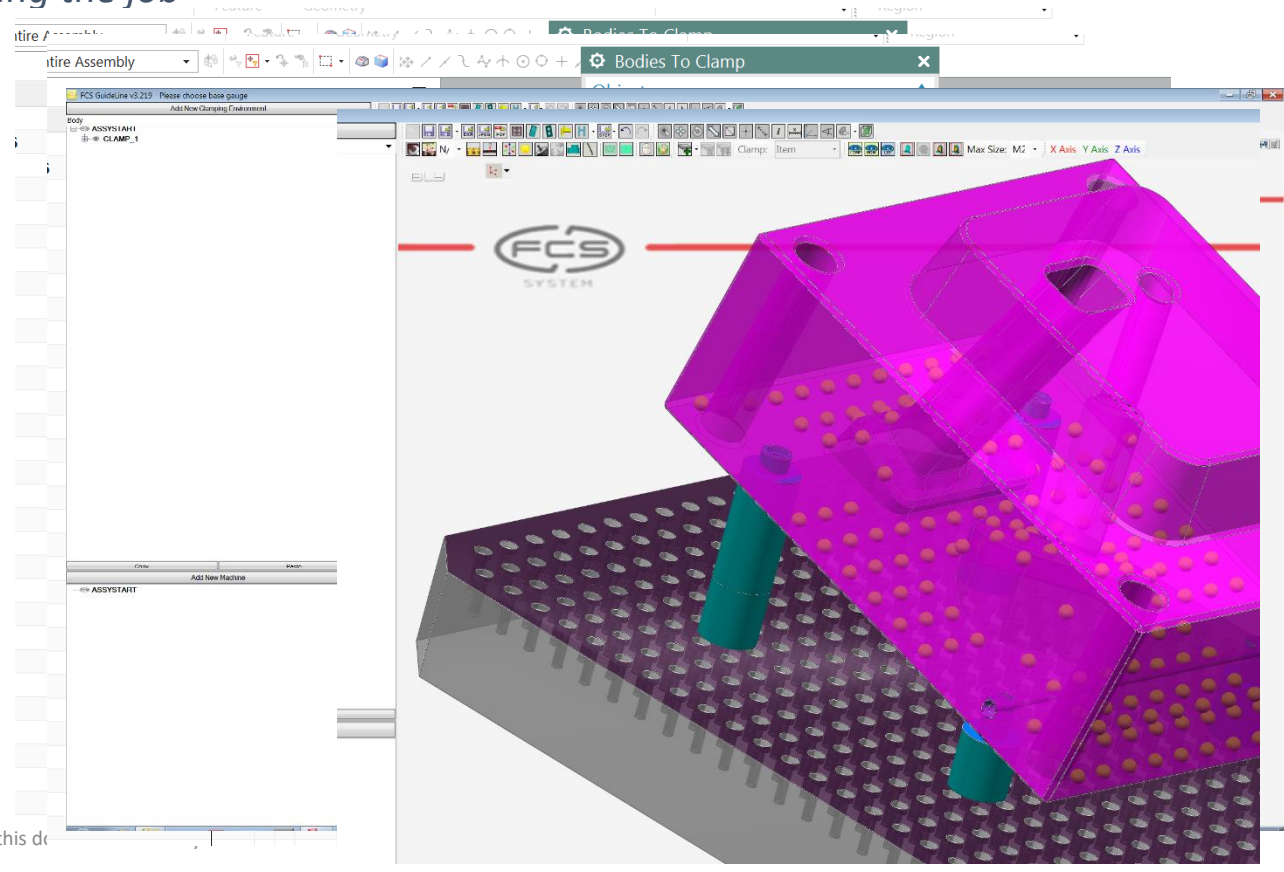
❑ *Creating clamps in NX #1: using the job*

Body to be exported must be selected first: from Guideline window, press «Select Bodies to Clamp»  , click on the job's solid body and confirm with OK.

Guideline's window will show that one item has been selected: you can now press «Export in Guideline»  and launch Guideline.

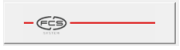
In Guideline you can select the base gauge and clamp as usual.

Once done, Save the clamping: Guideline will transfer all clamping components to NX, respecting the relative positioning while not changing job's CSYS.



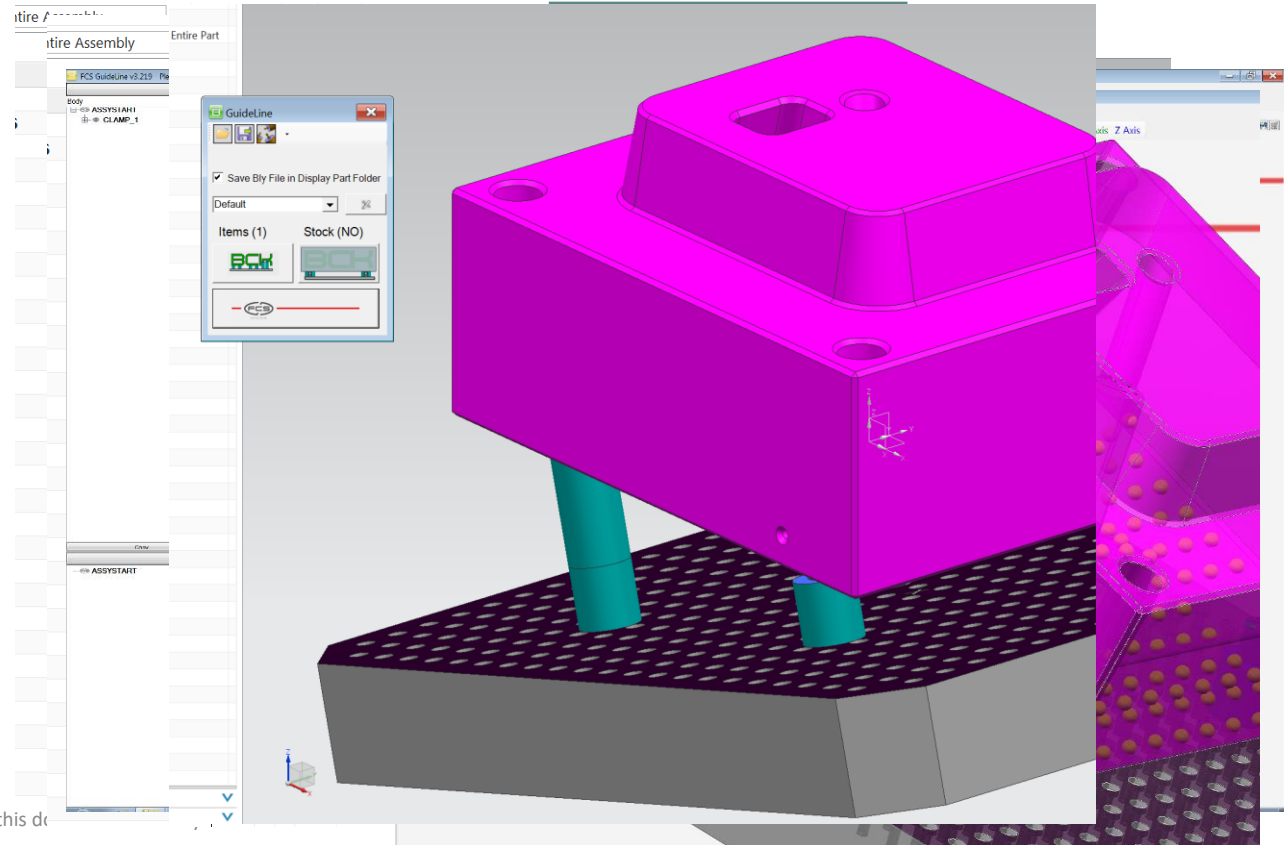
❑ *Creating clamps in NX #1: using the job*

Body to be exported must be selected first: from Guideline window, press «Select Bodies to Clamp»  , click on the job's solid body and confirm with OK.

Guideline's window will show that one item has been selected: you can now press «Export in Guideline»  and launch Guideline.

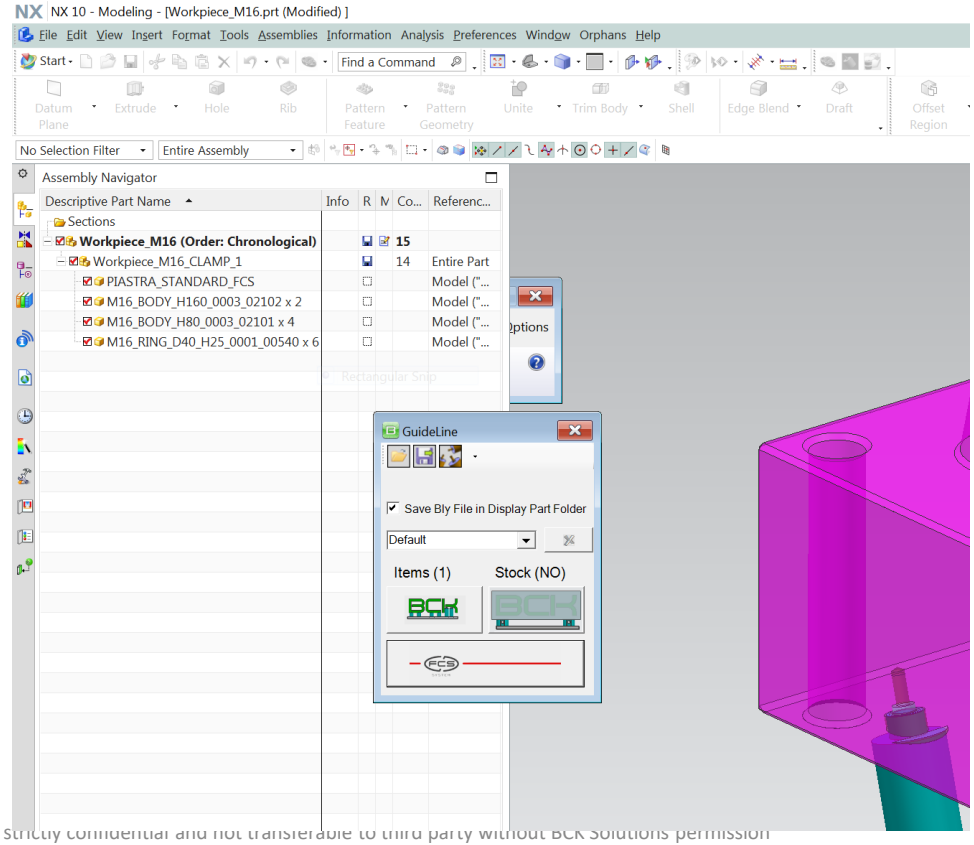
In Guideline you can select the base gauge and clamp as usual.

Once done, Save the clamping: Guideline will transfer all clamping components to NX, respecting the relative positioning while not changing job's CSYS.



❑ *Creating clamps in NX #1: using the job*

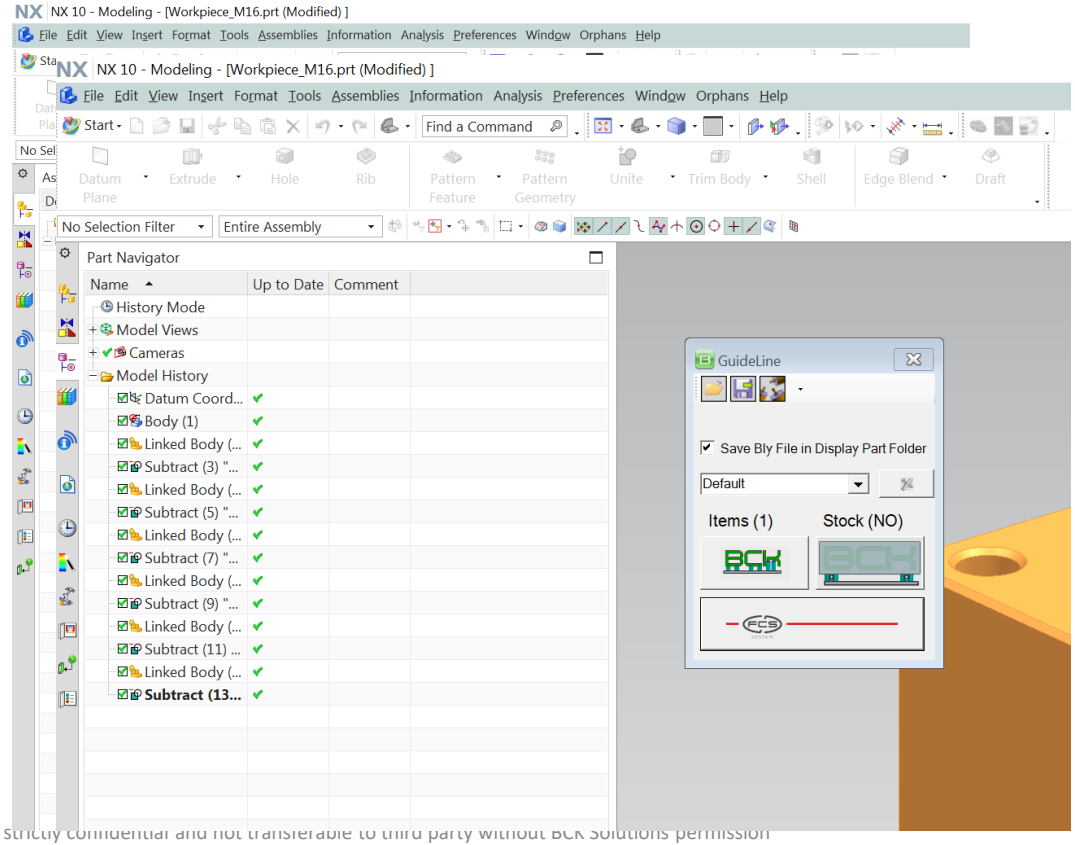
Guideline creates also the clamping's assembly structure in NX: clamping components are added as components of the job's file and can be navigated as usual.



❑ *Creating clamps in NX #1: using the job*

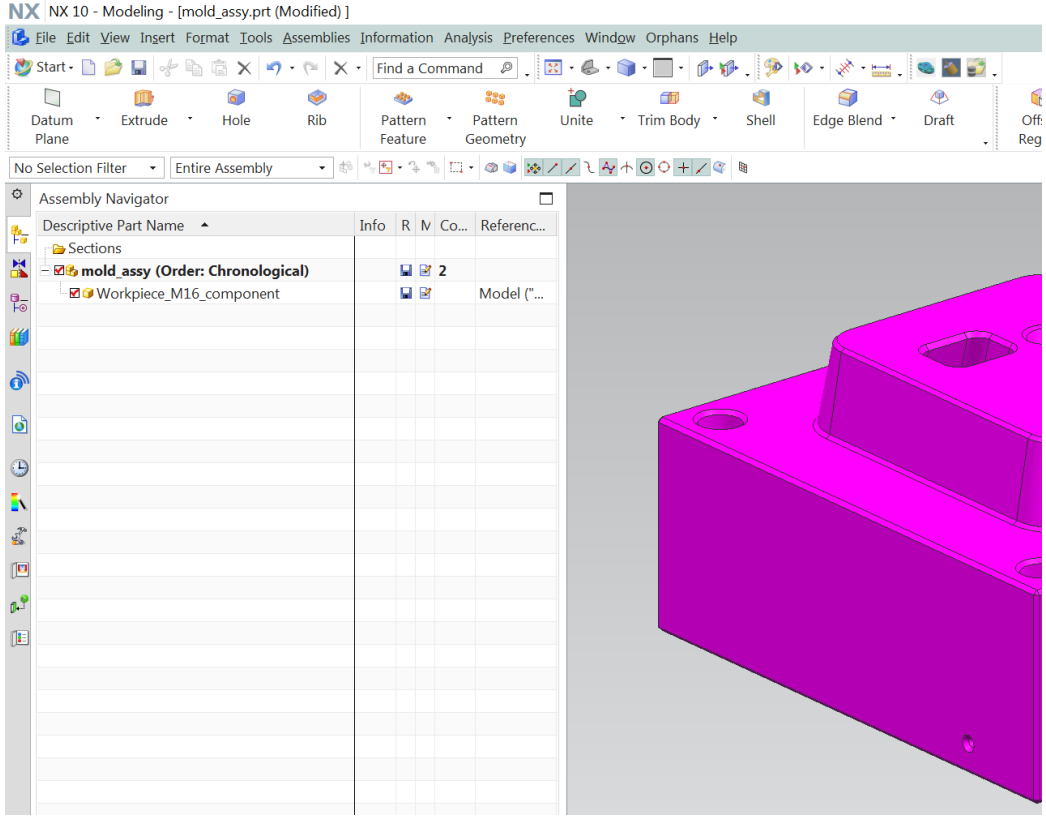
Guideline creates also the clamping's assembly structure in NX: clamping components are added as components of the job's file and can be navigated as usual.

At this stage, locating holes can be created directly from Guideline window: press «Create all locating holes» and NX will create linked bodies of the seats within job's part and subtract them from its solid body. Features are created as usual for NX.



❑ *Creating clamps in NX #2: using the assembly*

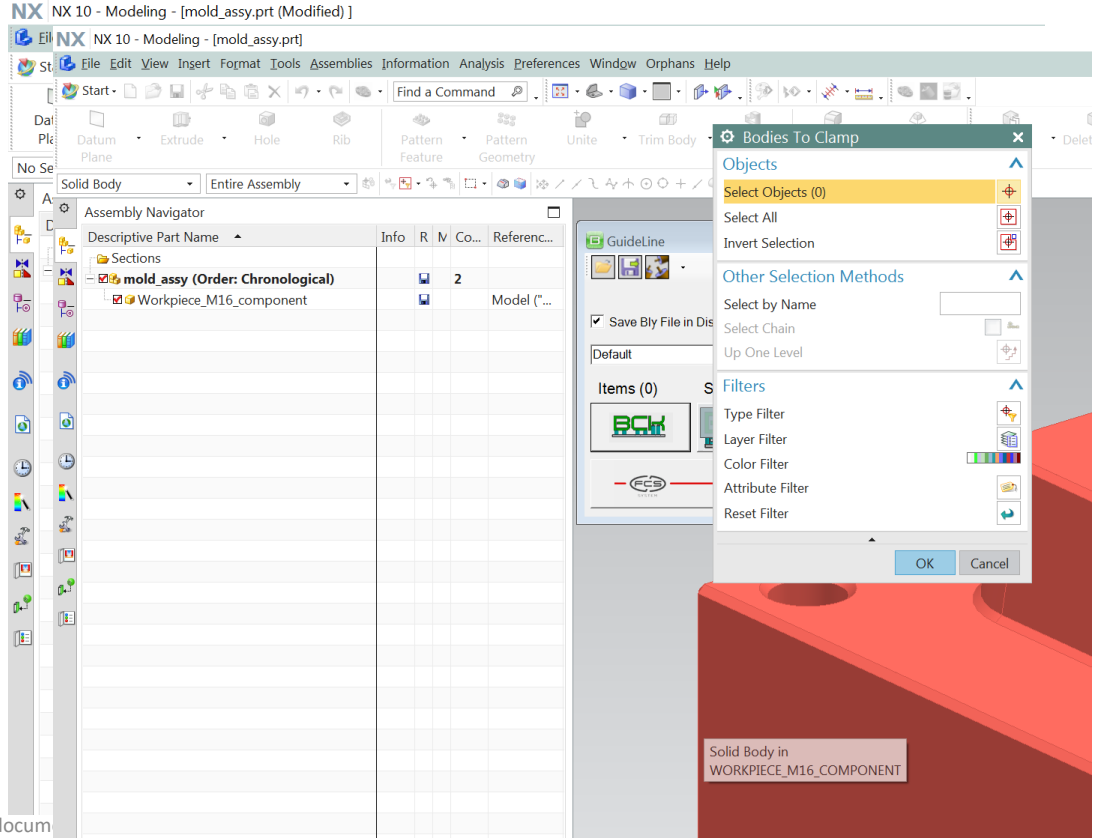
In NX you must select or create an assembly for the job to be clamped: this can be either a dedicated assembly as well as a project's sub-assembly.



❑ *Creating clamps in NX #2: using the assembly*

In NX you must select or create an assembly for the job to be clamped: this can be either a dedicated assembly as well as a project's sub-assembly.

Launch GL integration: «Select Bodies to Clamp»  will allow you to select the job no matter the level it is in the assembly's structure.

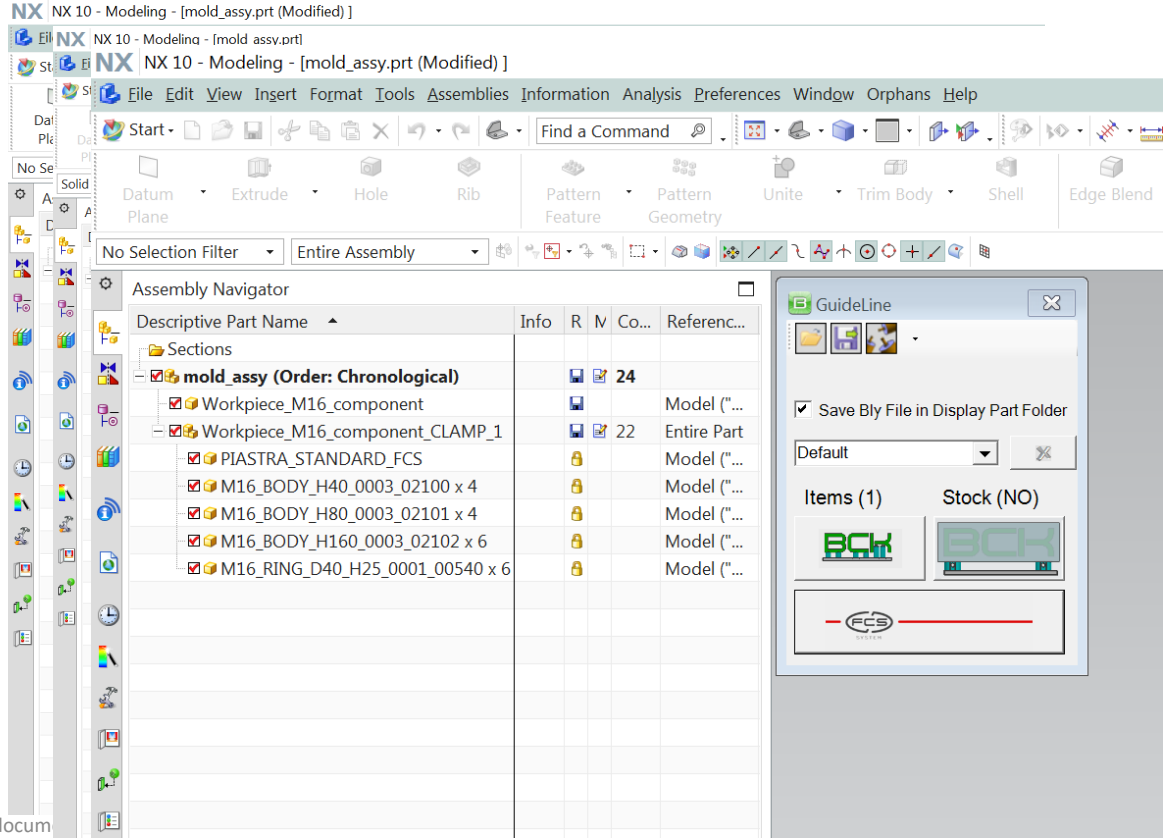


❑ *Creating clamps in NX #2: using the assembly*

In NX you must select or create an assembly for the job to be clamped: this can be either a dedicated assembly as well as a project's sub-assembly.

Launch GL integration: «Select Bodies to Clamp» will allow you to select the job no matter the level it is in the assembly's structure.

You can then launch Guideline and create the clamp(s) as usual: once saved, clamping will be tranferred back in NX and the components added under the job's parent.



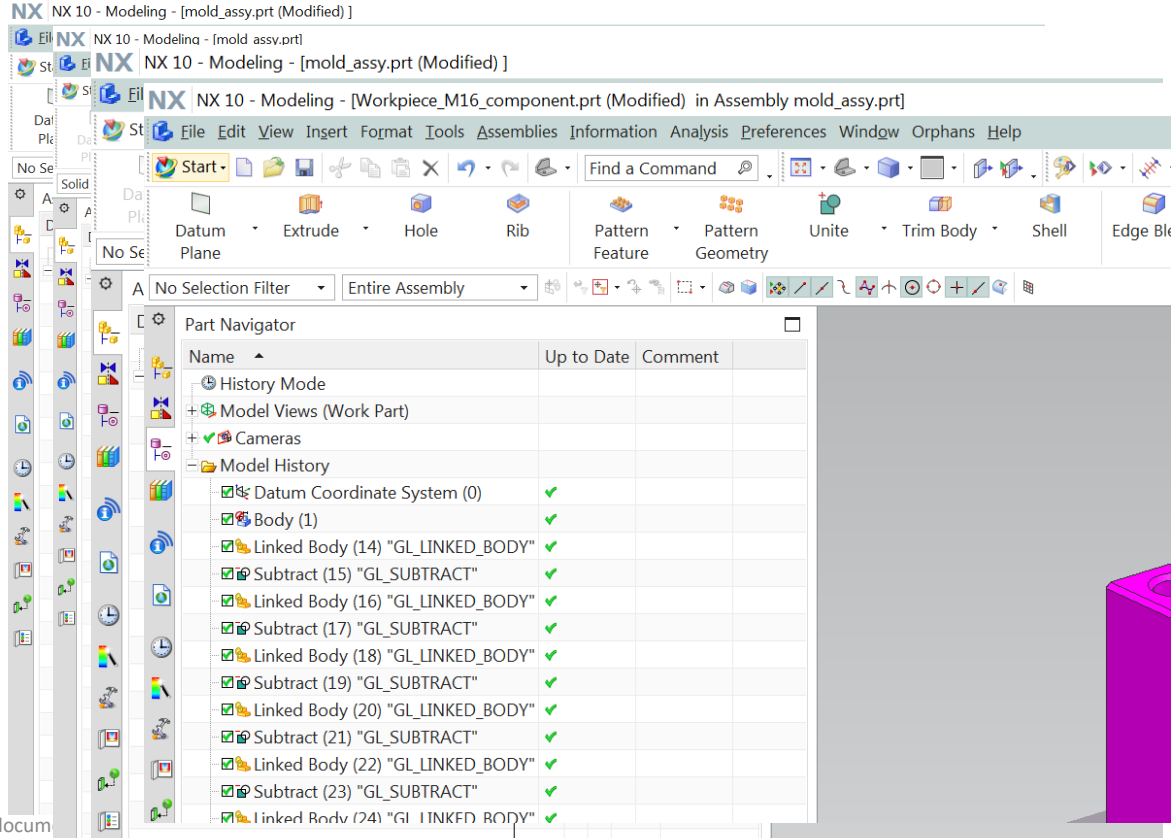
❑ *Creating clamps in NX #2: using the assembly*

In NX you must select or create an assembly for the job to be clamped: this can be either a dedicated assembly as well as a project's sub-assembly.

Launch GL integration: «Select Bodies to Clamp» will allow you to select the job no matter the level it is in the assembly's structure.

You can then launch Guideline and create the clamp(s) as usual: once saved, clamping will be tranferred back in NX and the components added under the job's parent.

However, locating holes will be correctly created in the job's file, as shown in the picture.



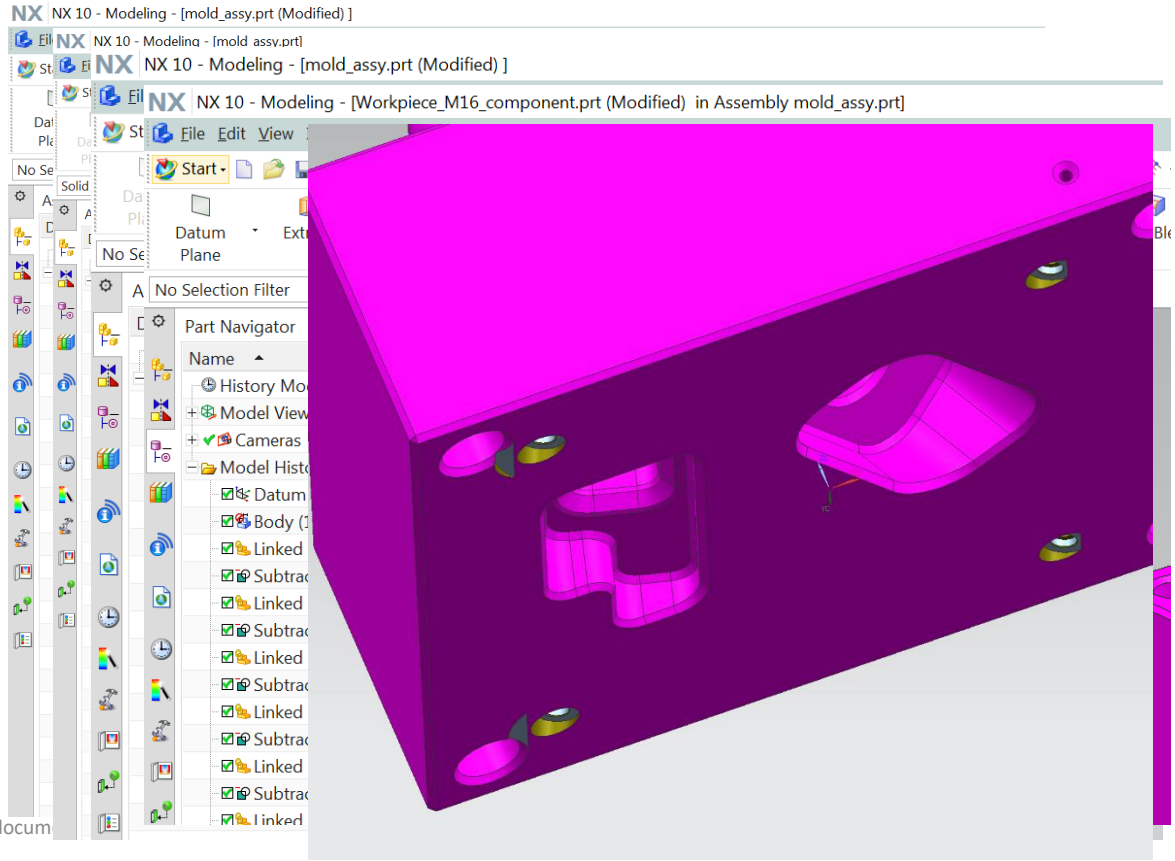
❑ *Creating clamps in NX #2: using the assembly*

In NX you must select or create an assembly for the job to be clamped: this can be either a dedicated assembly as well as a project's sub-assembly.

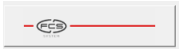
Launch GL integration: «Select Bodies to Clamp» will allow you to select the job no matter the level it is in the assembly's structure.

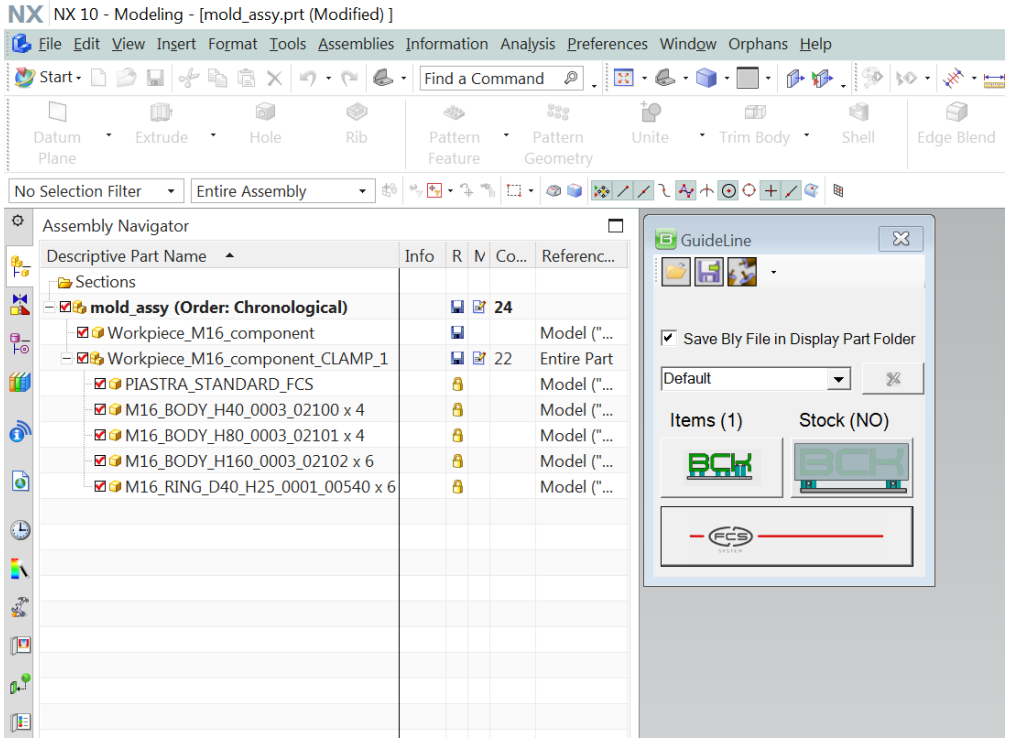
You can then launch Guideline and create the clamp(s) as usual: once saved, clamping will be tranferred back in NX and the components added under the job's parent.

However, locating holes will be correctly created in the job's file, as shown in the picture.



❑ *Modifying a clamp*

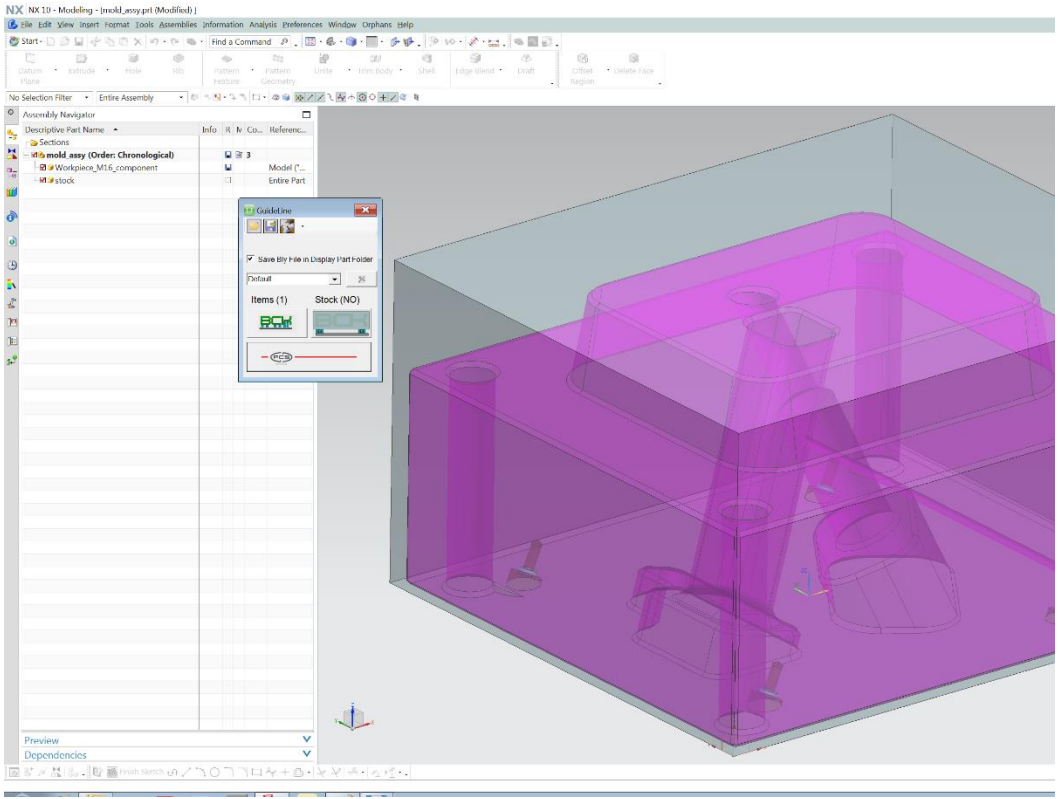
Once clamp(s) has been created, GL integration detects it and make it available for modification/update: to access it, press «Export in Guideline»  , with no need to selected the body once again.
Modifications made in Guideline will be then transferred back to NX when saving.



❑ *Stock addition*

GL Integration supports stock placement too.

Once stock is created in NX (either in the job or in the assembly), press «Select Stock»  button.

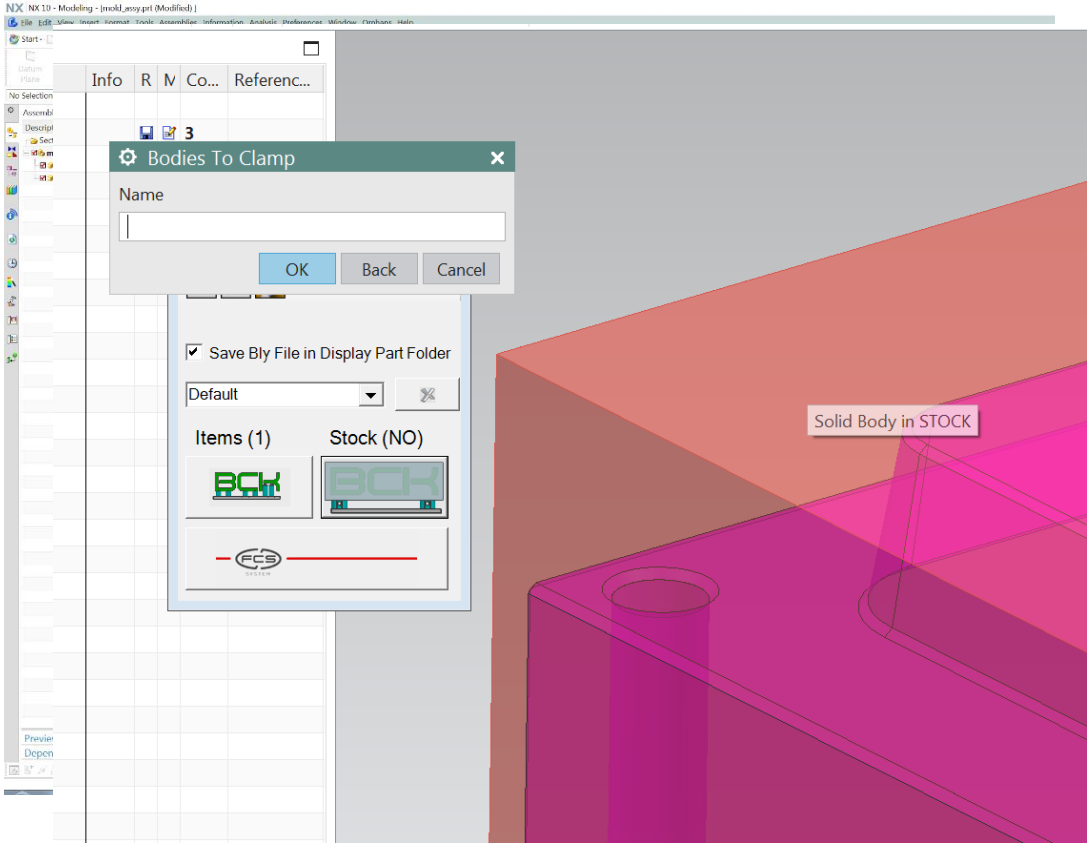


❑ *Stock addition*

Gl Integration supports stock placement too.

Once stock is created in NX (either in the job or in the assembly), press «Select Stock»  button.

You can then select the stock's body.



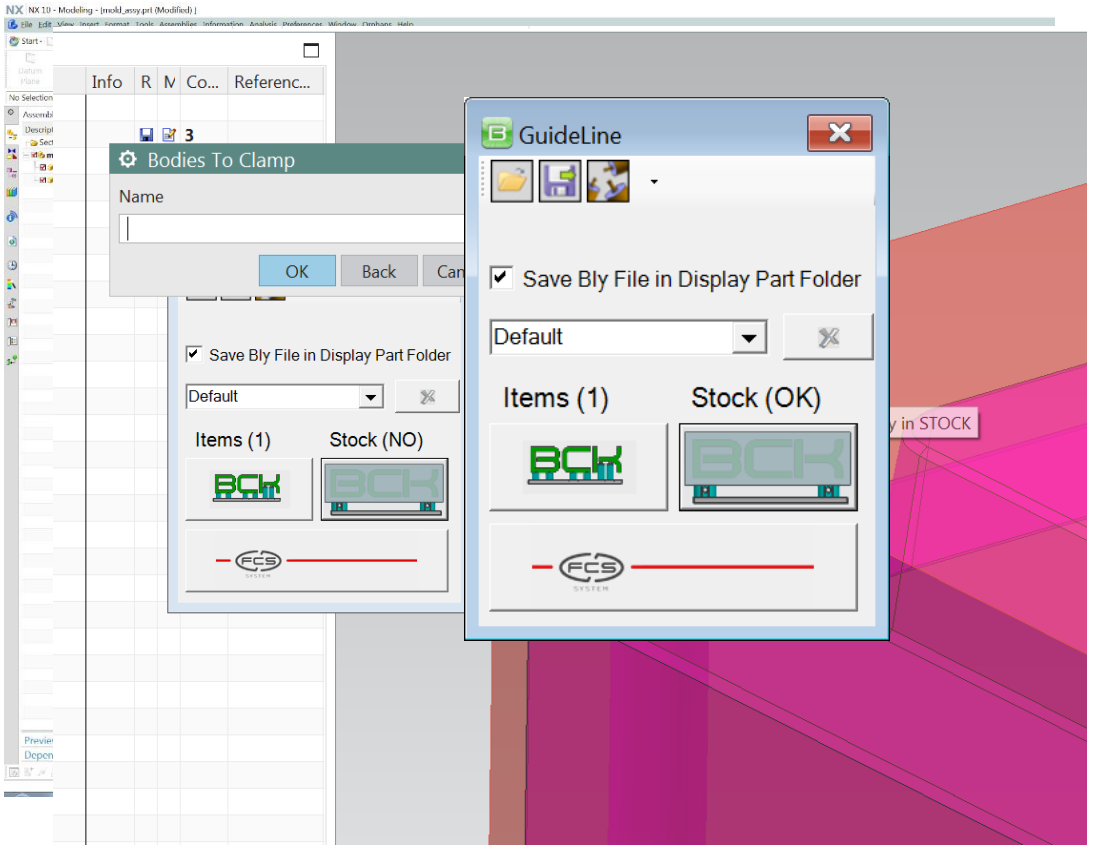
❑ *Stock addition*

Gl Integration supports stock placement too.

Once stock is created in NX (either in the job or in the assembly), press «Select Stock»  button.

You can then select the stock's body.

GL Integration window will change the stock's status to «OK». You can then launch Guideline.



❑ *Stock addition*

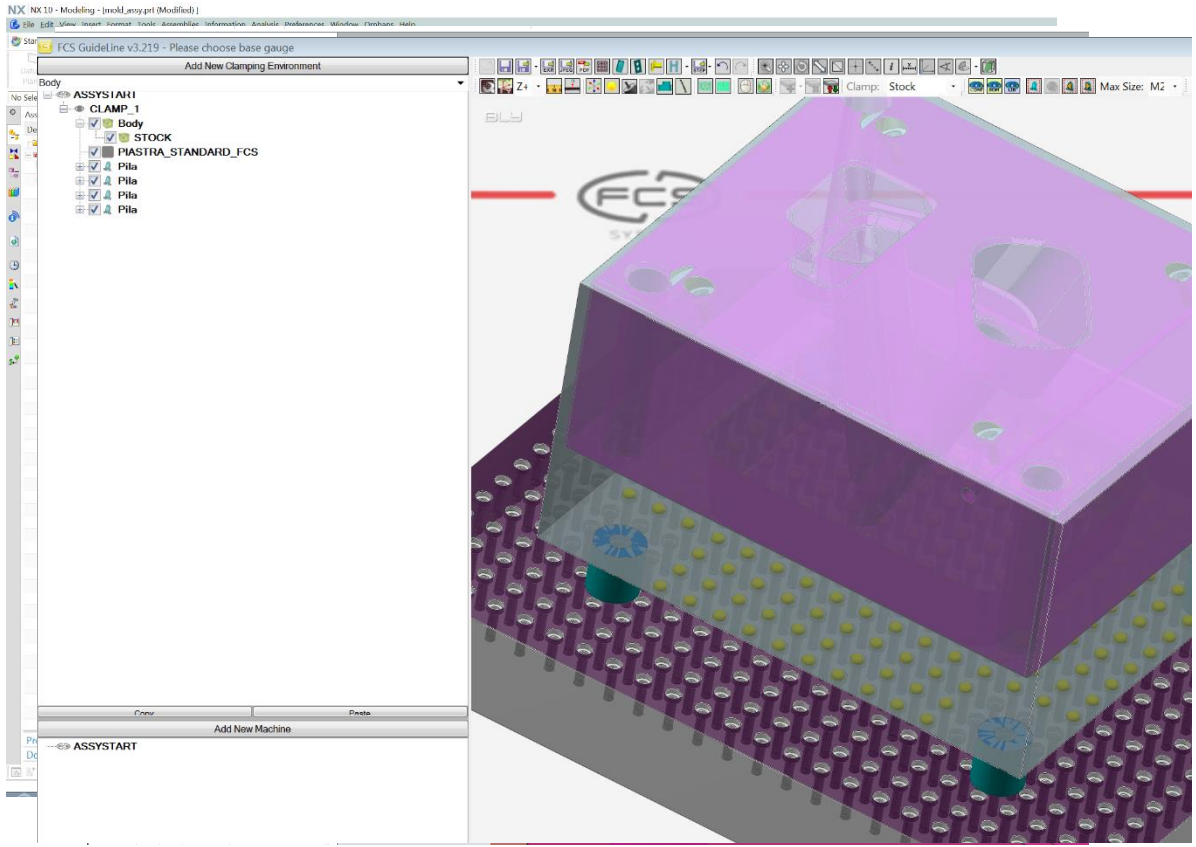
Gl Integration supports stock placement too.

Once stock is created in NX (either in the job or in the assembly), press «Select Stock»  button.

You can then select the stock's body.

GL Integration window will change the stock's status to «OK». You can then launch Guideline.

In Guideline you can clamp as usual, switching from item to stock.



❑ *Stock addition*

Gl Integration supports stock placement too.

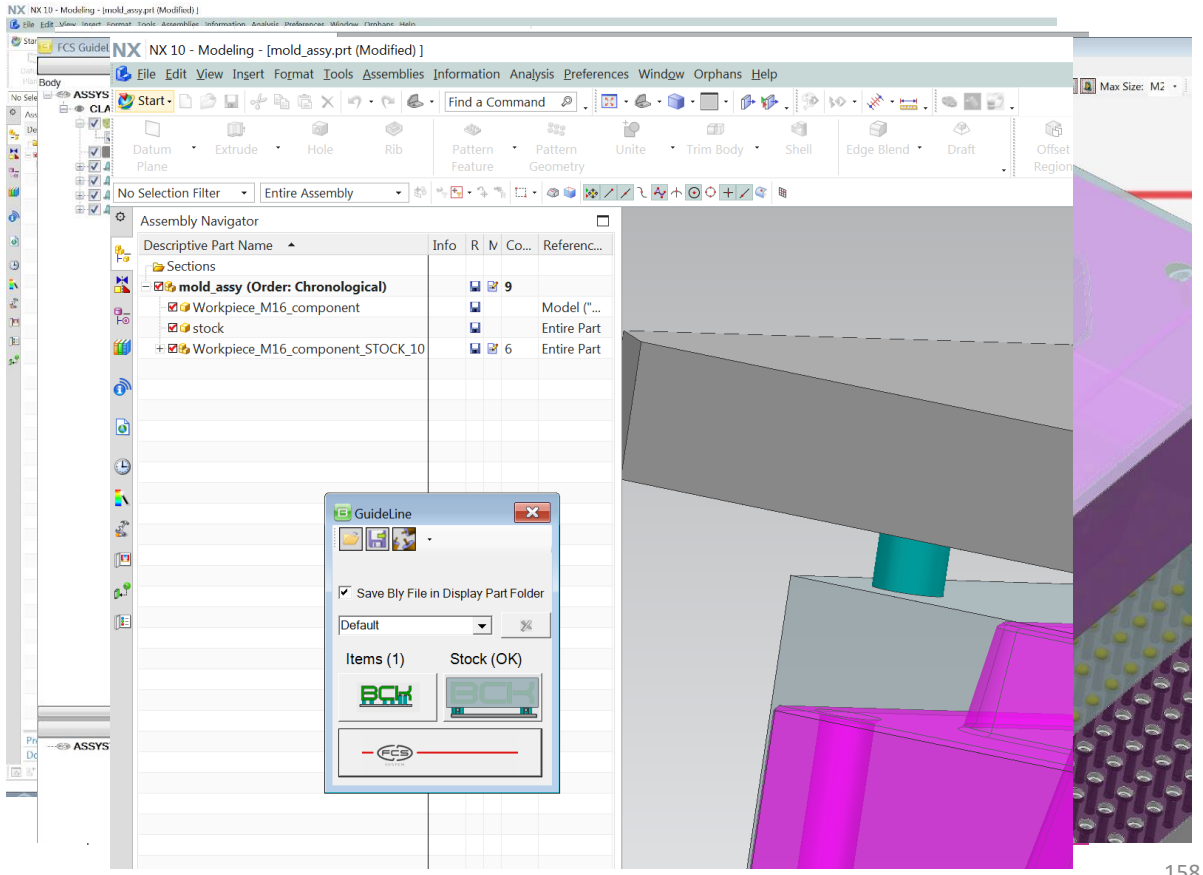
Once stock is created in NX (either in the job or in the assembly), press «Select Stock»  button.

You can then select the stock's body.

GL Integration window will change the stock's status to «OK». You can then launch Guideline.

In Guideline you can clamp as usual, switching from item to stock.

Once done, Save it and clamping components will be transferred to NX as previosly described for the job and ready for seats creation.



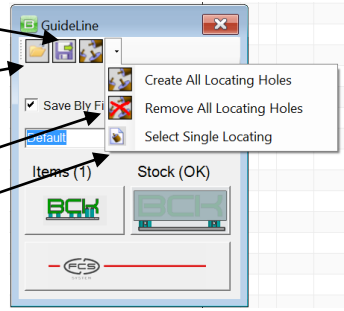
❑ *Other commands*

Save BCK File: saves on the file system the clamping created. It can then be used in other clamping: components will be placed as they are saved and can then be repositioned

Open BCK File: opens a previously saved clampling

Remove all locating holes: deletes the seats created on the job and their features on the model

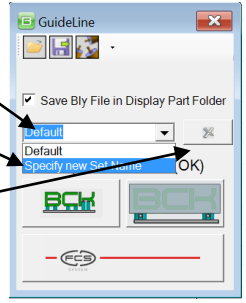
Select single locating: let you select the seats and creates only those ones



Select Reference Set: let you select a reference set to store the clamping components

Specify new Set Name: let you create a new reference set to store the clamping components

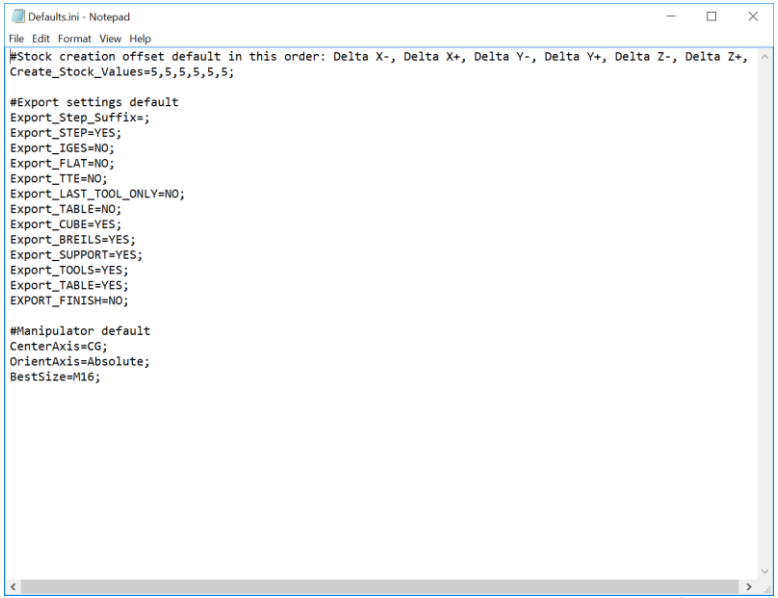
Delete reference set: cancel the selection made, without removing components



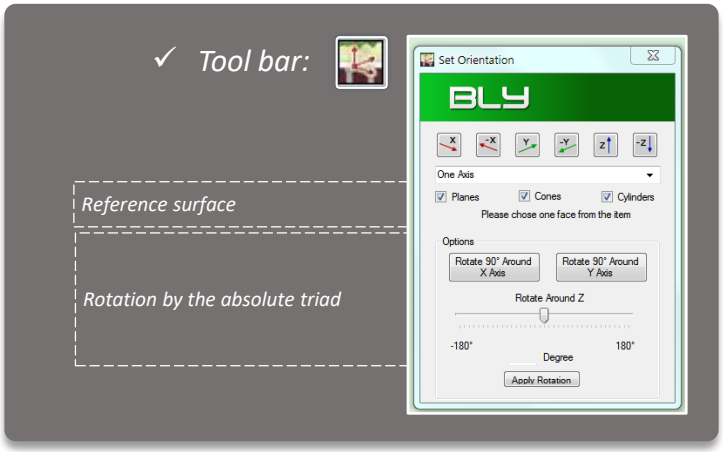
REFERENCE GUIDE ADDITIONAL COMMANDS

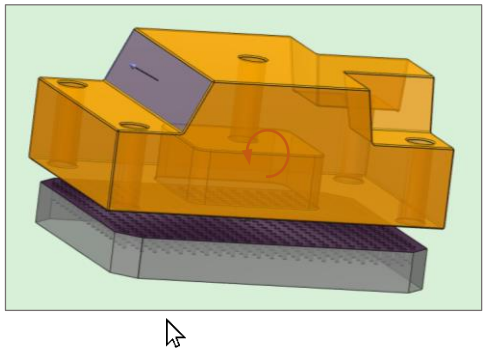
❑ *Default settings*

Some of the clamping parameters can be setup as default via a text file named «Defaults.ini», located in «\FCS\GuideLine3\Misc» directory.
The file can be modified using a text editor (i.e. Notepad), allowing the user to set the most convenient defaults for his daily work: obviously, the values can be modified in any moment during work.
Changes in defaults require to restart Guideline to be effective.
The parameters available should be self-explaining.
Lines beginning with «#» are comments.

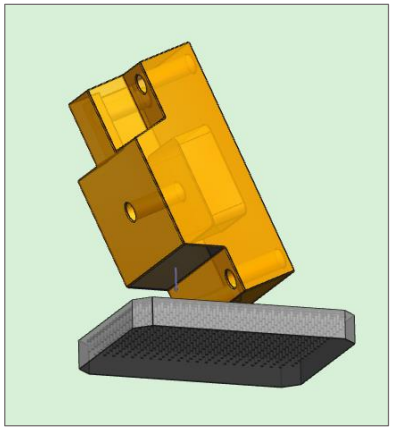


- *Rotation & translation of the workpiece in relation to the base gauge*
  Orient item
  Orientation by any surface of the workpiece





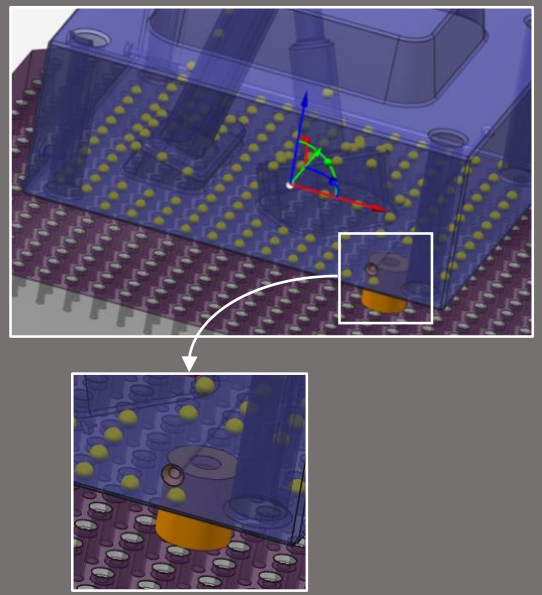
The selected surface…



…is addressed towards the base gauge becoming the clamping surface of the workpiece.

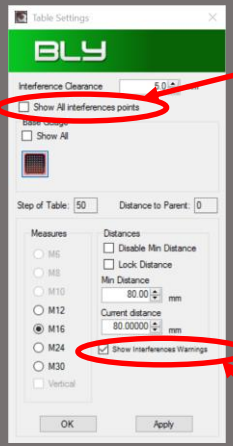
❑ *Table Settings: Interference Clearance*

✓ In case of interference the stack is highlighted in orange



✓ Setting

Toool bar: 



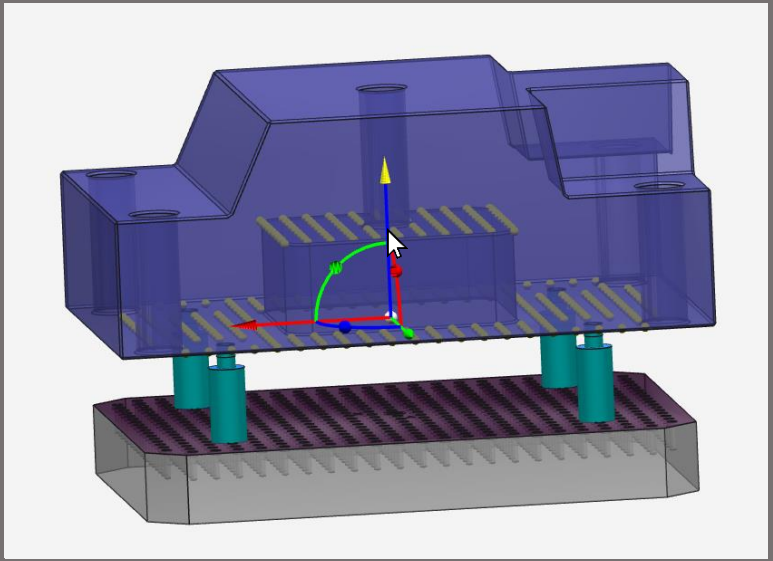
Keyboard: T

Finds and shows all the points violating the clearance, not only the closest

Displays the warning message at bottom

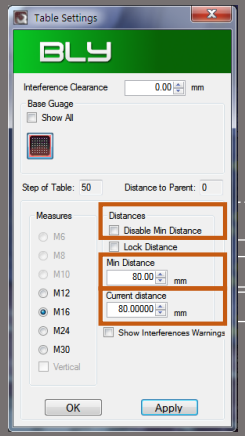
❑ *Table Settings: workpiece placement*

✓ Drag&Drop on the arrow of the vertical axis



✓ Setting

Tool bar: 



Disable the min distance

Minimum distance from the base gauge

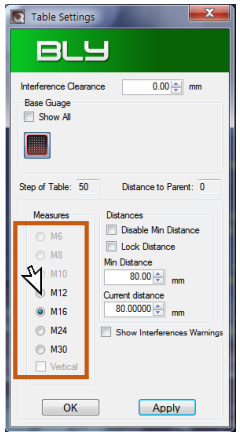
Actual distance

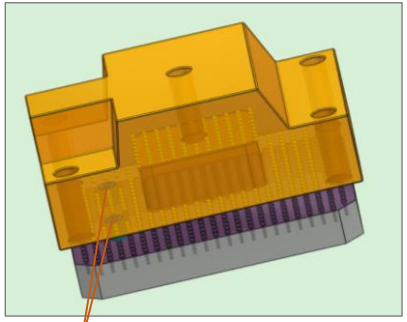
Keyboard: T

REFERENCE GUIDE - ADDITIONAL COMMANDS

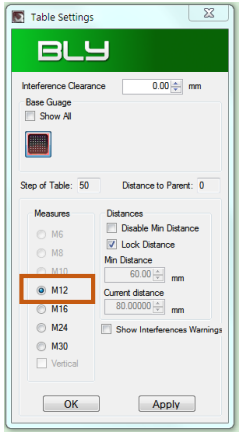
❑ *Table Settings: transition between holes having different size*

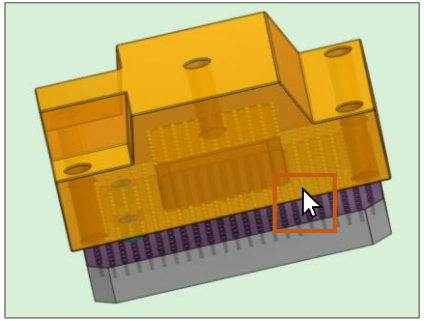
✓ *Tool bar:* 





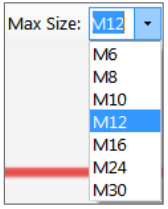
M16 holes





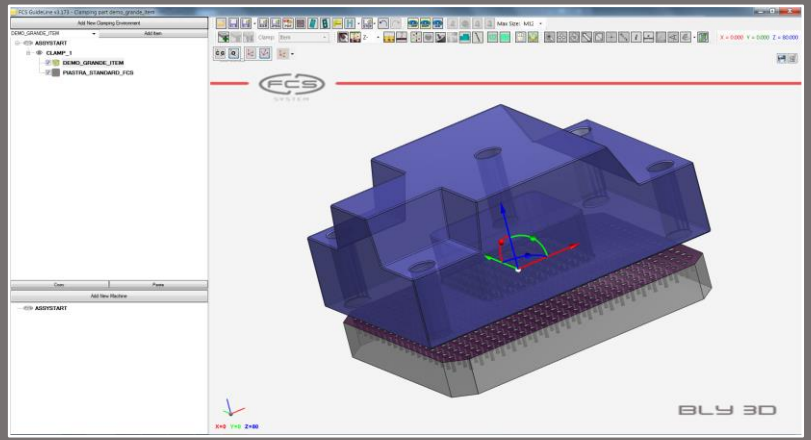
M12 hole made on the workpiece

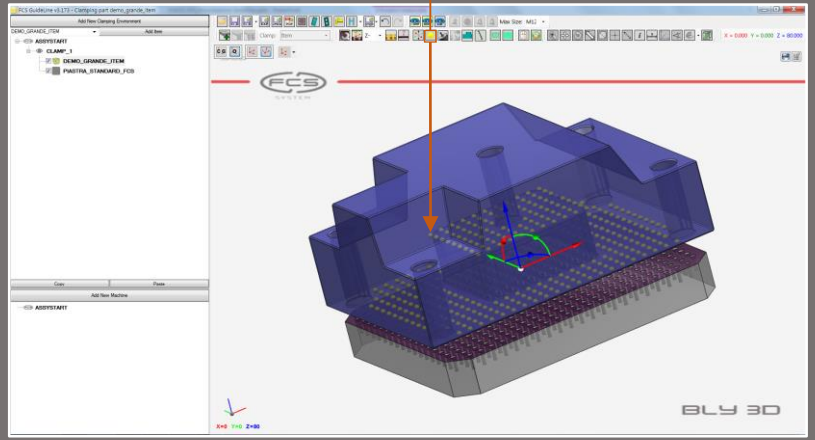
✓ *Keyboard:* 

✓ *Tool bar:* 

❑ *Projection of the clamping points on the workpiece*

✓ *Tool bar:* 

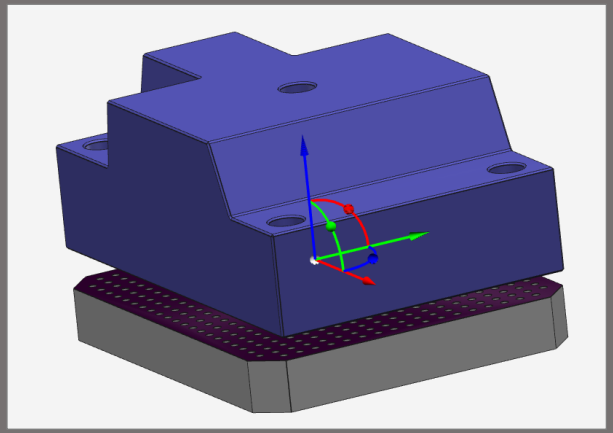




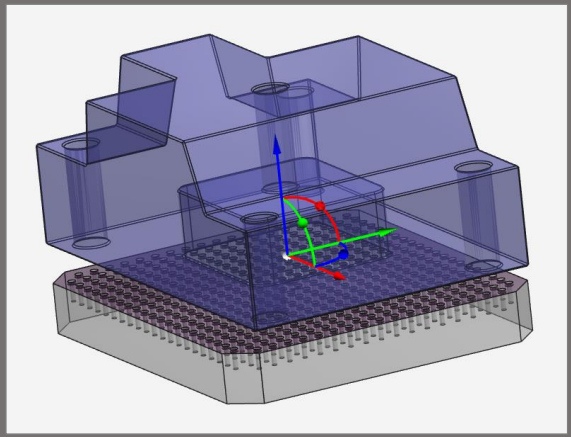
The clamping spots (yellow spheres) are projected on the workpiece clamping surface

❑ Visualization control: Trasparency

✓ Tool bar:  



Reduce the trasparency



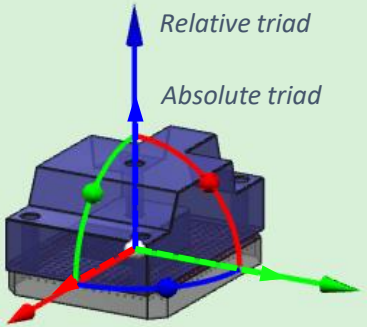
Increase the trasparency

✓ Keyboard: L K

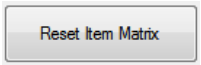
❑ Absolute and relative coordinate systems

By clicking on the workpiece, will appear the relative triad and its specific tool bar (make sure to activate previously the button  *to select the workpiece)*

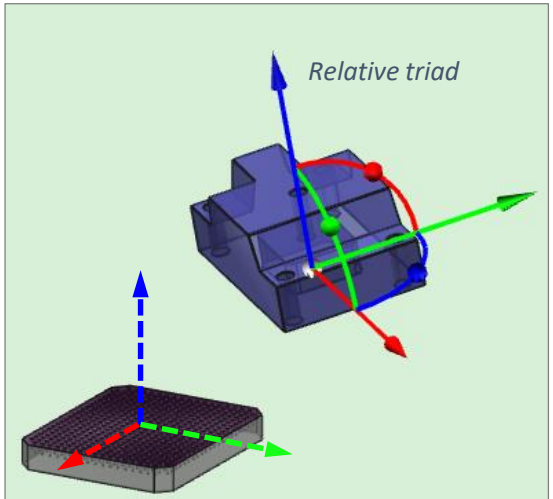
Relative triad=Mobile triad glued on the workpiece
Absolute triad=Fixed triad corresponding to what was defined in the workpiece dwg
(it is not represented in Guideline and it has origin in the base gauge center on its top surface)



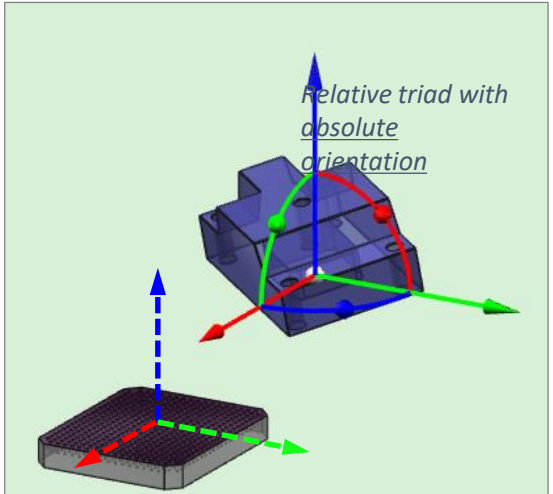
Relative triad corresponding to the absolute triad



** See Rotation & traslation of the workpiece in relation to the base gauge (pag.14)*

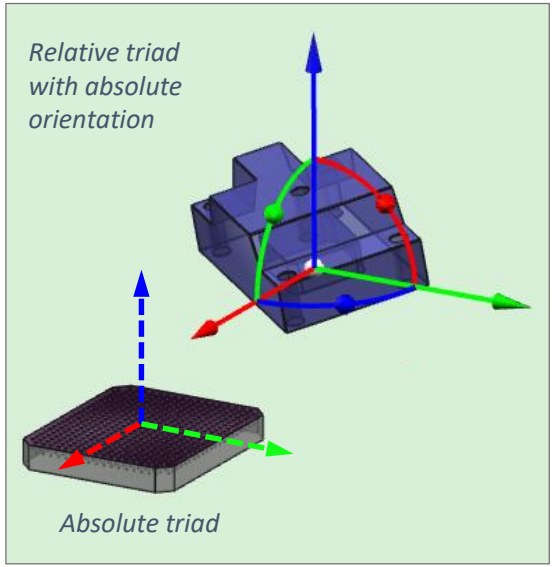


Tool bar: 



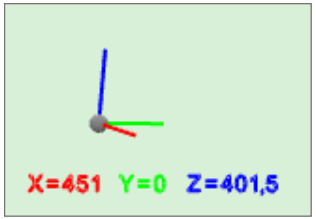
Tool bar: 

❑ Absolute and relative coordinate systems



Tool bar:

Visualization area
At the bottom-left side



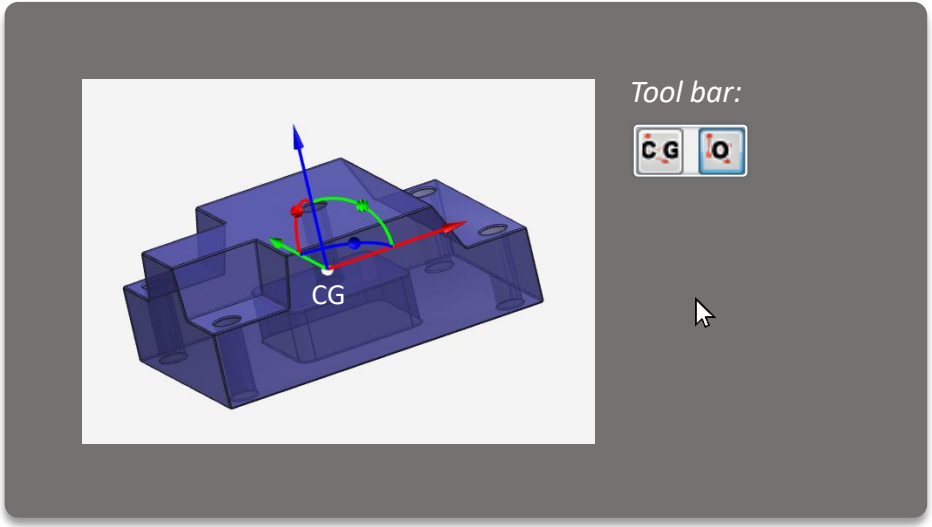
Orientation of the absolute triad

Coordinates of the relative triad origin

✓ Note: the absolute triad is not pointed-out in Guideline. It is just represented here by the dashed line for explanation purpose only.

❑ *Relative triad: origins*

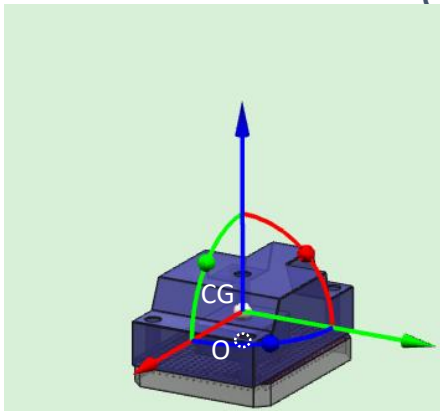
To display the workpiece's relative triad, just click once on the piece itself

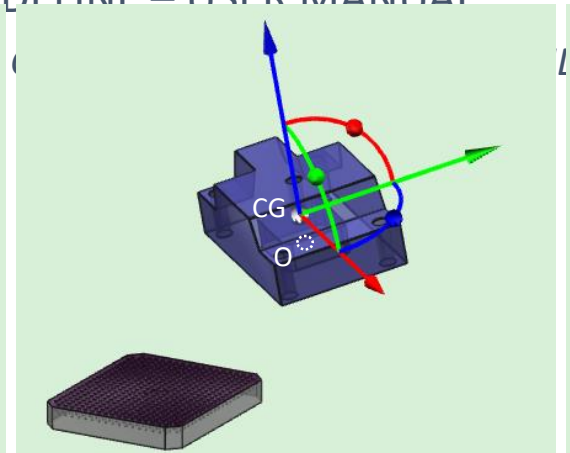


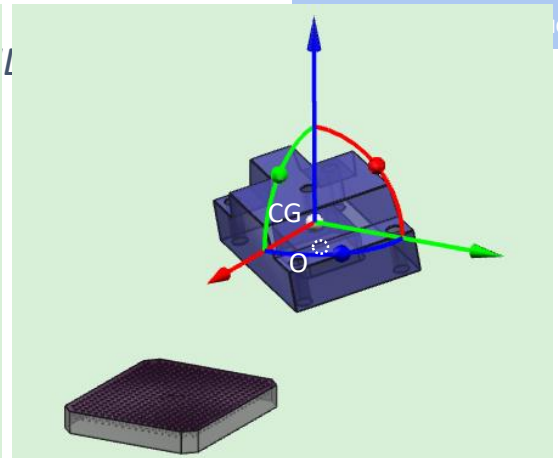
❑ *Relative triad: origins*

Center of Gravity ORIGIN

Tool bar:



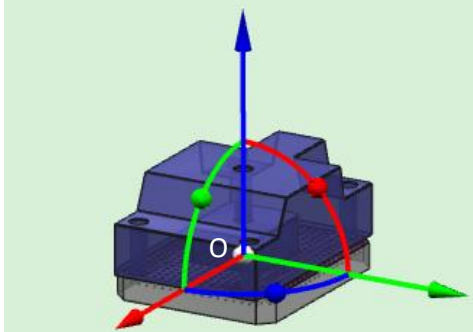




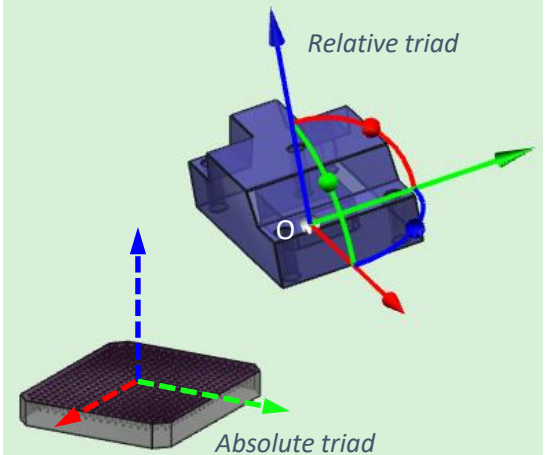
O ORIGIN

Tool bar:

Relative triad corresponding to the absolute triad

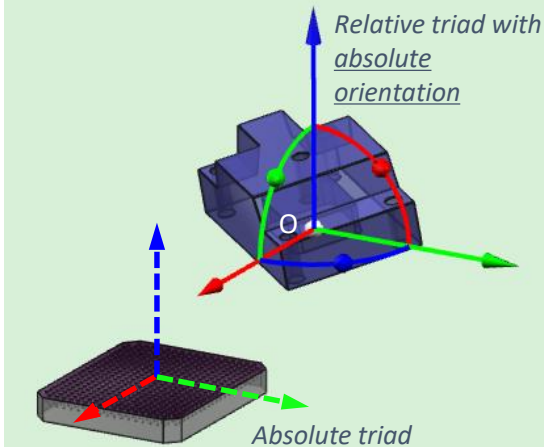


Relative triad



Absolute triad

Relative triad with absolute orientation



Absolute triad

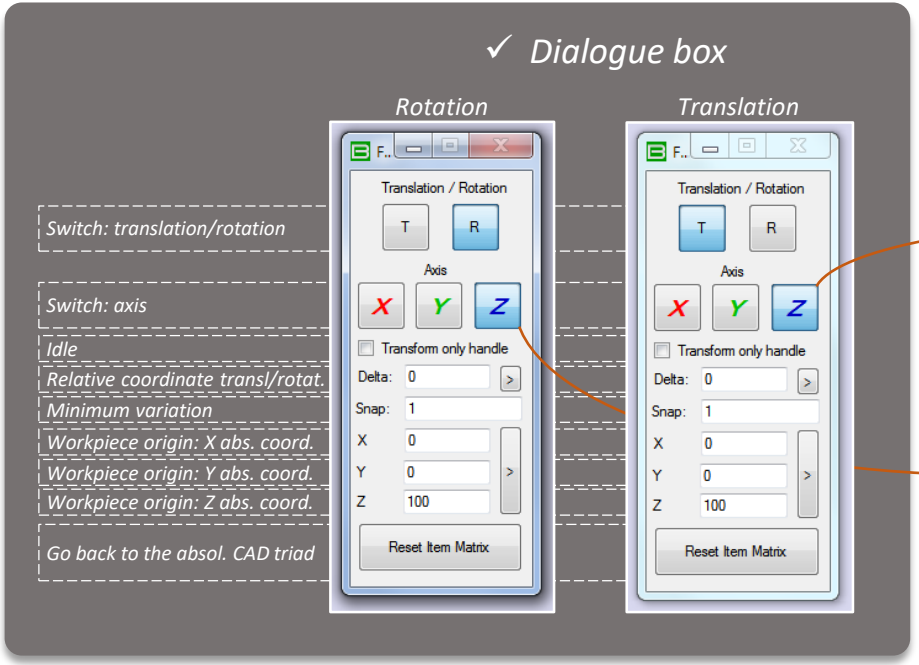
Reset Item Matrix

Tool bar:

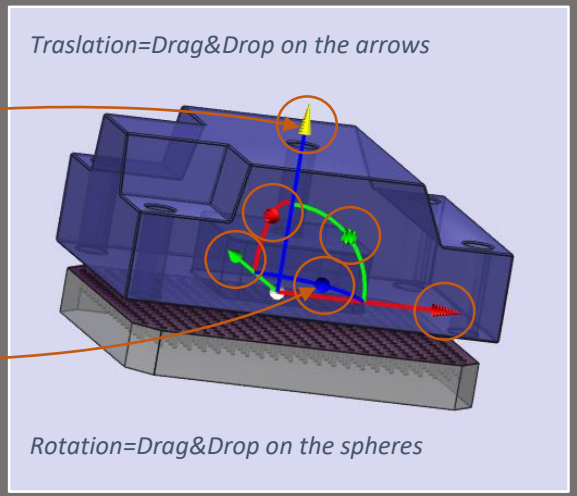
Tool bar:

❑ Rotation & translation of the workpiece in relation to the base gauge

✓ Dialogue box

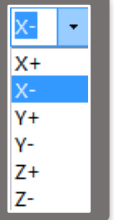


✓ Workpiece (relative triad)

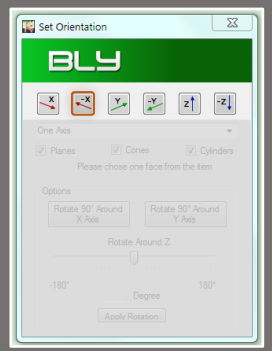


❑ *Rotation & traslation of the workpiece in relation to the base gauge*

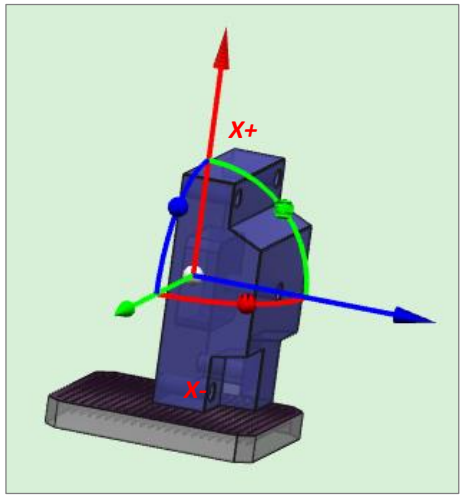
✓ *Tool bar*



✓ *Tool bar*



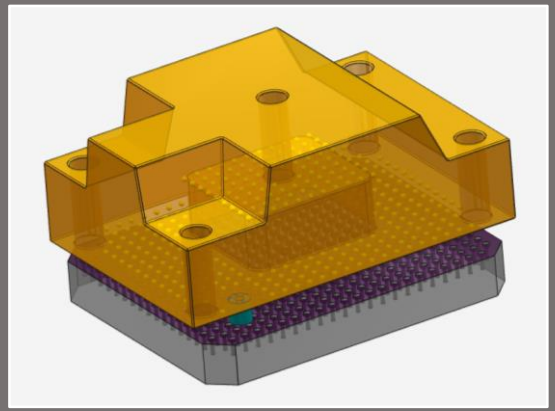
X- becomes the clamping surface



X- side surface of the workpiece is addressed to the base gauge; therefore X- surface becomes the clamping surface.

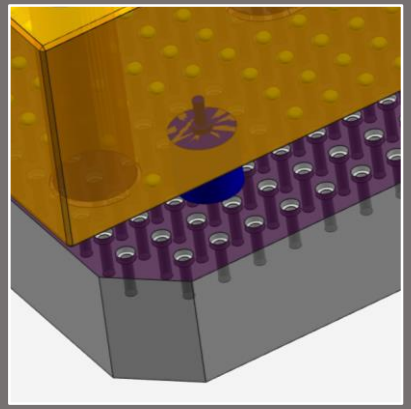
❑ *Generation of the clamping holes*

✓ *Click by the left mouse button on the corresponding projection point*



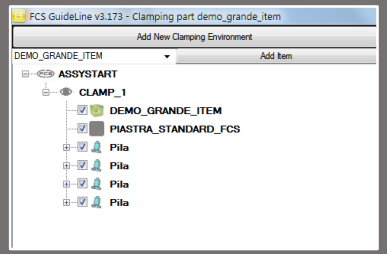
✓ *Visualization of the clamping holes on the workpiece*

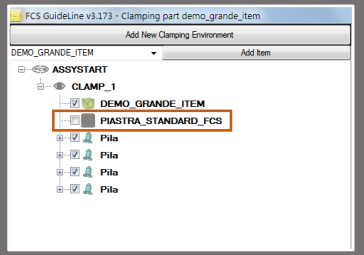
Toool bar: 

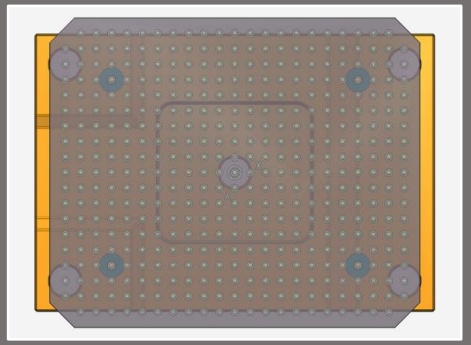


❑ *Management of the components by the treeview structure*

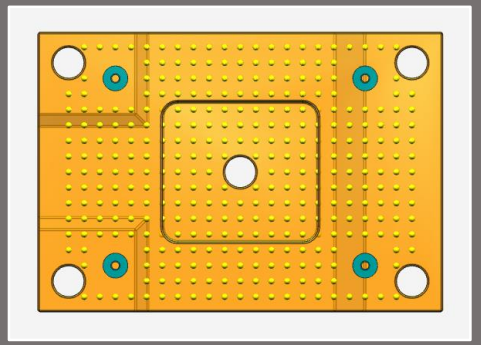
✓ *Flag on the component to show/hide it*







Show (the base gauge)



Hide (the base gauge)

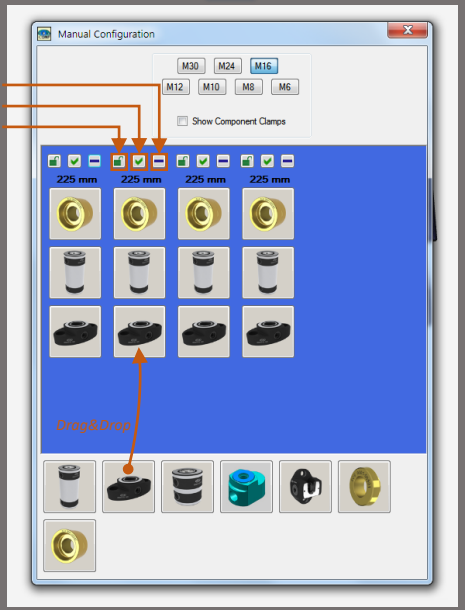
REFERENCE GUIDE - ADDITIONAL COMMANDS

❑ *Library management* *Manual configuration of the stacks* *Bill Of Materials*

✓ *Tool bar:* 

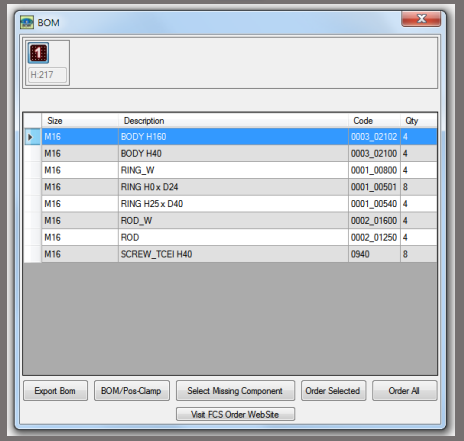
The softwares uses only the selected components to build the stack

✓ *Tool bar:* 



The components of the stack are manually forced

✓ *Toool bar:* 

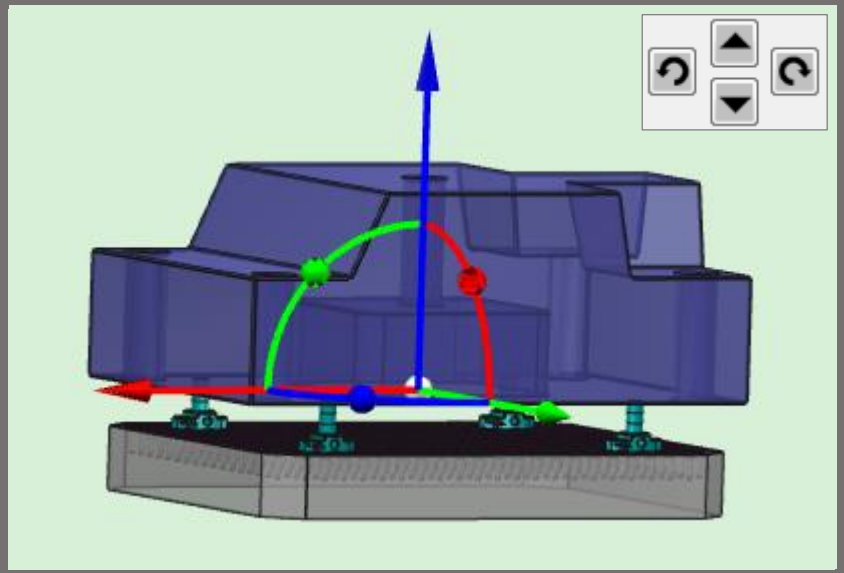


List of the components used to clamp the workpiece

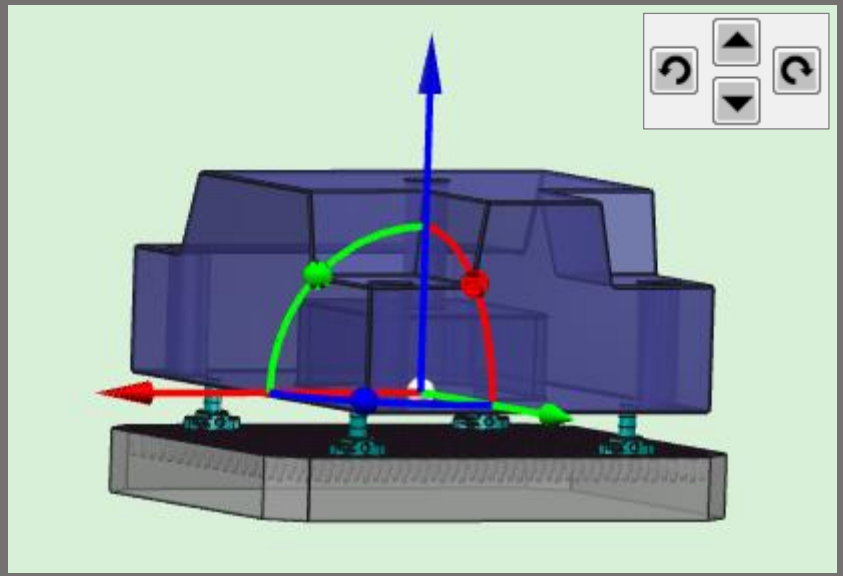
✓ *Keyboard:* X M B

❑ *Variation of the clamping position in CAM*

Minimum variation of the clamping height

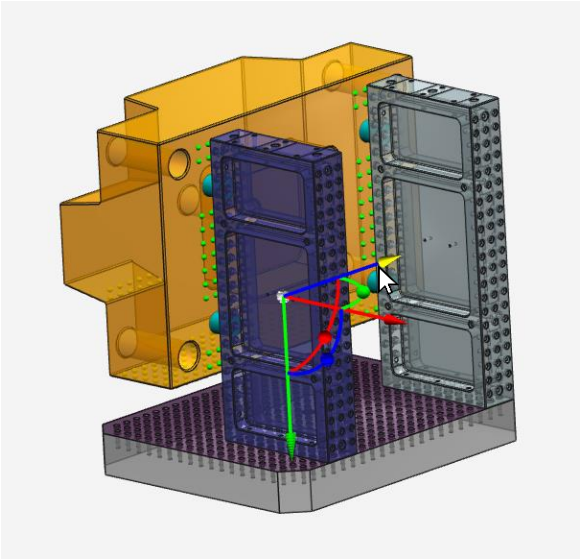


90° rotation

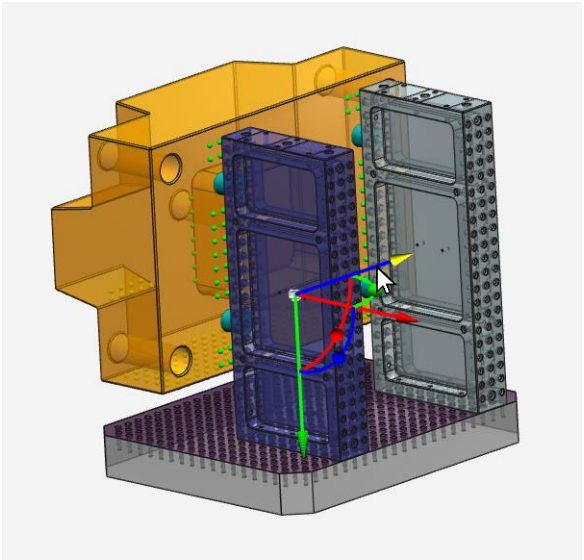


❑ *Variation of the clamping position in CAM*

Tool bar: 



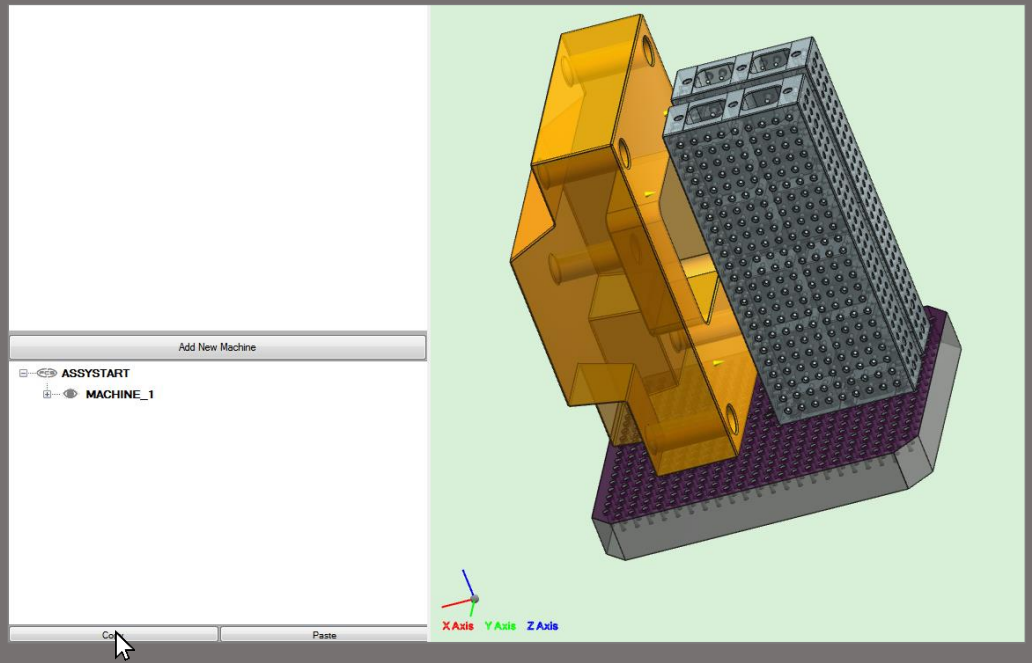
The workpiece and the shoulders are shifted together (as they were a single block)



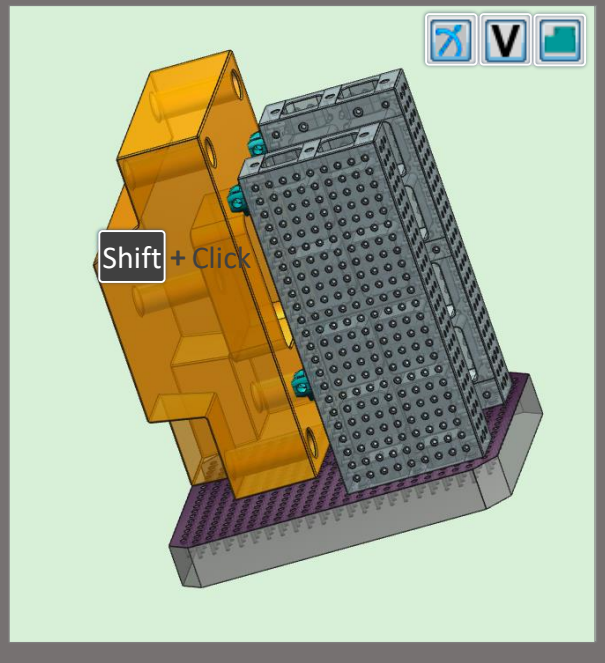
The shoulder is shifted only

❑ *The specific functions of the Machine Environment (CAM)*

❑ *Additional clamping components - copy & paste*



❑ *Body Vertical application*



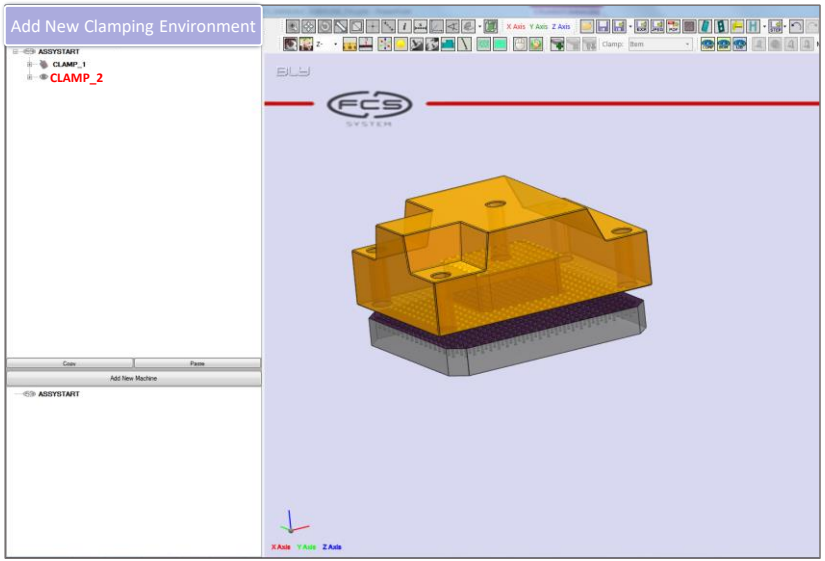
❑ *Obtain the workpiece geometry information*

Tool bar:

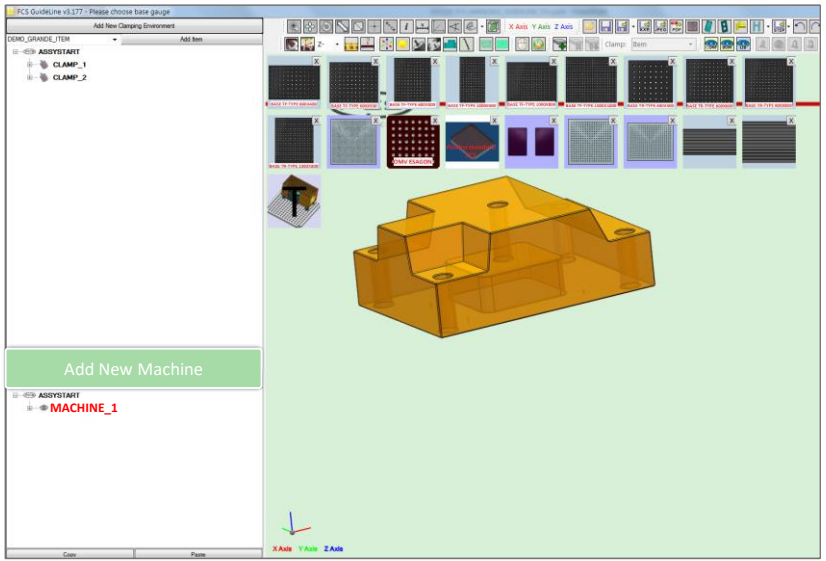
- ✓ *Absolute coordinate* 
- ✓ *Distance between two points* 
- ✓ *Distance information* 
- ✓ *Axis* 
- ✓ *Angle* 

❑ *A new clamping session generation*

CAD



CAM



To skip from one to another session, click on the treview or press the corresponding buttom on the keyboard:

1 2 n

❑ *Export and save*

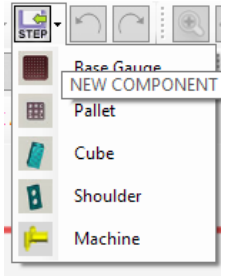
Tool bar:

- ✓ *Save* 
- ✓ *Save as* 
- ✓ *Export to STEP/IGES* 
- ✓ *Save JPEG* 
- ✓ *Export the clamping instructions* 

❑ *Adding a new component in the library*

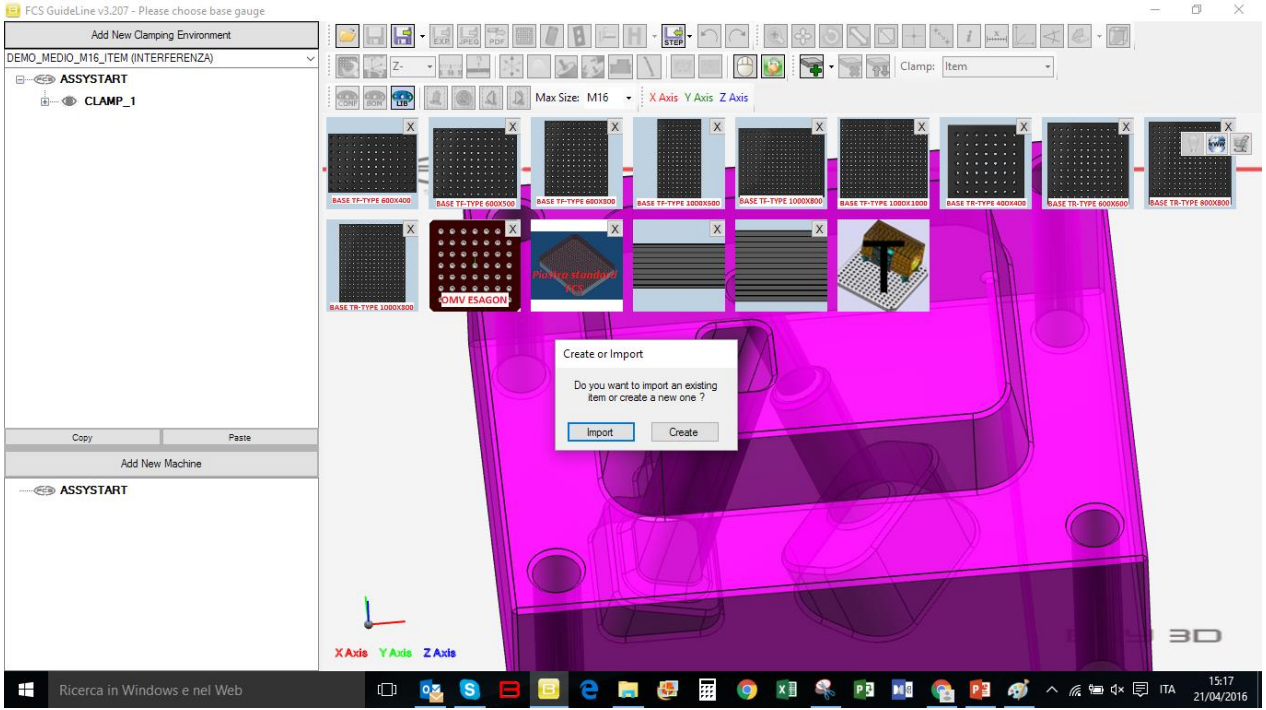
In order to allow the users to define a clamping using custom components, Guideline gives the possibility to add them to the standard library.

To do so, just press



in the toolbar and select the type of component you want to add (Base Gauge in our example).

N.B: only for Base Gauge, the software will ask you to Import or Create a new component. All the other categories can be imported only.

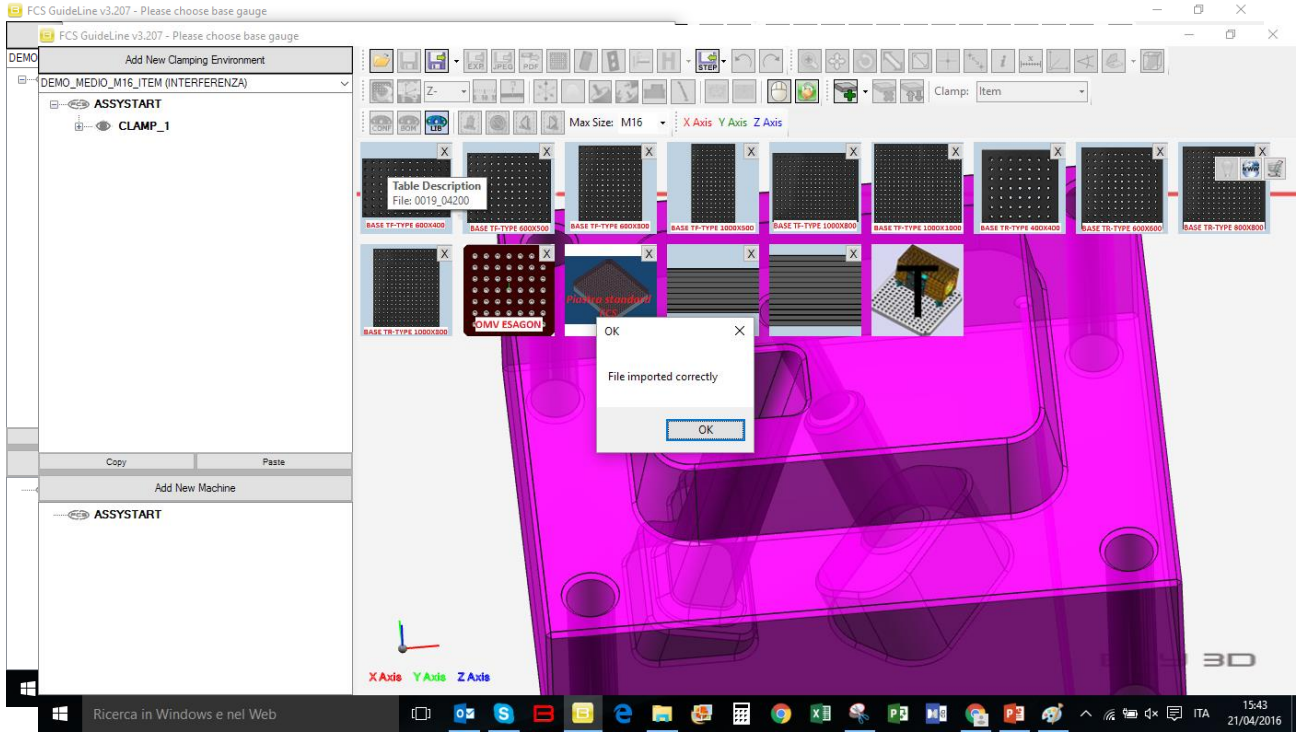


❑ *Adding a new component in the library: import the component*

If you choose Import, the system will ask for a STEP file to be added to the library.

N.B: only STEP files are accepted.

A confirmation message is then displayed.



❑ *Adding a new component in the library: import the component*

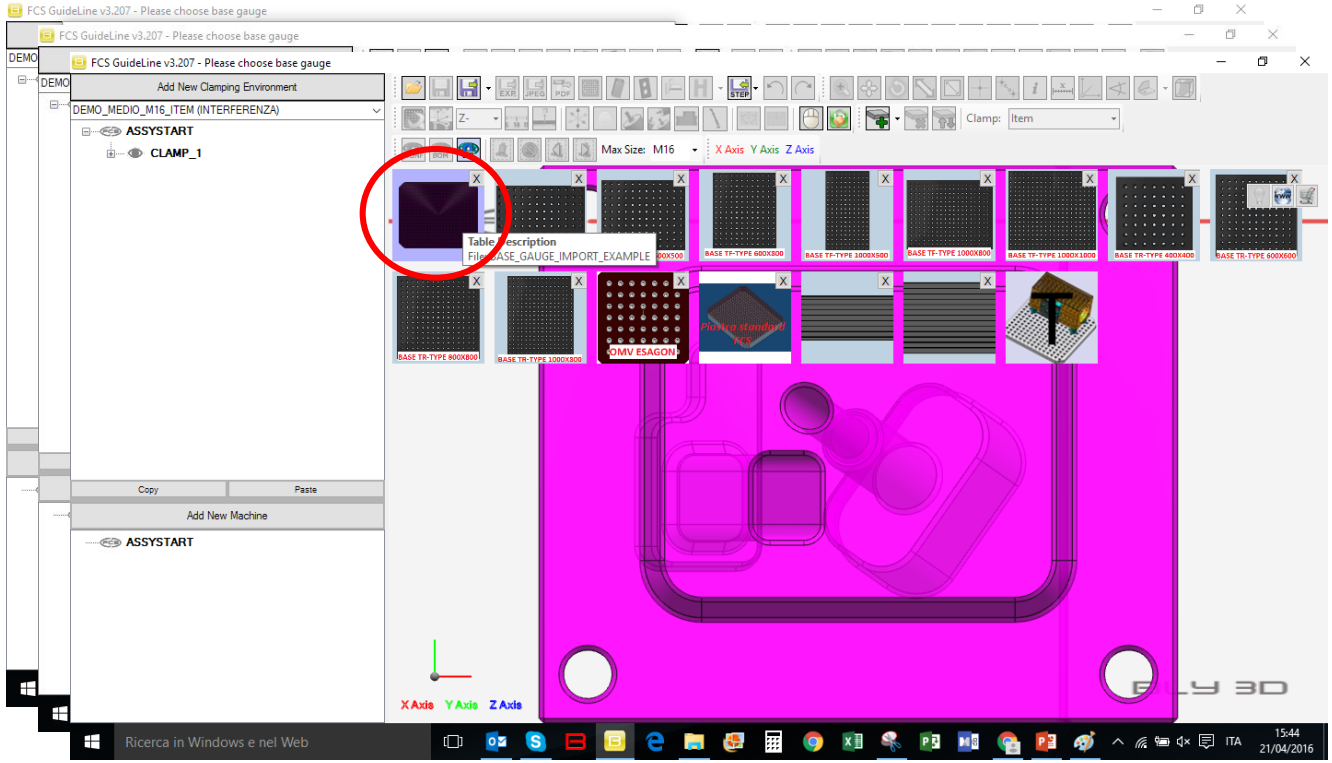
If you choose Import, the system will ask for a STEP file to be added to the library.

N.B: only STEP files are accepted.

A confirmation message is then displayed.

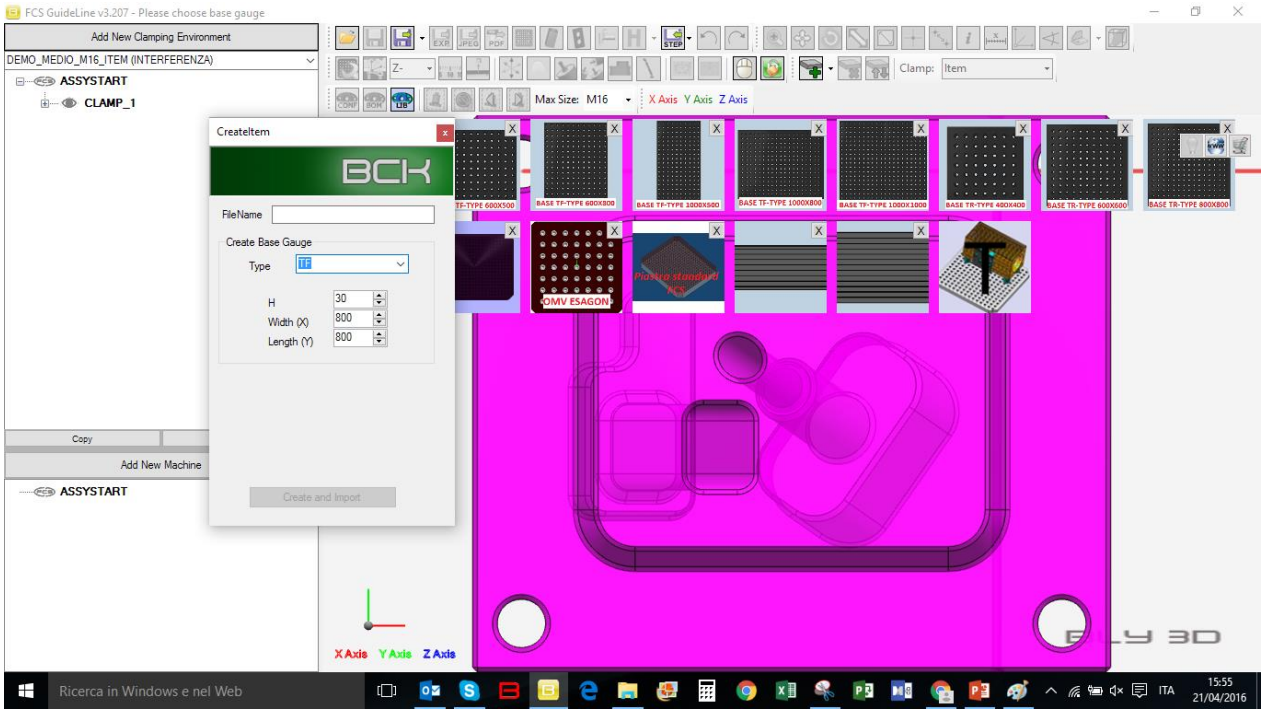
To see it in the library, open a new file (or re-open the piece you were working on) and it will be displayed in the available gauges.

To remove it, just press the X in the top-right corner of the icon.



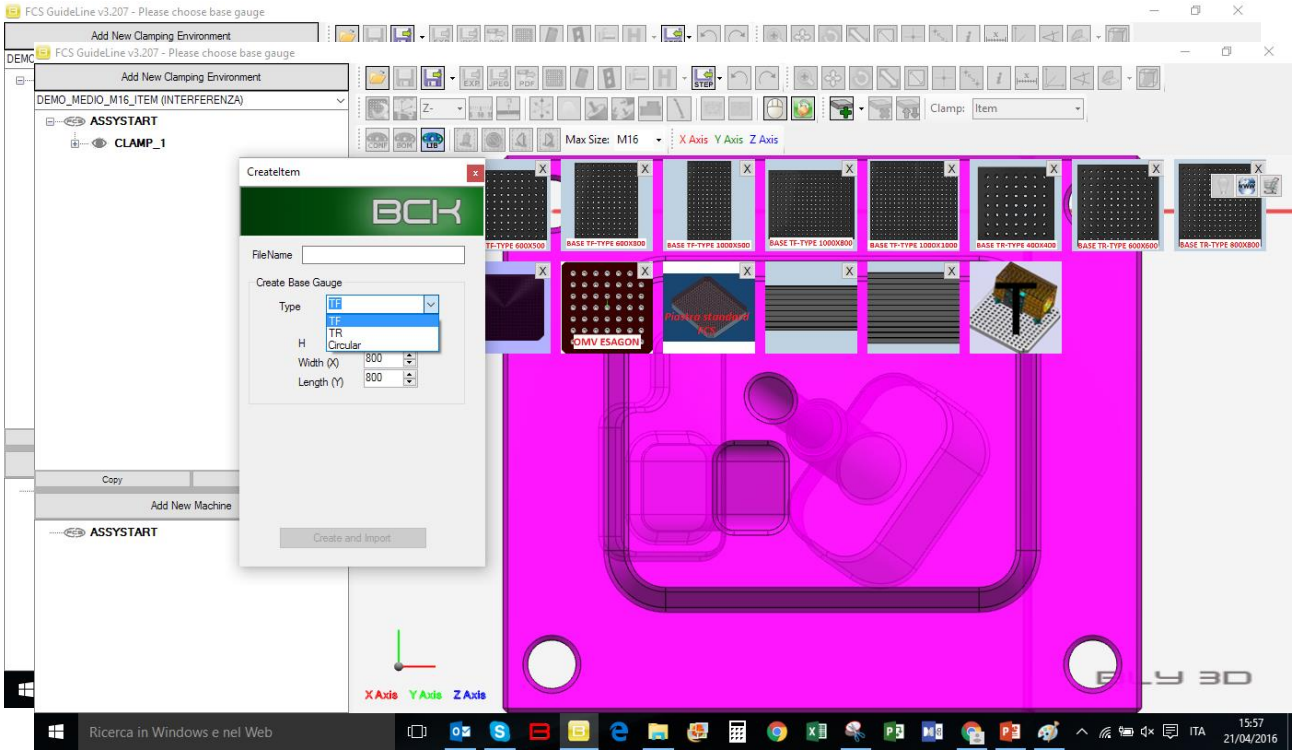
❑ *Adding a new component in the library: create the Base Gauge*

If you choose Create, the system will show a window where you can specify the characteristics and the name of the plate you want to add.



❑ *Adding a new component in the library: create the Base Gauge*

If you choose Create, the system will show a window where you can specify the characteristics and the name of the plate you want to add.

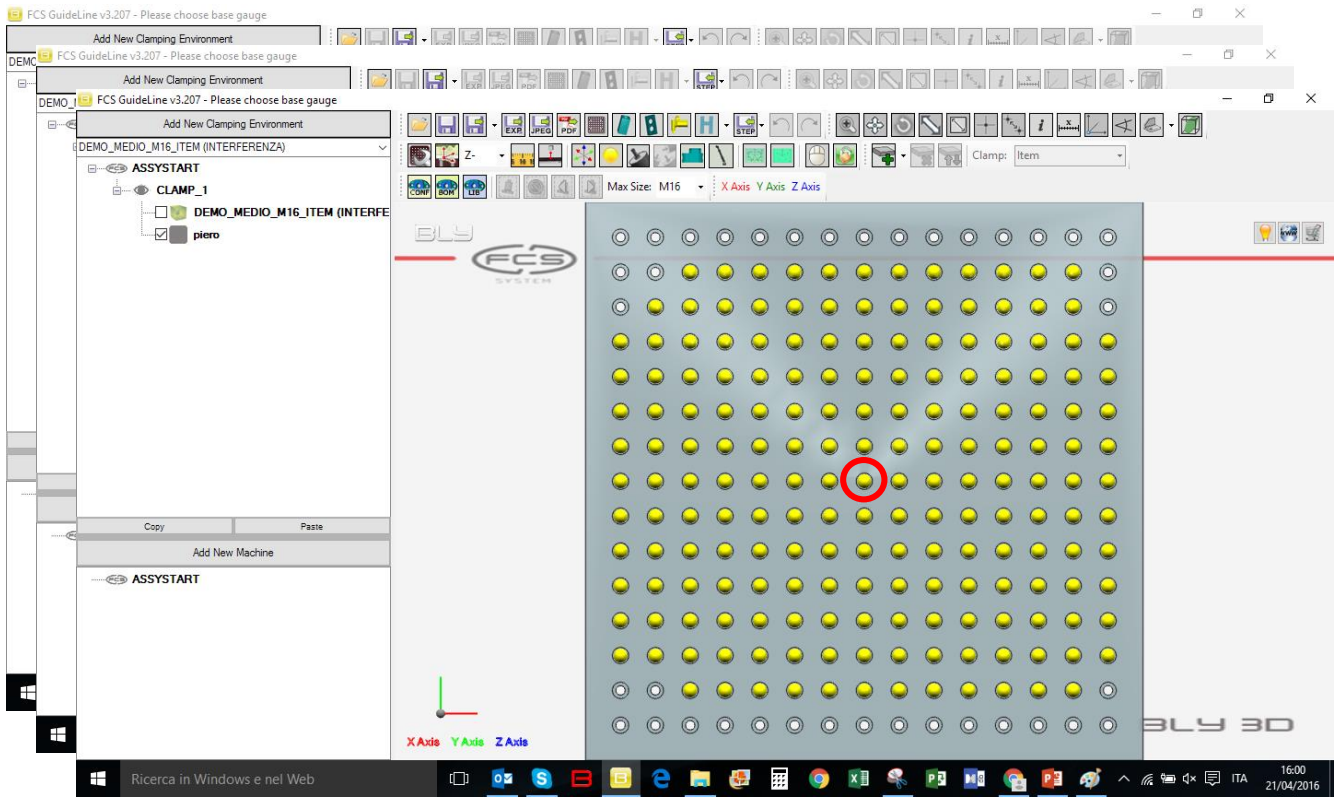


❑ *Adding a new component in the library: create the Base Gauge*

If you choose Create, the system will show a window where you can specify the characteristics and the name of the plate you want to add.

Dropdown menu options are:

- TF: the plate will have a placement hole in the center;

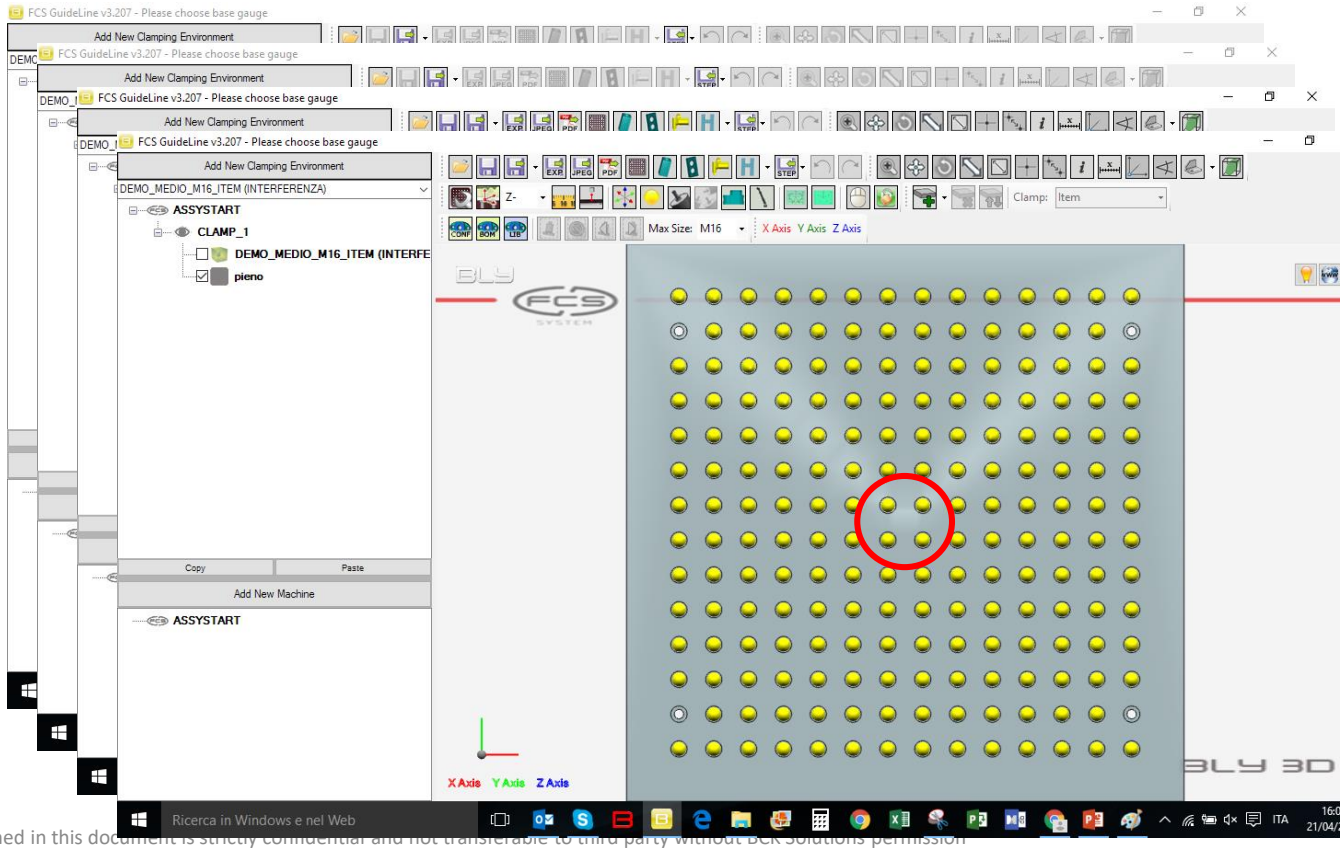


❑ *Adding a new component in the library: create the Base Gauge*

If you choose Create, the system will show a window where you can specify the characteristics and the name of the plate you want to add.

Dropdown menu options are:

- TF: the plate will have a placement hole in the center;
- TR: the plate will have the center massive, surronded by four placement holes;



❑ *Adding a new component in the library: create the Base Gauge*

If you choose Create, the system will show a window where you can specify the characteristics and the name of the plate you want to add.

Dropdown menu options are:

- TF: the plate will have a placement hole in the center;
- TR: the plate will have the center massive, surronded by four placement holes;
- Circular: creates a circular plate.