

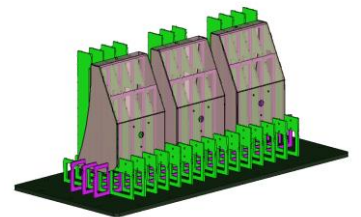


NucleoTM
Fixture

Getting Started Guide

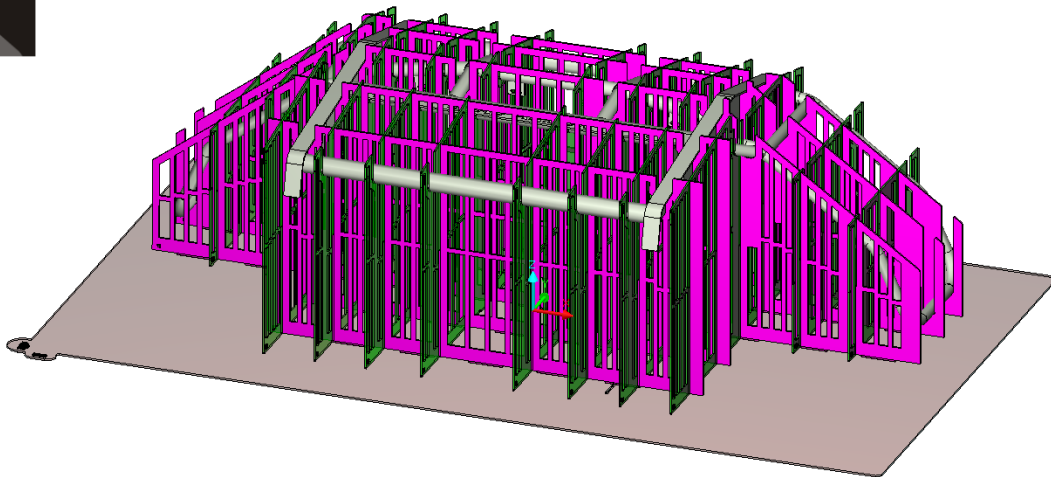


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(Revision 2016/06)



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Introduction

NUCLEO Fixture Design is a module built on top of the OPTICAM Classic CAD/CAM system, designed specifically for the rapid design of part fixtures for the general Welding and Fabrication industries. OPTICAM Classic has been under constant development since 1983 under several brand names; over 40,000 licenses have been issued Worldwide.

NUCLEO Fixture provides an automated solution for the creation of inexpensive part fixtures formed from interlocking flat sheet material profiles. Whether the component is an enclosure, an assembly of tubes or a family of similar parts, NUCLEO Fixture drastically reduces design, complexities, time and cost consumed by traditional fixture design methods.

How It Works

NUCLEO Fixture creates an interlocked grid of X and Y fixture blades connected to a mating base plate from an imported solid CAD model. The top profile of each X or Y blade matches the underside of the component at the blade's insertion position, creating a "cradle" into which the component is located. NUCLEO Fixture includes a range of standard CAD translators and optional translators for most popular CAD systems. These include IGES, Solidworks, SolidEdge, IronCAD, Inventor CATIA, Pro Engineer, STEP and Unigraphics.

Fixture Design Basics

The number of X and Y blades are entered and shown on screen; the user can determine to use all or some of the positions or edit them (move, extend etc.). Tabs are created on the base of each blade to mate to the base plate; these may also include a clip feature allowing the blades to grip to the base plate (which will accordingly be elevated on risers).

Once the details are entered, NUCLEO Fixture calculates and annotates each blade so that they are easily referenced to the base plate (for easy assembly). Finally, the blades and base plate are saved in DXF format and transferred to a suitable CAM system for nesting and profiling or directly to a laser/waterjet machine.

Clamps

Optionally, NUCLEO Fixture allows the importation of standard clamp solids files (available from most major clamp manufacturers via the internet). When included in the model for fixturing, these can be inserted into the fixture assembly. An additional "clamp rail" can also be defined for the X and Y direction; this is clipped to the side of the blades in an XY plane orientation and clamps can be attached at the manufacturing stage onto the rail.

Offsetting

NUCLEO Fixture supports component offsetting enabling the profile of each blade to be offset by a user-definable amount. "Gripper Points" can be included also; these are small semi-circles at set intervals along the blade profile. As needed, these can be filed or ground down if the actual component does not adequately match the computer model.

Weight Reduction

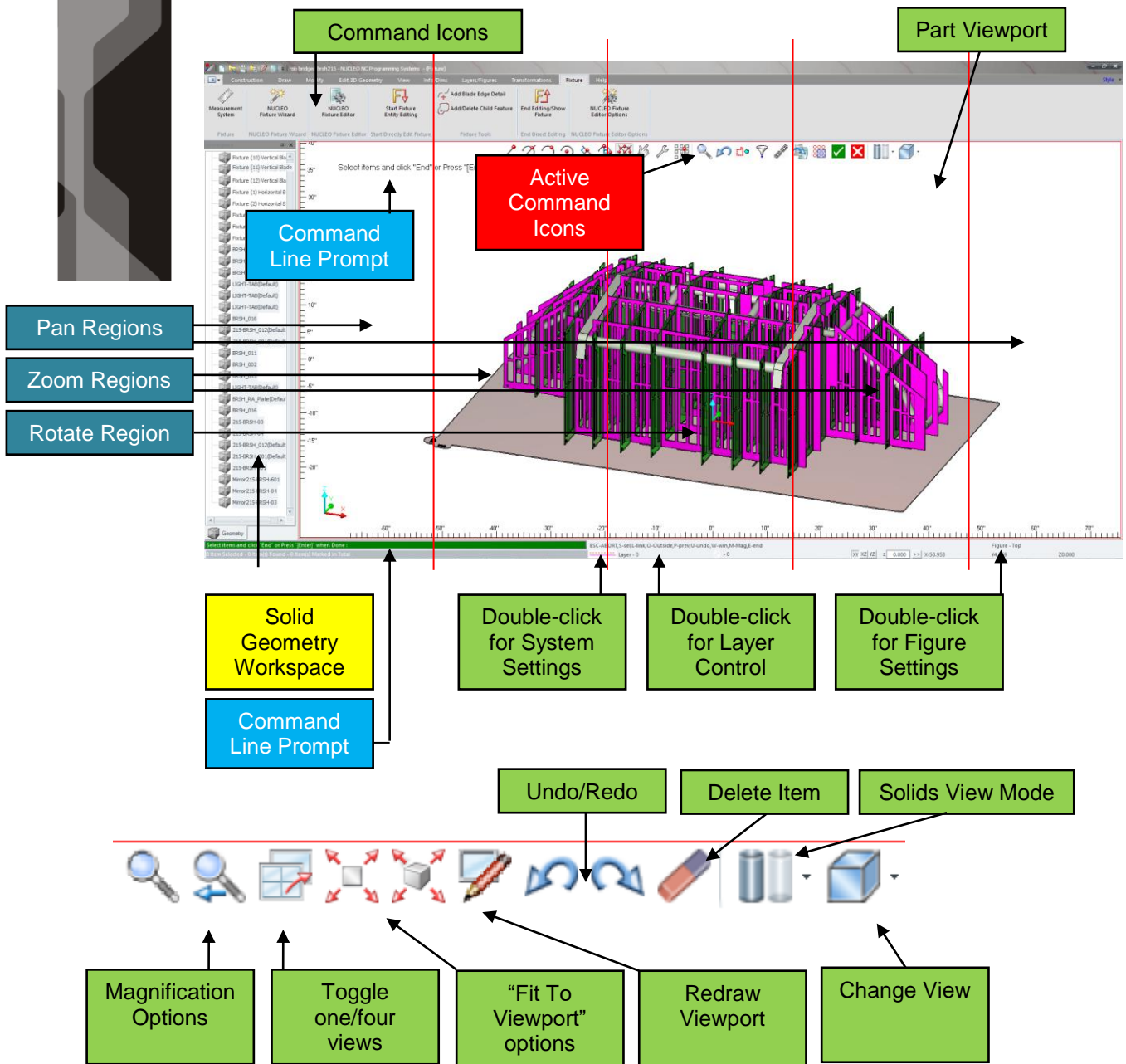
For large fixtures, NUCLEO Fixture includes an option to create internal cut-out features inside each blade to reduce the weight of the overall fixture assembly.

Storage

Unlike traditional fixtures, when using the base tab clip mechanism, NUCLEO Fixture enables the fixture assembly to be dismantled when not needed and laid flat - significantly reducing storage requirements and therefore precious square footage.

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NUCLEO Fixture User Interface



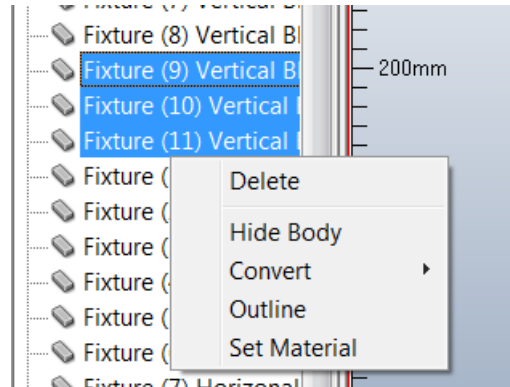
Notes

From time to time, depending on the function being used, the **Command Line prompt** will appear (this will appear as a white or green prompt line at the bottom of the screen AND in the top left side of the graphics viewport). The command Line Prompt will wait for user keyboard input or the **Active Command Icons** will change accordingly, so the user can select a sub command or choose a particular option. In addition, the right mouse click will show the same icons. When the Command Line Prompt is idle (not in use) the icons will appear as above (Magnification/View options etc.).

It is possible to zoom/rotate/pan the Part Viewport at any time by simply dragging the mouse pointer in the Viewport with the left key pressed; depending on the initial location of the pointer, the effect will change from **panning** (far left/right of the Viewport), **zoom** (more centralized) and **rotate** (in the center). In addition, when a dialog box is shown on screen it is also possible to manipulate the view in the same manner; when a command line prompt is

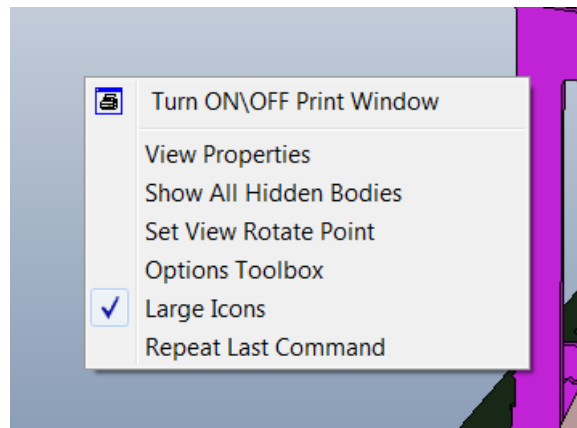
active, **pressing the SHIFT down key along with the same drag and click with the mouse**, enables the user to change the view mid-command.

The **Solid Geometry Workspace** lists all the solid entities within the Part. From this Workspace, multiple entries can be selected with the SHIFT or CTRL key - then with a right-mouse click, a context menu appears. From this menu, the solids can be hidden or their attributes changed (for example, the transparency):



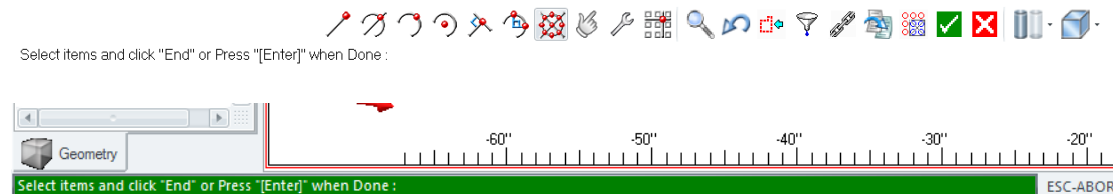
When the command line prompt is idle, it is possible to show all hidden solids by right-mouse clicking anywhere in the Viewport and selecting "Show All Hidden Bodies":

Also, one entry in the above context menu worthy of extra description is the "Set View Rotate Point"; this allows the user to reset the point of rotation – it may be that the part has been zoomed in and the rotation point for viewing is somewhere outside of the current view; by selecting this option the user can reset the rotation point to somewhere on the part by simply clicking a coordinate, end point or other.



Picking Positions

When the command line is active, the Active Command Icons will normally change:



The leftmost ten icons are the Snap modes for picking positions on the part within the Viewport. The default is normally set to "Automatic" (the seventh icon) but another snap mode can be forced by pressing the necessary icon. The snap modes are apparent from the tool tip that will be shown as the mouse pointer is floated over the icon. If the right mouse button is clicked anywhere in the Viewport during a command, the same icons are replicated on a context menu.

To the right of the Snap mode icons are the sub command icons for the active command. The green Check completes the command whereas the red cross cancels the command (the keyboard equivalents to these are ENTER and ESC respectively). Again, pressing the SHIFT key within a Command Line Prompt enables manipulation of the Viewport with the mouse as previously described.

TIP:

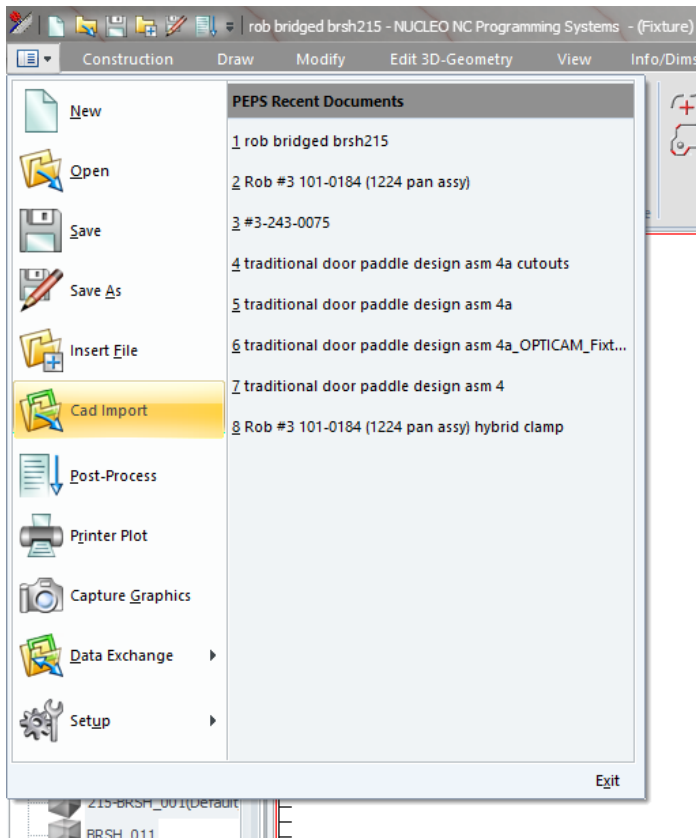


The snap modes set in the ribbon bar are "modal", in that they state the active snap state.

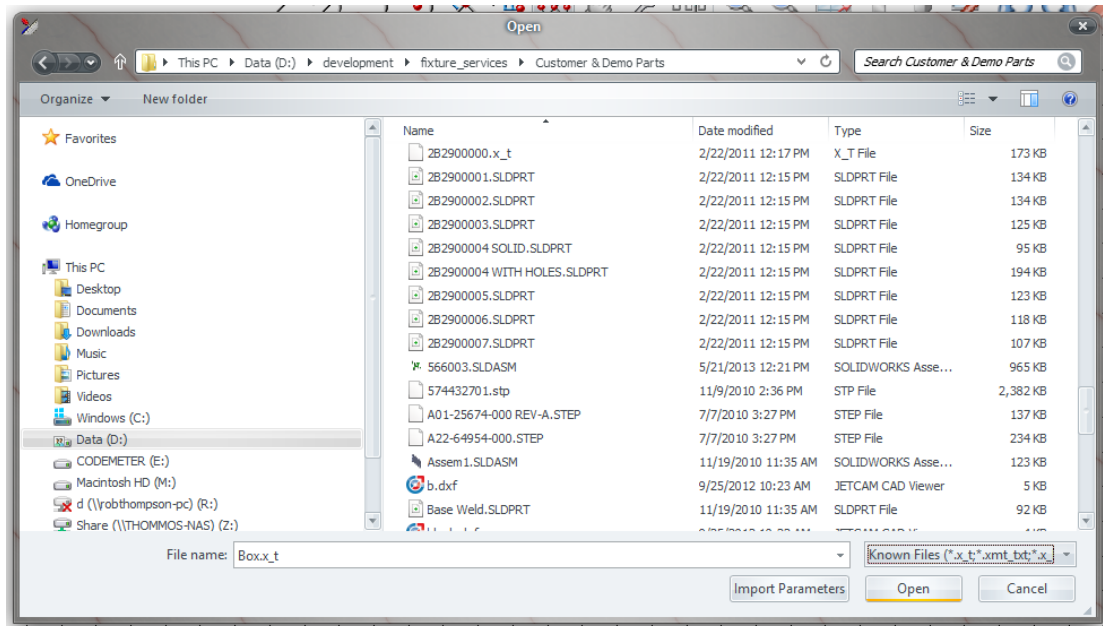
If you want to force a snap type for a given click, you can either:

- Use the right-mouse context menu when over the entity and select the snap mode for that pick, or
- With the cursor over the necessary point (ignoring that the cross-hairs may jump to an alternative snap), press F1 for Endof, F2 for Intersection, etc. (F1-F10 in the same order the snap modes are shown).

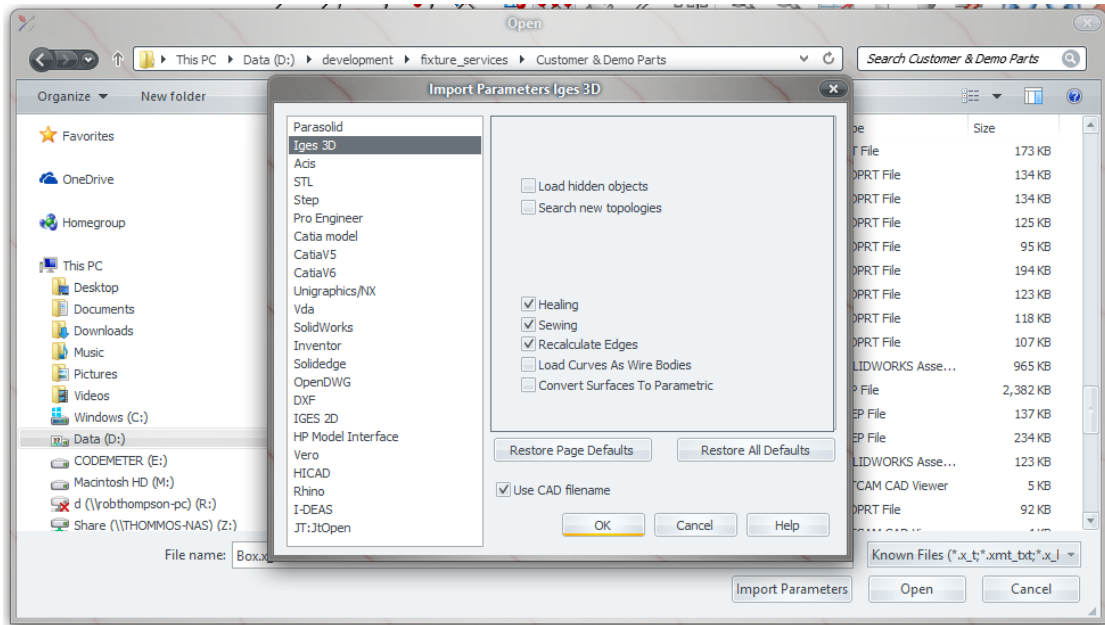
Importing a CAD Model



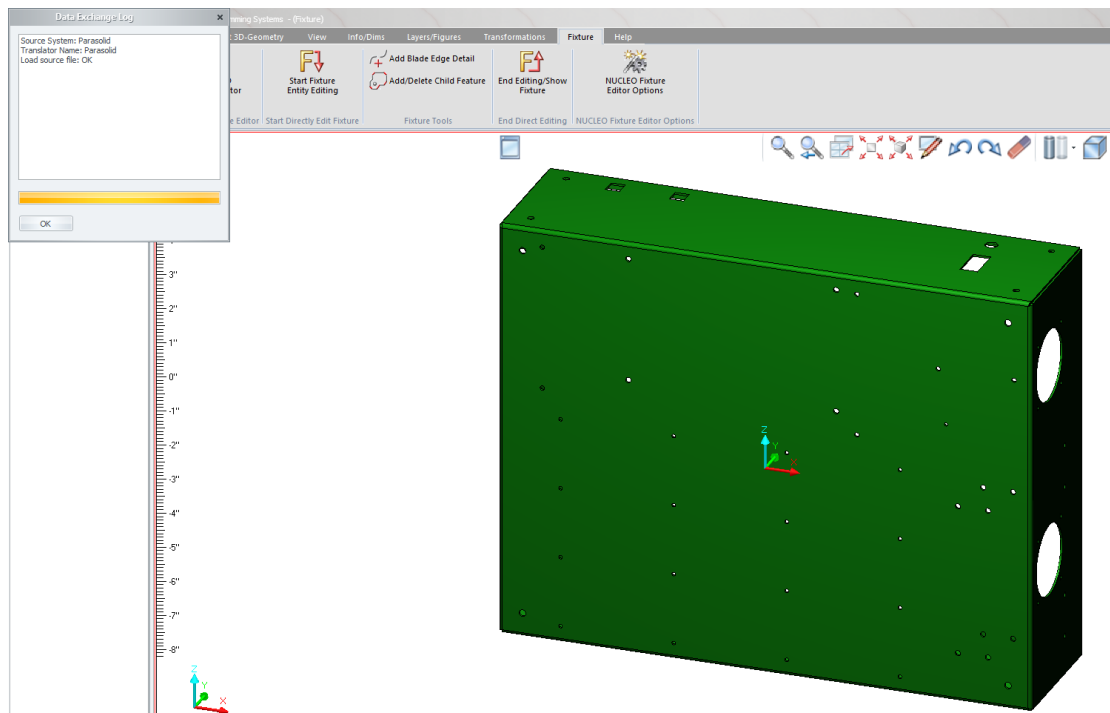
NUCLEO Fixture supports multiple CAD formats – depending on the options purchased there may be translators in addition to the standard 3D IGES and Parasolid ones.



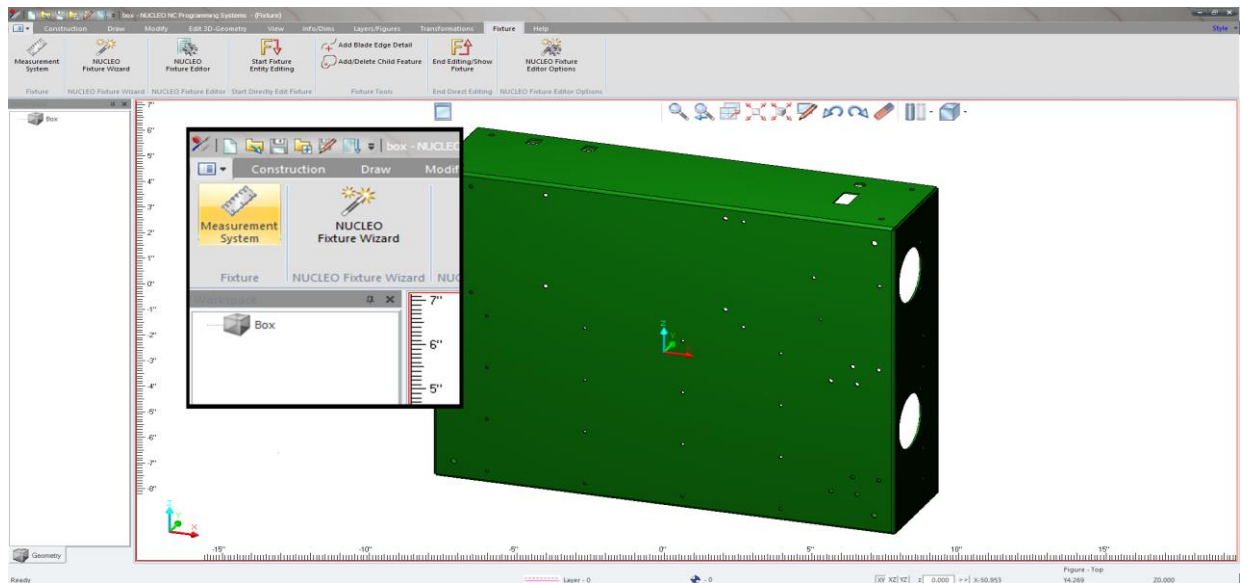
Additional options for CAD import are accessible by clicking the "Import Parameters" button. Depending on the CAD file type, setting the Healing and Sewing options can be useful to simply models (3D IGES for example). The "Use CAD filename" is good for setting the NUCLEO document name automatically.



Once the CAD model has been imported, the Solid entities will be listed in the Solids Geometry Workspace.



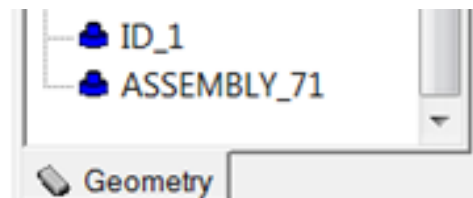
Note the Workspace below:



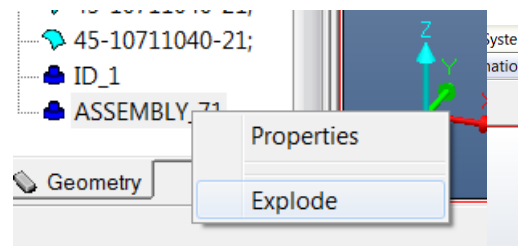
In the workspace, all the solid bodies found in the CAD file are listed. Generally, it is better to reduce the numbers of solids needed to create a good fixture, to reduce memory overhead and increase speed. Unsewn sheet bodies will also slow down the process, so a clean model is a good target.

It is important to review the Workspace before creating a Fixture. If it is evident in the Workspace that the imported CAD model contains assemblies, **these assemblies must be exploded prior to making a fixture.** NUCELO will warn you about this if the assemblies remain intact prior to starting the Wizard. It may not be obvious that assemblies exist in the CAD model – they may be listed at the bottom of the Solids Geometry Workspace so may only be apparent once the list is scrolled downwards.

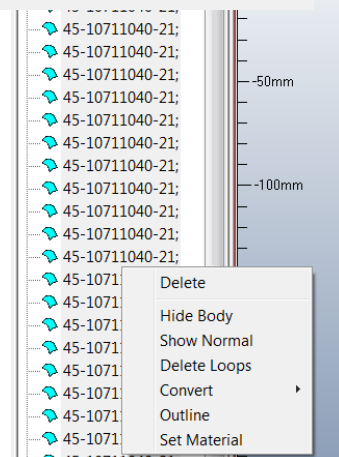
The image on the right shows an example of what assemblies look like in the Workspace. To explode the assemblies, repeat the following process until no assemblies exist in the Workspace (choosing the bottom most one first typically):



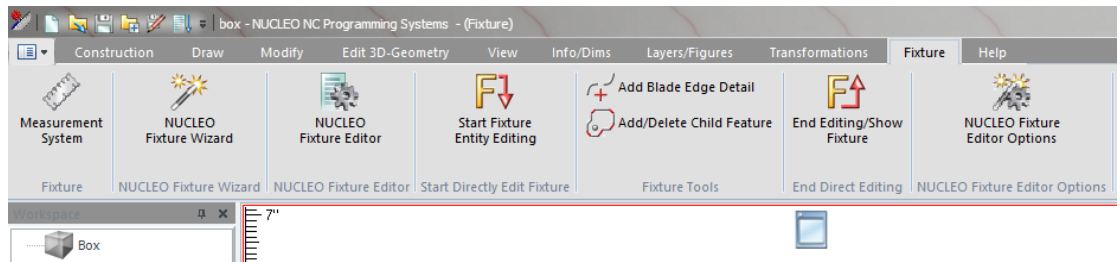
Right mouse click each entry and select “**Explode**”. If an assembly doesn’t appear to explode, try exploding another one first (as assemblies can be part of other assemblies so they need to be exploded in the correct order).



If there are extra “sheet bodies” in the Workspace that are unnecessary duplicates, it is better to delete them; it is preferable to have a sewn solid for Fixture Design. There are several ways to delete the remnant solids; one way is to use the **SHIFT** key along with the left mouse click from the last-but-one entry in the Workspace, then with the **SHIFT** key depressed, another left mouse click on the first entry (this will highlight the selection of Solids). Once selected, right mouse click in the Workspace and select “**Delete**”:

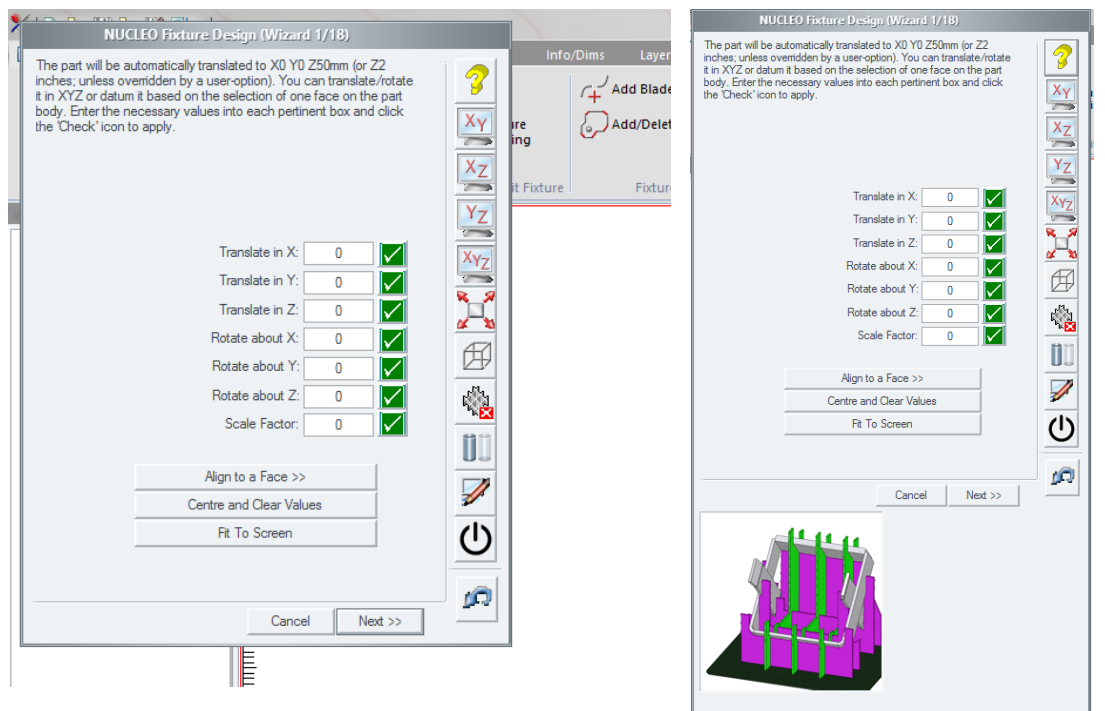


Running the NUCELO Wizard and Creating a Fixture



The “Fixture” icon set contains several entries: The first menu entry enables the system to be set to inch or metric. The second menu entry runs the NUCLEO Fixture Wizard that takes the user through a step-by-step process to design the fixture. The third option – “NUCLEO Fixture Editor” enables the user to go back and modify an existing fixture. Additional options are explained later.

Clicking on “NUCLEO Fixture Wizard...” starts the Wizard with this initial dialog:



Clicking the “?” icon shows an image helper, that outlines each important input on the wizard page (for the first page, there isn’t anything to really outline).

On each Wizard page, there are a set of standard quick-access icons on the right side. These manipulate the view, hide/show any fixture or part model, redraw, reset the default parameters for the given page and on some pages, provide an Undo feature.

The pages are self-explanatory - from this dialog it is possible to relocate the part or parts to a desired position/orientation. The “**Align to a Face**” option enables the user to select a face on the solid and align it to a given plane (normally XY); by doing so all attached solids are also transformed so that a part that is imported at a compound angle can be nicely orientated to XY. For this part, we want to rotate it 90 degrees about X to get it to our preferred orientation. To do this, we type in **90** and click the adjacent tick button:

Rotate about X:

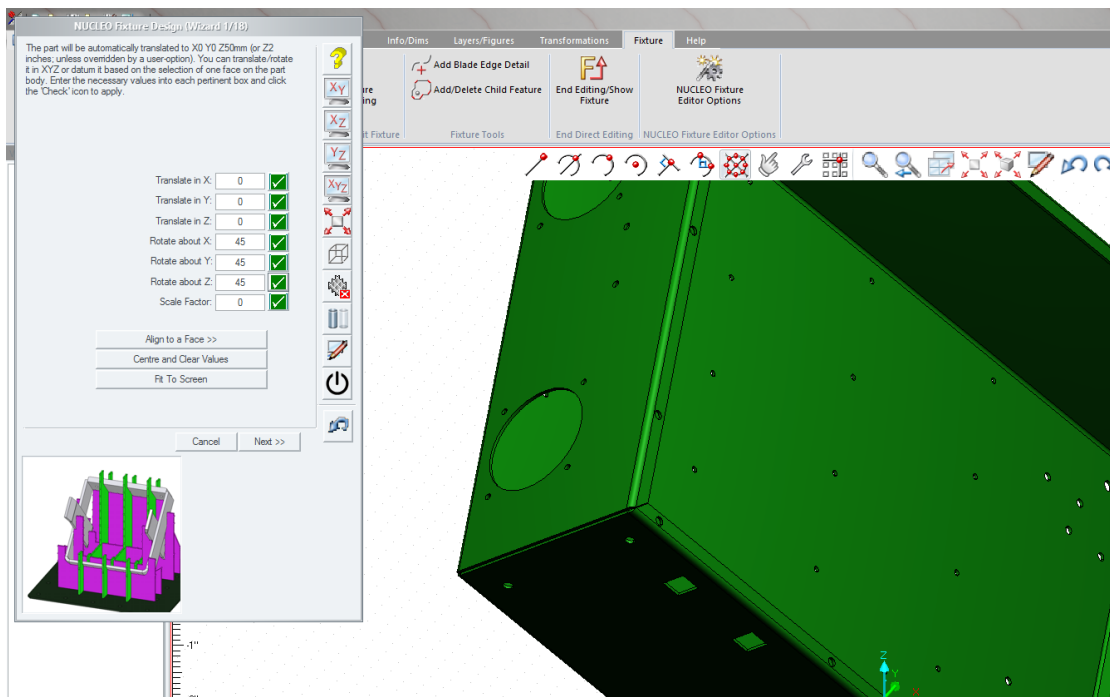
Alternatively, to understand the “Align to a Face” option, let’s get the part into an undesired compound angle:

Rotate about X:

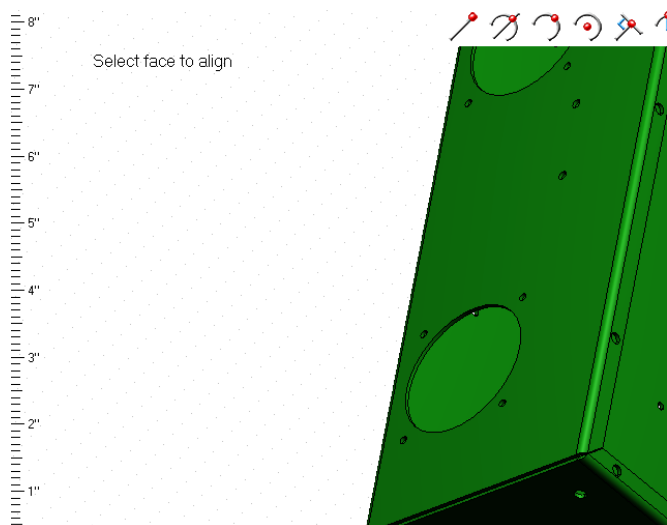
Rotate about Y:

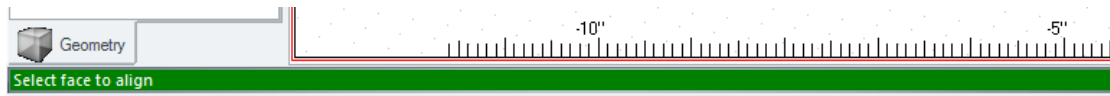
Rotate about Z:

Click each adjacent tick icon until the part appears like this:



Next, click “Align to a Face >>”

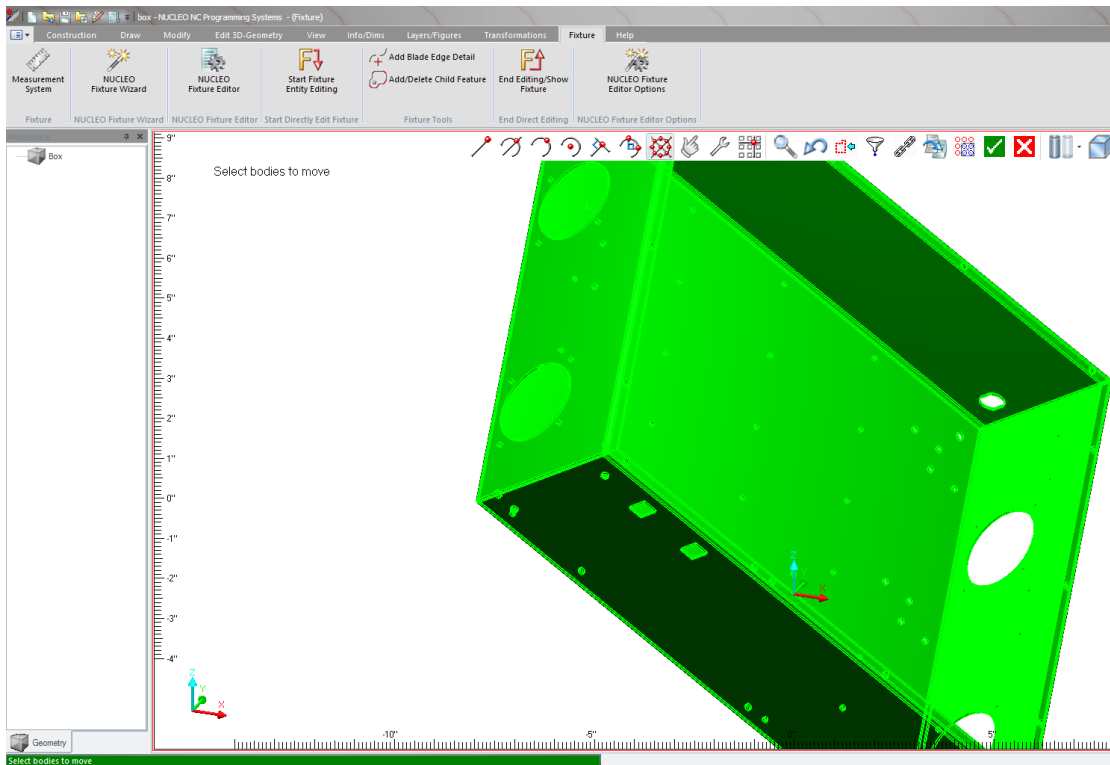




Note the prompt appears in two places, at the top in the viewport and at the bottom of the application window.

Hover the cursor over the middle part of the model, it will highlight in brown, then click it:

Next, follow the prompt and click the solid (this could be in fact a set of solid bodies in other fixture runs, so you could click each one individually or windows them). The selected solid turns bright green:



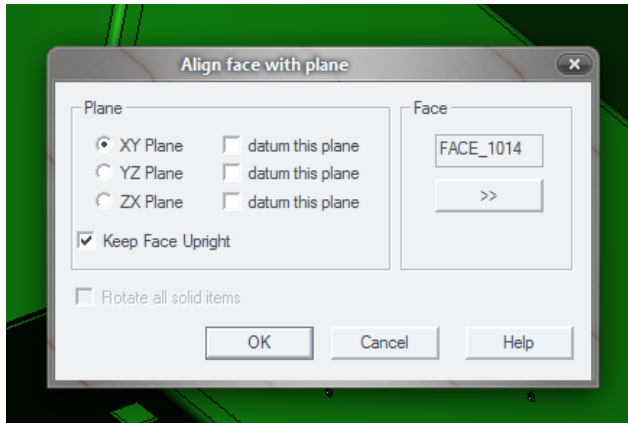
As we've only one solid, and we don't need to pick any more; press **[ENTER]** or click the Green Tick icon:



TIP:

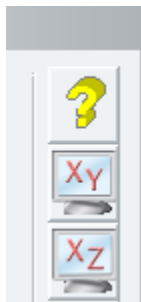
If you're every stuck, wondering what to do next, you probably need to press the **[ENTER]** key to a command line prompt.

The following dialog appears:



In this case, all options are defaulted correctly. We chose the middle of the box model, so we want to keep that selected face upright. If we want the box the other way up, we'd uncheck that option. Click **OK**.

In the Wizard, click the **XY** icon to go to an XY view:

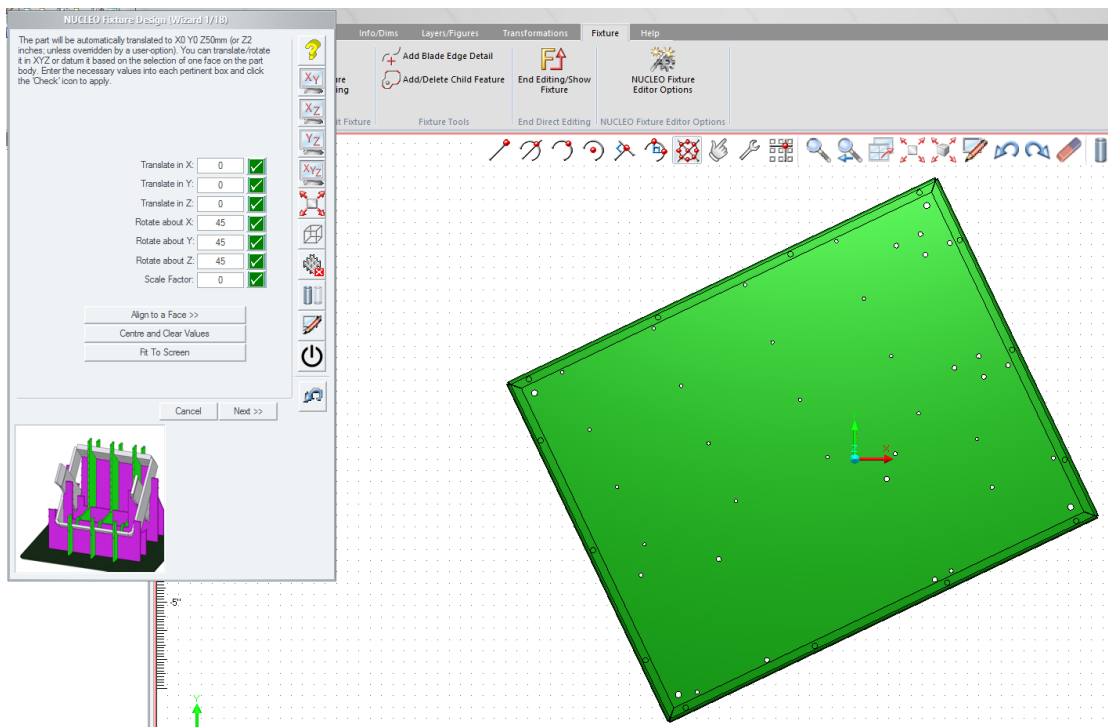


TIPS:

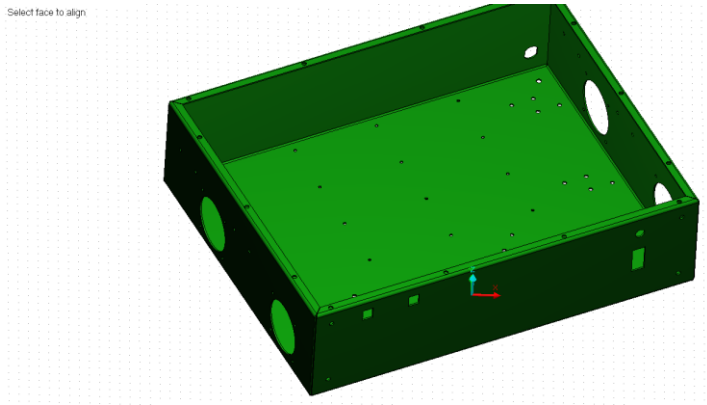
Pressing the **SHIFT** key down while in the middle of a command line prompt allows you to drag the mouse cursor to manipulate a the view (by rotating, zooming or panning).

The mouse wheel acts as a zoom button.

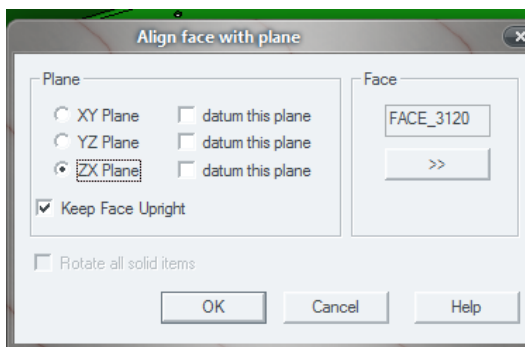
Both mouse keys together act as an **[ENTER]** key hit



Repeat the “**Align to a Face >>**” option. When the prompt appears for “**Select face to align**”, press and keep the **SHIFT** key down. Note the prompt disappears and the cursor changes. Drag the cursor in the middle of the viewport so it rotates the view similar to this:

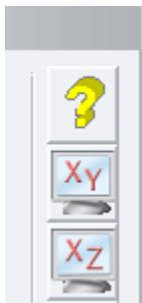


Release the **SHIFT** key. The prompt returns to pick a face, so pick the front face – near the WCS/Origin, then pick the solid body, finally followed by **[ENTER]**.

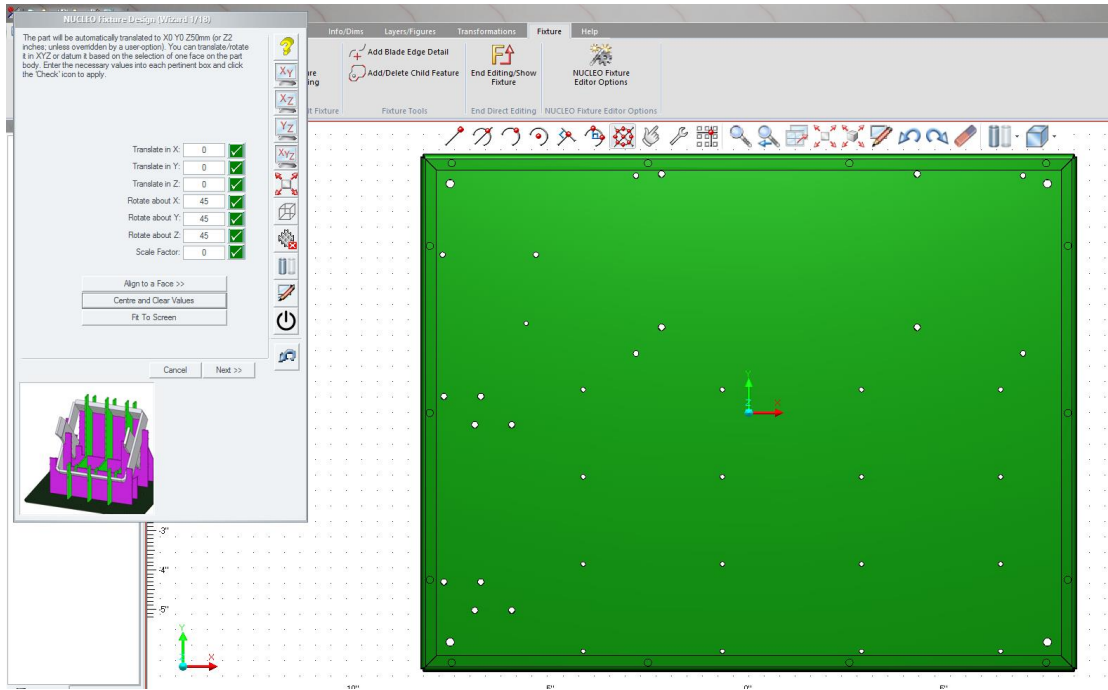


Select the **ZX Plane** option, then **OK**.

In the Wizard, click the **XY** icon to go to an XY view:

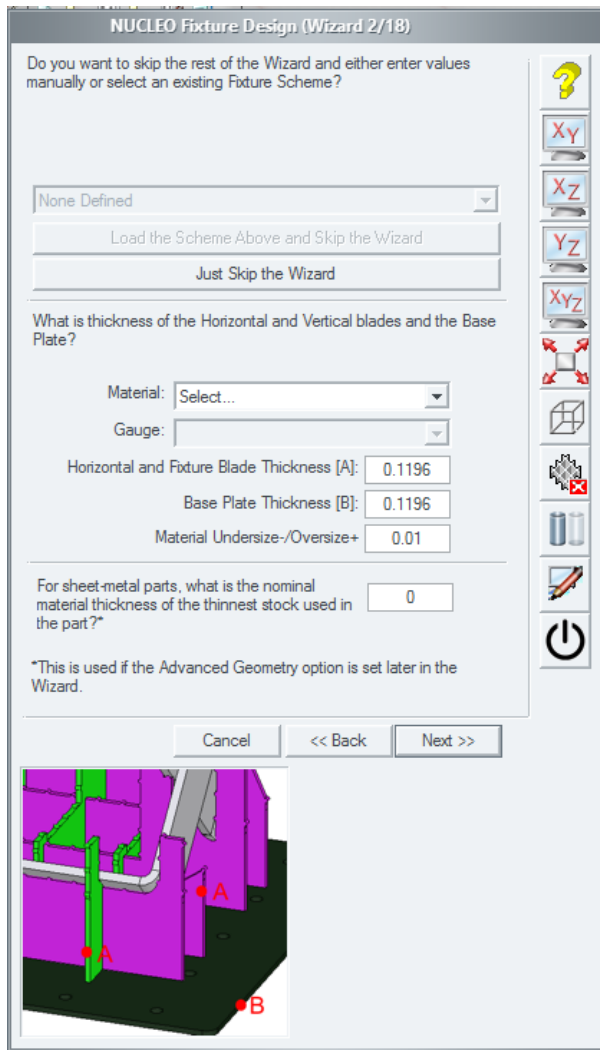


The part is now nicely aligned, however, it isn't necessarily in the correct place above Z0 or (if desired) about X0 Y0. To remedy this, click "**Centre and Clear Values**".



The part is now located at **X0 Y0 Z2** (the default position).

Once this is complete, the "**Next**" button is clicked to move on to the next stage.



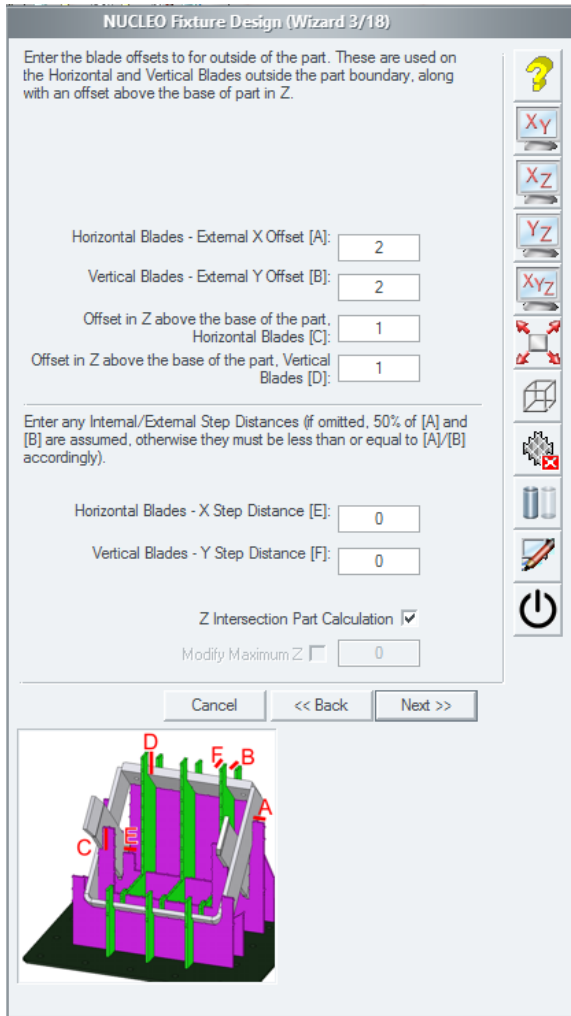
It is OK to leave it unset.

This wizard page enables the user to either load a pre-existing set of parameters for the fixture (a “**Scheme**”) or continue on with the Wizard, entering the pertinent values into the edit boxes. If the option is toggled, a pictorial image is shown with each dialog to explain each parameter, in the case of this dialog – the material thickness for the “**Blades**” [A] and the “**Base Plate**” [B].

Standard Material Gauges are listed in a drop list box.

The “**Material Undersize-/Oversize+**” option allows for a clearance to be added to the interlocking slots that each blade contains, so to interlock to blades going at 90 degrees in the other direction.

The “**nominal material thickness**” input is used to define the thinnest material section the software is likely to find. If you can imagine a “T” shaped solid section, the thinnest material dimension would be the X dimension across the very base of the T (when the T is upright in Z). This value is not critical nor does it need to be exact, but it helps in calculating the best settings used by the **NUCLEO/a** algorithm.



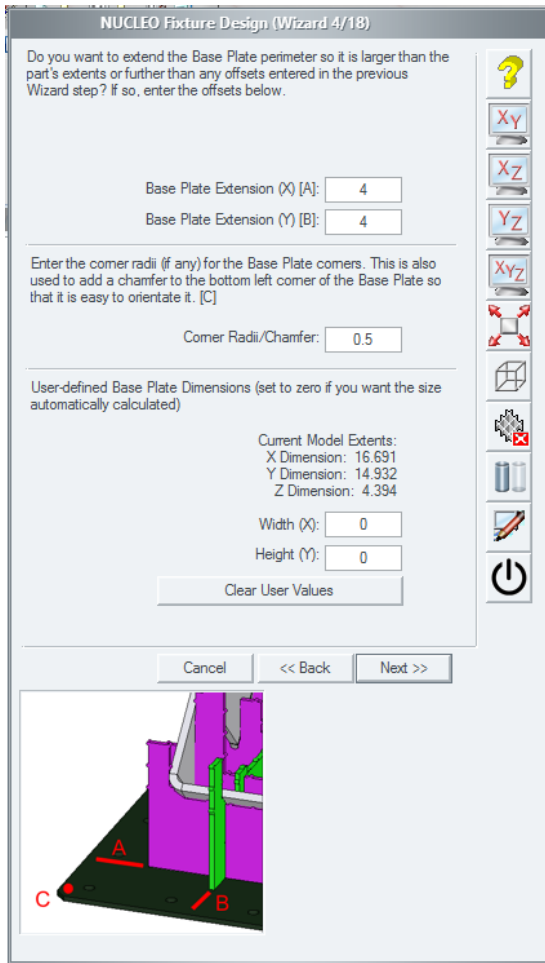
The page specifies the “encapsulating box” to create around the parts:

Typically, with a Welding Fixture, the part or parts need to be enclosed on several sides to keep the component parts or metal faces tightly clamped.

This dialog allows the entry of the offsets from the part extents to accomplish this; the offsets for both the horizontal and vertical blades along with a Z value from the base of the parts for both (“horizontal and vertical” is specified for the purposes of Fixture Design as being so when viewing the part or parts in an XY plan view).

Two advanced options are also provided on the dialog: **“Z Intersection Part Calculation”** and **“Modify Maximum Z Height”**; respectively, these change the method the fixture is generated if the part is tilted at an angle and also the maximum height the fixture will reach if the first option (“**Z Intersection Part Calculation**” is not active).

The latter option - **“Modify Maximum Z Height”** is seldom used, the default is to use **“Z Intersection Part Calculation”**.



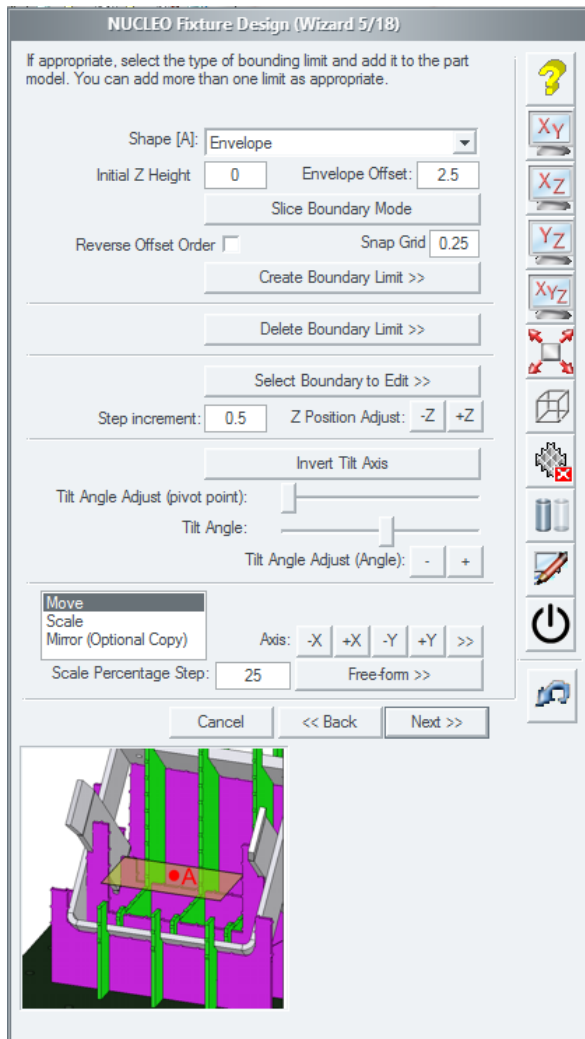
Next page:

The XY offsets for the base plate are entered here. It is advisable to add some offsets so that the optional base plate label (which can be laser etched or cut through) can be made easily visible and won't overlap with the notation on the base plate of the blade position numbers or the optional base plate bolt holes.

If the offsets are left as zero, the base plate will extend to the same extents as the horizontal and vertical fixture blades.

For those with a rotary axis robot table (head/tailstock), it is possible to enter a user-defined base plate dimension, so to fit the requirements of the rotary attachment.

Note that an optional corner radii is available for the corners of the base plate. This is used on three corners whereas on the bottom left corner, it is created as a chamfer so to easily identify the bottom left corner on the fixture assembly.



The “Bounding Limit” page:

This is a very useful and important function.

It is useful when a condition may exist in the part geometry which will result in some or all of the fixture blades protruding too high in the Z axis **or** to force the system to produce a fixture to a given Z position.

By adding this boundary, the blades that are created under the same location as the bounding limit, meet the limit. Also, adding boundaries provide cleaner results for some situations, providing more “meat” to fixture to.

The **NUCLEO/a** algorithm considers the boundaries just the same as the part model. Boundaries are particularly useful on high-tube component fixtures.

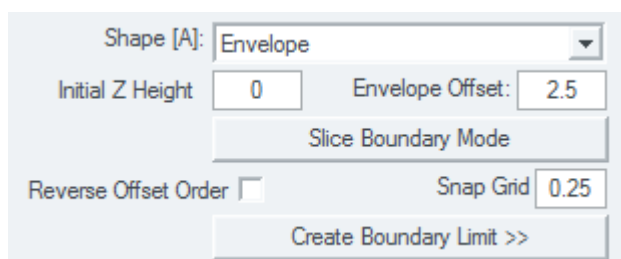
A different mode - “**Slice Boundary Mode**” – allows a rectangular boundary to be defined similar to the “Bounding Limit” method. The difference is that the slice boundary simply cuts the fixture solids off at the given height.

It is possible to create several different

shaped boundaries; unless an initial Z is entered in the “**Initial Z Height**” edit box, the Z position for the boundary is determined by the first left mouse click when the first point for the boundary shape is picked in the Viewport.

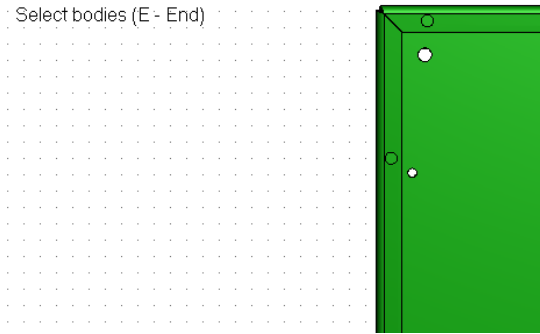
Boundaries may be moved up and down in Z, scaled, moved, mirrored, tilted etc., or deleted if no longer required, using the pertinent option within the dialog.

There are several bounding methods, “**Envelope**” being the most used. To demonstrate the effect of a boundary, we will add an envelope on this part:

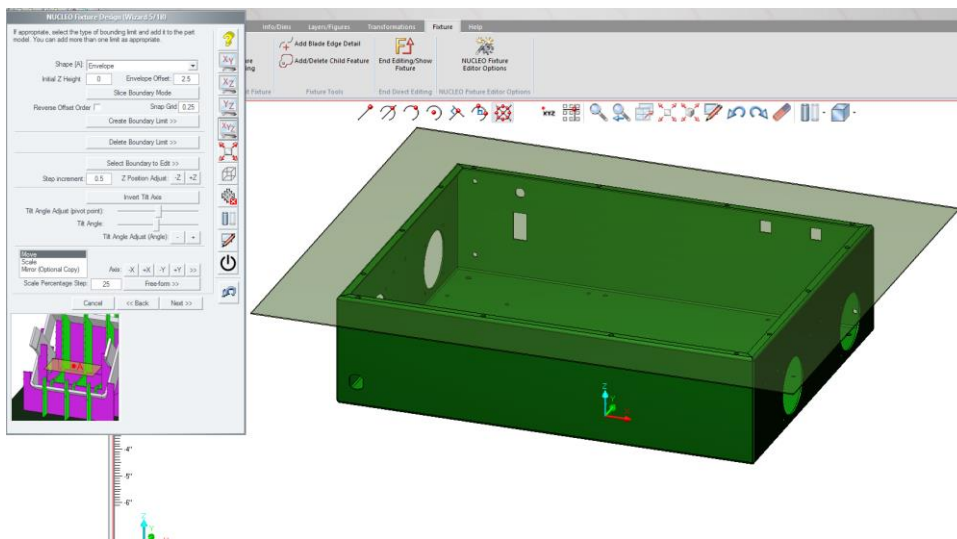


Ensure “**Envelope**” is selected, then click “**Create Boundary Limit >>**”:

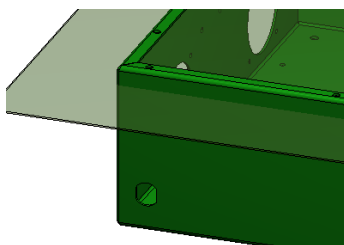
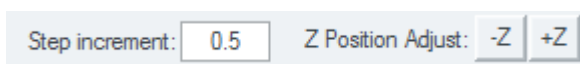
Select bodies (E - End)



Pick the body. In our case, there's just one. Press [ENTER]. Next, click the XYZ icon:



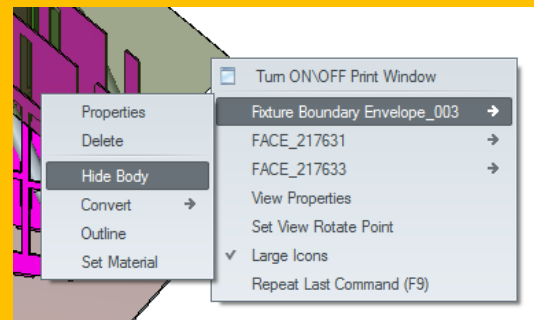
Note the new boundary. For the sake of example, lets move it down in Z a little. Click the “-Z” button:



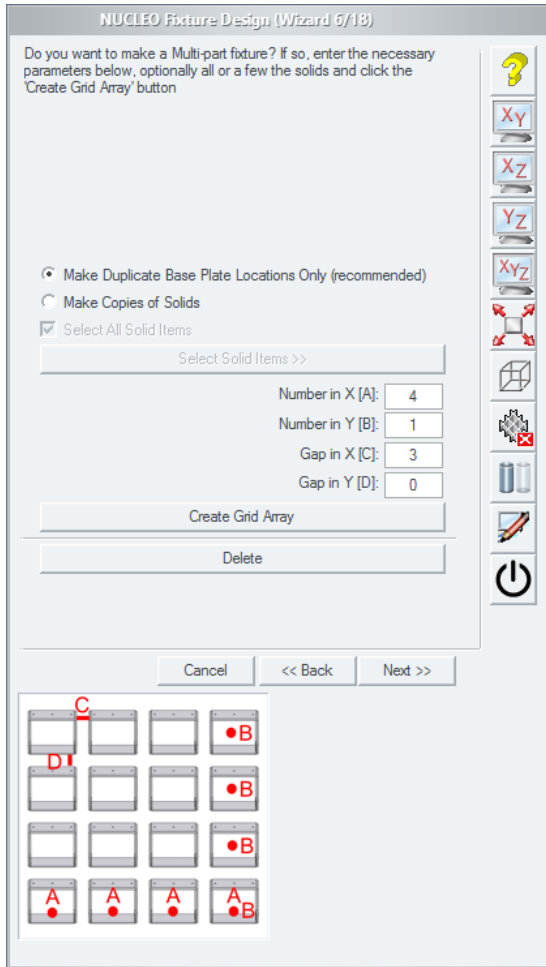
As the step distance is preset to 0.5”, the boundary moves minus Z0.5.

TIP:

If you later find a boundary is in the way of your model - that is sitting underneath it - and you can't readily snap to a position – you can easily hide it by right-mouse clicking on the boundary and selecting “Hide Body”



To restore all hidden bodies later, simply click in the viewport while not floating your cursor over an entity or solid. Next, right-click and select “Show All Hidden Bodies”.

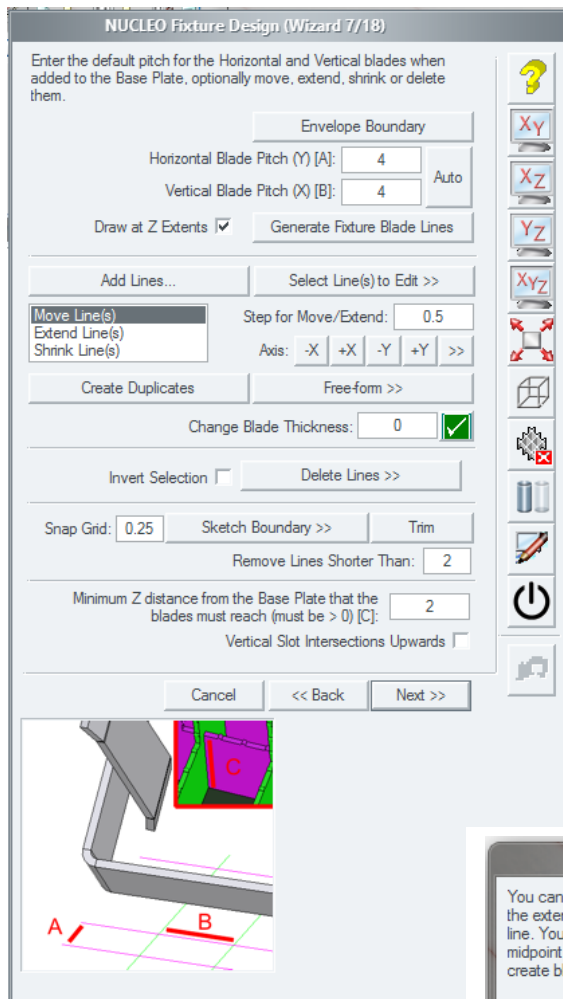


Next page:

It is possible to create a fixture that holds multiple identical parts. This page provides an easy way to duplicate the part in X and Y with a given gap between parts (a “**Grid Array**”). Also, if “**boundary limits**” have been created, these are also duplicated.

There are two methods to make a Grid Array; either by simply reproducing the tab feet apertures on the base plate to accommodate multiple sets of blades or by copying the solid parts to the necessary locations. **The first option is recommended** as it dramatically reduces the calculation time needed to create the fixture.

To create a grid of parts, the “**Create Grid Array**” button is clicked. Otherwise, the next step in the Wizard is taken.



Sometimes adding fixture lines manually is quicker; this is attained using the **“Add Lines...”** option.

When selected, a sub-dialog appears. Within this dialog, you can choose your method of adding lines; across the extents of the model, at a user-defined length or within a boxed-window. In addition, when adding lines using the first two options, you can choose to align the line through a snap position (for example, a hole-centre). This is activated by the **“Prompt for Alignment Point”** checkbox option.

From this sub-dialog, it is also possible to undo changes and delete fixture line positions.

On the parent page, the **“Snap Grid”** option, along with the **“Sketch Boundary >>”** button allows a freehand sketch to be drawn around the part periphery so that unnecessary line extensions can be trimmed back (useful for say an “L” shaped part). Once defined, the **“Trim”** button removes the remnant line segments.

Blade Position Definition Page:

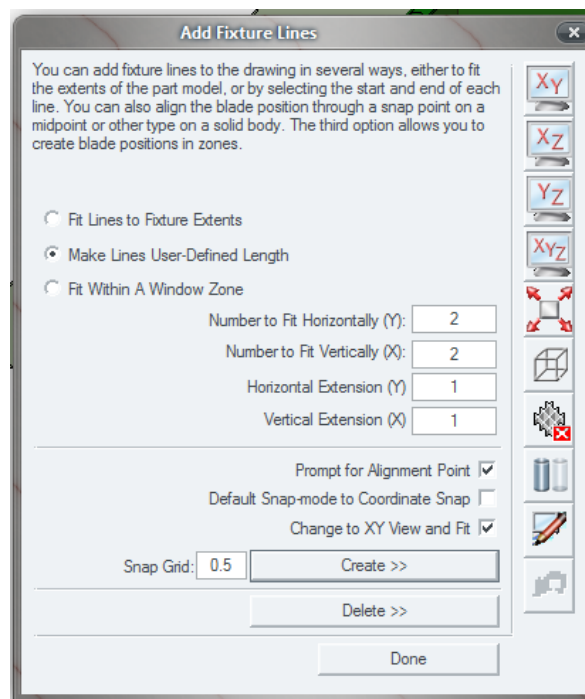
This page is the central function of the fixture generation process and has a wide range of options. It specifies the position of the horizontal and vertical blades.

A standard pitch in X and Y is entered and the position lines drawn (once the **“Generate Fixture Blade Lines”** is clicked or upon **“Next”**).

The **“Auto”** button will calculate a suitable pitch for the lines based on the part’s extents.

Fixture position lines can be moved, extended or shrunk (by a given increment supplied in the **“Step for Move/Extend”** edit) or moved to a fixed position by clicking in the Viewport using the **“>>”** button.

You can also change the target material thickness for any line or set of lines so to use alternative fixture sheet metal stock.

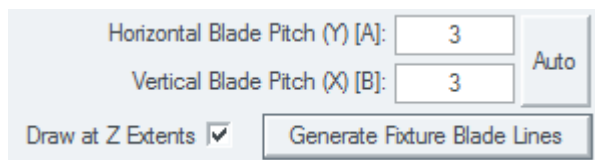


The “**Invert Selection**” check box swaps modes between *picking* what grid lines we want to keep versus *deleting* what we don’t want to use.

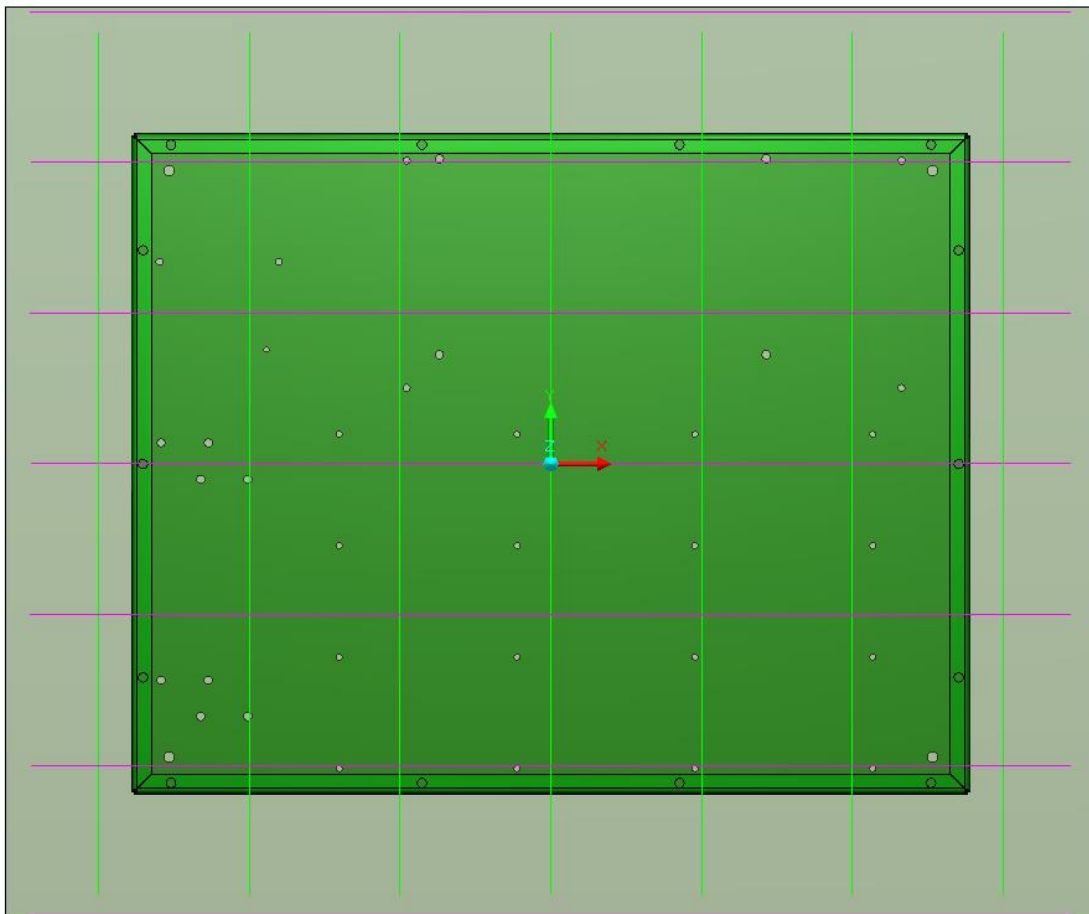
The “Minimum Z Height [C]” edit box is important: this determines the minimum height that every blade must reach – it cannot be zero. Depending on the part geometry, this value will come into effect – if the part is a simple box it will not be needed, but if – for instance, the part is irregular in XY, some blades will need extending to the periphery of the base plate at this given Z height (especially if there are no boundaries present).

Lastly, the “**Vertical Slot Intersections Upwards**” checkbox allows for the interlocking slots that marry the horizontal and vertical blades – to be reversed. This may be useful for parts that are rotated about X or Y during welding (on a rotary axis).

For this example, we will add a fixture blade pattern at 3” intervals:



Enter **3** and **3** in both edit boxes, then press “**Generate Fixture Blades Lines**”. Result:



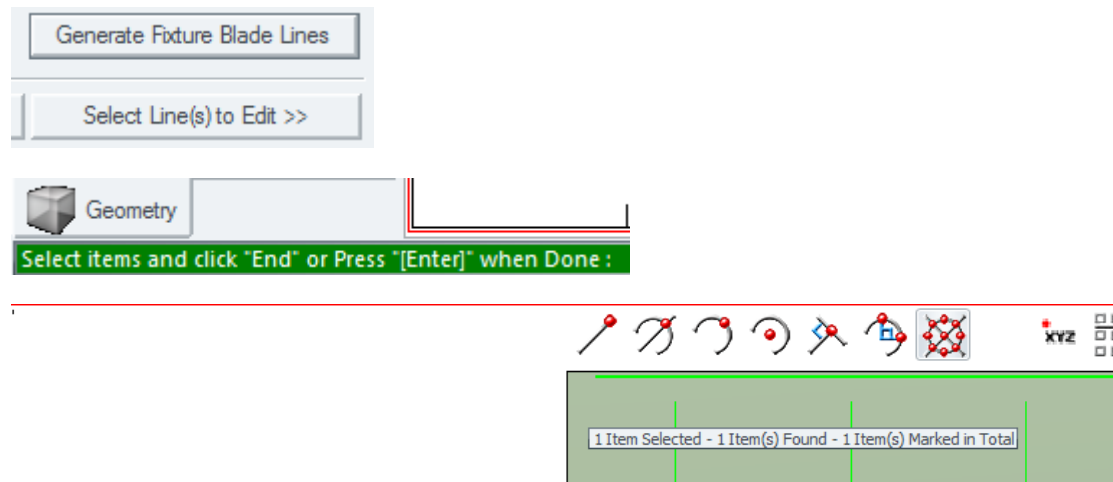
Because we defined a boundary, the boundary is considered when the fixture lines are calculated (unless when the boundary is created, we deliberately excluded it from being so). Note that the top-most and bottom-most magenta (horizontal) lines are not overlapping the green ones. This is OK, we will just move them in $-Y$ and $+Y$ respectively, and move the outermost green (vertical) lines in $-X$ and $+X$ respectively.

By having a boundary drawn, we have more “meat” the fixture to. If the boundary didn’t exist, our fixture line result would be different. Note that if there wasn’t a boundary or a solid body where the two magenta lines are located, a blade would not be generated.

One benefit of this approach is that the four outermost vertical and horizontal lines act as “stiffeners” to the external shape of the fixture. If this is not wanted, the lines can be deleted.

To move the lines into a good position:

Click “**Select Line(s) to Edit >>**”.

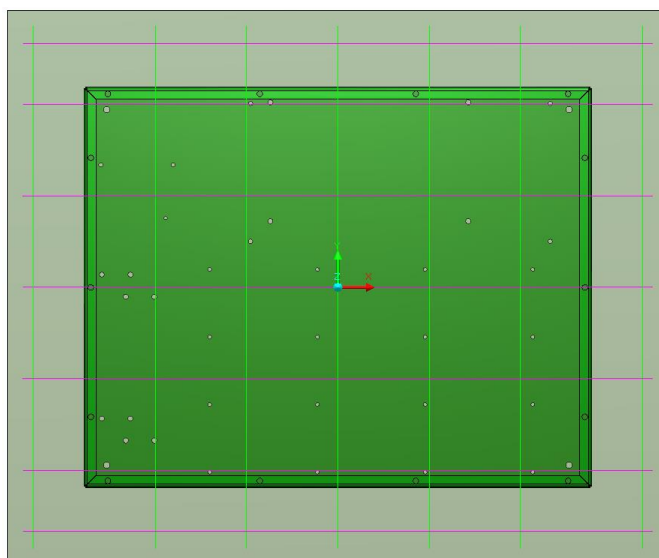


Pick top-most horizontal line, then press [**ENTER**]

Ensure “**Move Line(s)**” is selected then click the “**-Y**” button twice, to move the line down in Y by one inch.

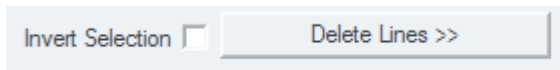
Repeat the “**Select Line(s) to Edit >>**” option, picking the lower horizontal line. Move it up using the “**+Y**” button.

Repeat two more times, for the vertical lines, using “**-X**” and “**+X**” respectively. The result should look like this:

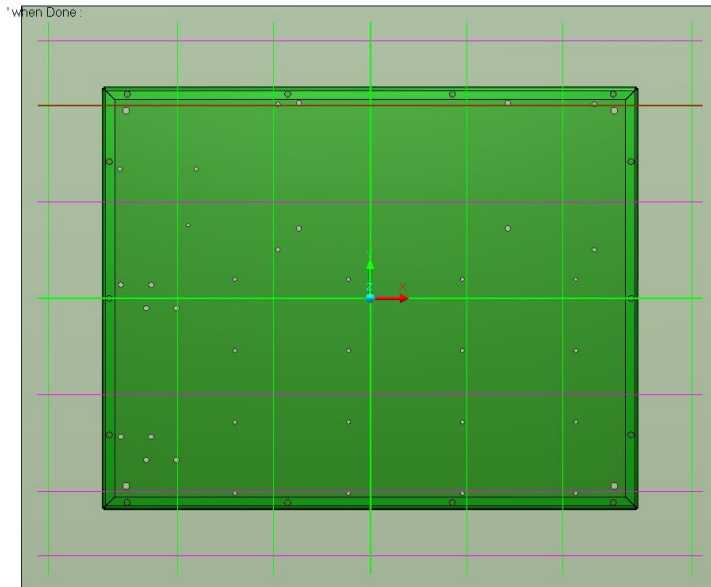


Next we’ll delete some lines we don’t need:

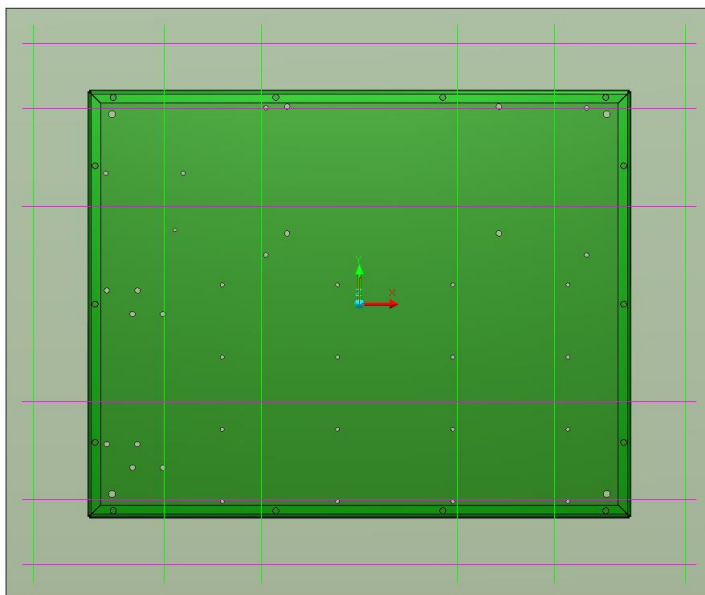
Click the **“Delete Lines >>”** button:



When prompted, click the two central lines:

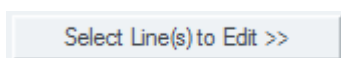


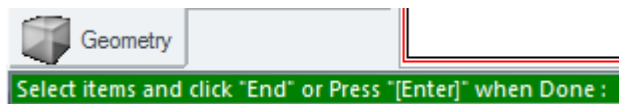
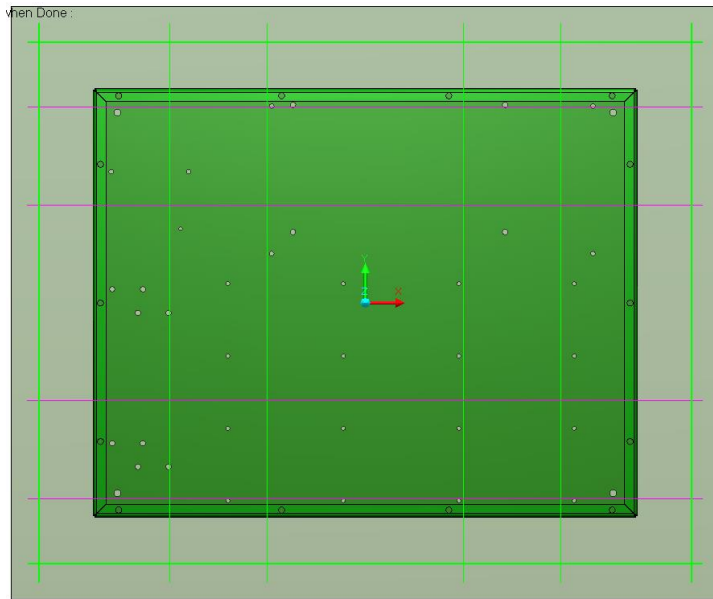
Press [ENTER]



Lets assume that welds are needed on the external corners of the box. At the moment, the fixture lines will inhibit access on the outside of the box, so we will shrink them back:

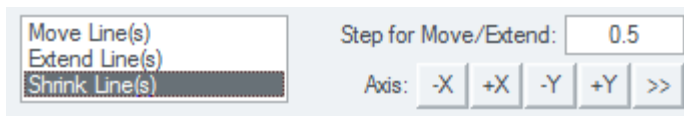
Click the **“Select Line(s) to Edit >>”** button:



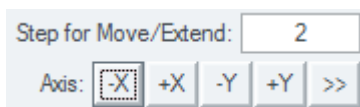


Pick the four external lines. Press **[ENTER]**.

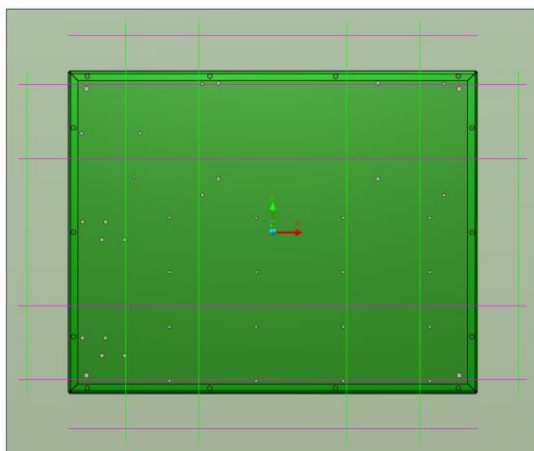
Select **“Shrink Line(s)”**



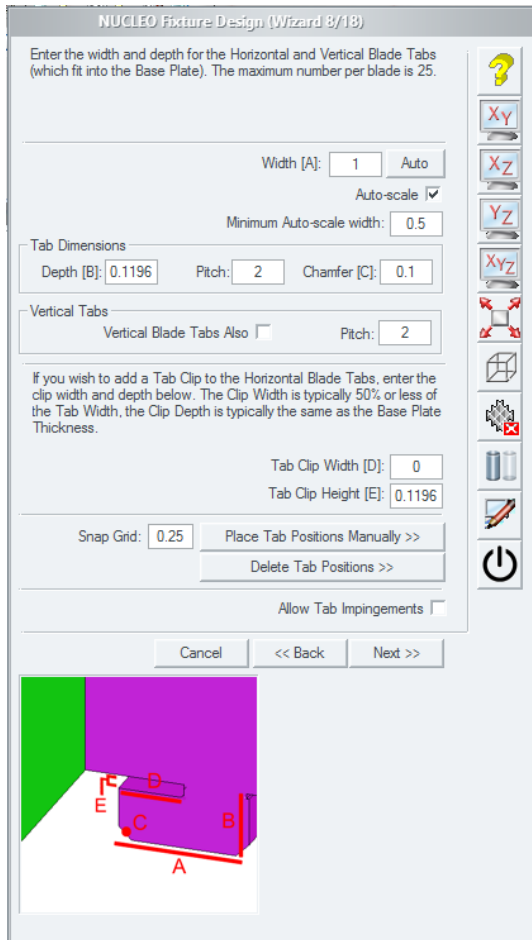
Change the step distance to **2**:



Click the **“-X”**, **“+X”**, **“-Y”**, **“+Y”** buttons in turn. The result should look like this:



We're done! On to the next page.



The “Tabs” page:

Each Horizontal Blade must have a “tab” created to mate to the base plate (which will have corresponding holes cut into it). Tabs are optional for the Vertical Blades. The geometric definition of the tabs is entered in this dialog.

The system will automatically generate tabs at the given width and at the same pitch. If “**Vertical Blade Tabs Also**” is also selected, these will be created where there is no conflict with the alternate horizontal blade tabs (assuming “**Allow Tab Impingements**” is not checked).

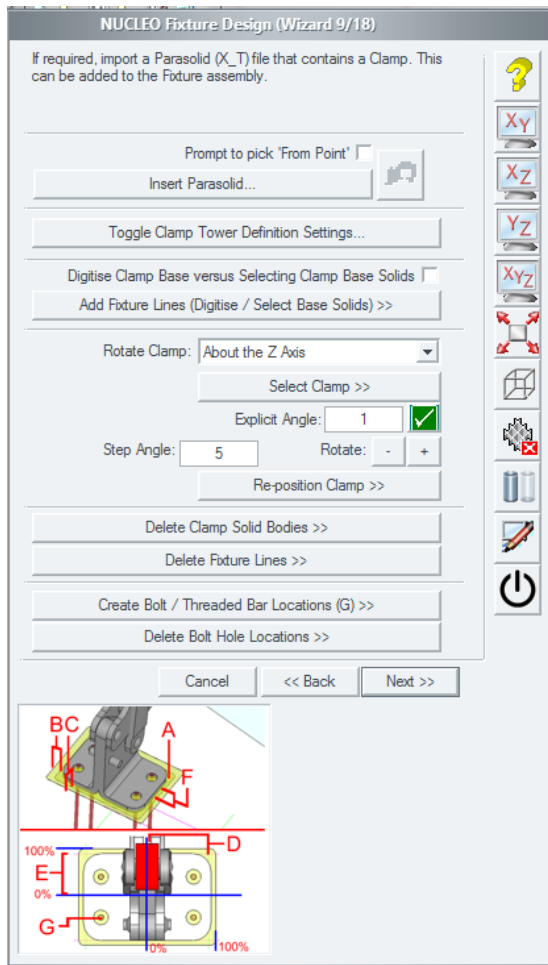
The “**Auto**” button will calculate a suitable size for tabs based on the part’s extents.

The “**Pitch**” value modifies the gap between each tab; it is a multiplication value based on the tab width.

If desired, user-defined tabs can be created using “**Place Tab Positions Manually >>**”; in which case positions are selected from the graphics window. A “**Snap Grid**” setting allows for easy alignment when picking the positions.

In addition to a simple chamfered rectangular tab, it is possible to create a tab with a clip mechanism: In this case, the tab should extend in Z beyond the bottom of the base plate (so be taller than the base plate thickness). A slot height dimension is entered that generally equals the base plate thickness, along with a slot width. The mating holes in the base plate are automatically compensated for the tab geometry so that when the blade is attached to the base plate, it can be “clicked” to the left to lock to the base plate. The intersecting Vertical blades ensure a tight and secure assembly that should not require spot welding. The base plate itself will of course need to be raised on risers to accommodate the tab clips.

Using such a mechanism allows the fixture to be disassembled and laid “flat”, thereby reducing storage considerations. Optional “Storage Clips” can be created also to clamp the stored blades together in order for easy organization.

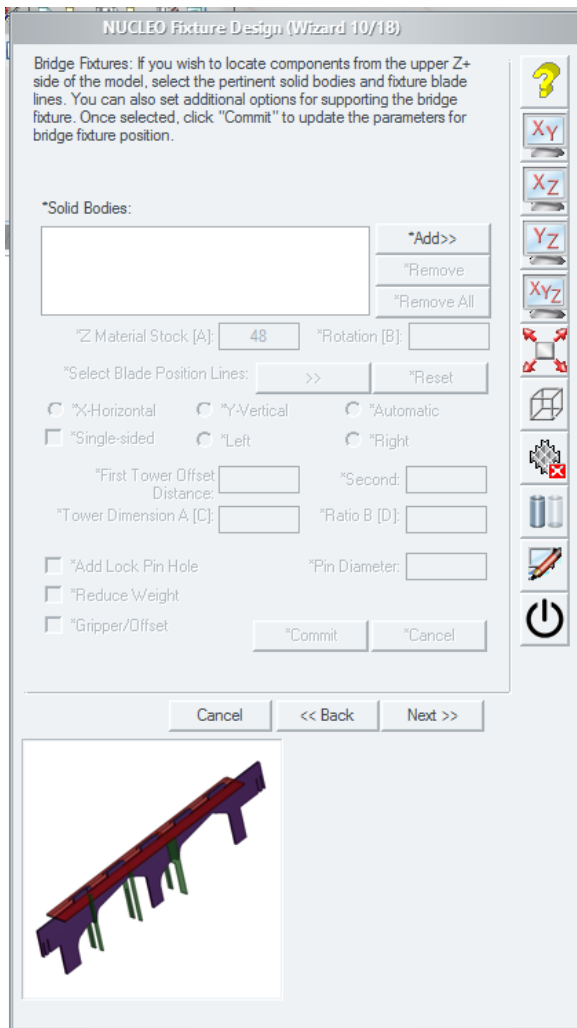


Next page: Clamp Import options:

This dialog supports the importation of Clamp CAD files in Parasolid X_T format. It is assumed that the CAD models are prepared as described in the section titled **“Using Clamp CAD Files”**.

By inserting CAD Clamp models into the fixture assembly, it is possible to very quickly create “Clamp towers” and matching clamp bolt-hole geometry.

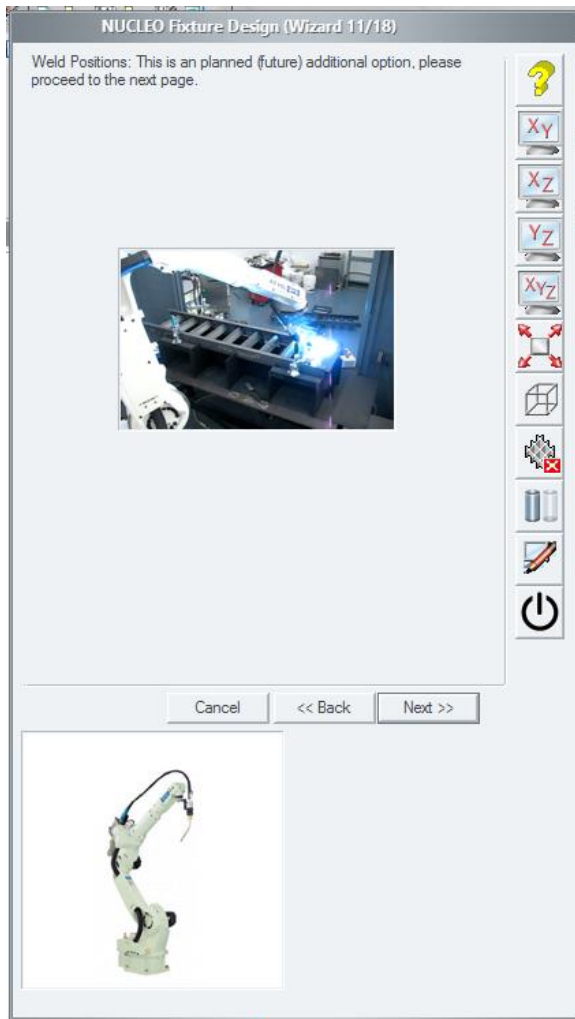
Because this function initially requires some initial preparatory work, it is explained further in the mentioned section.



Next page: Bridge Fixtures.

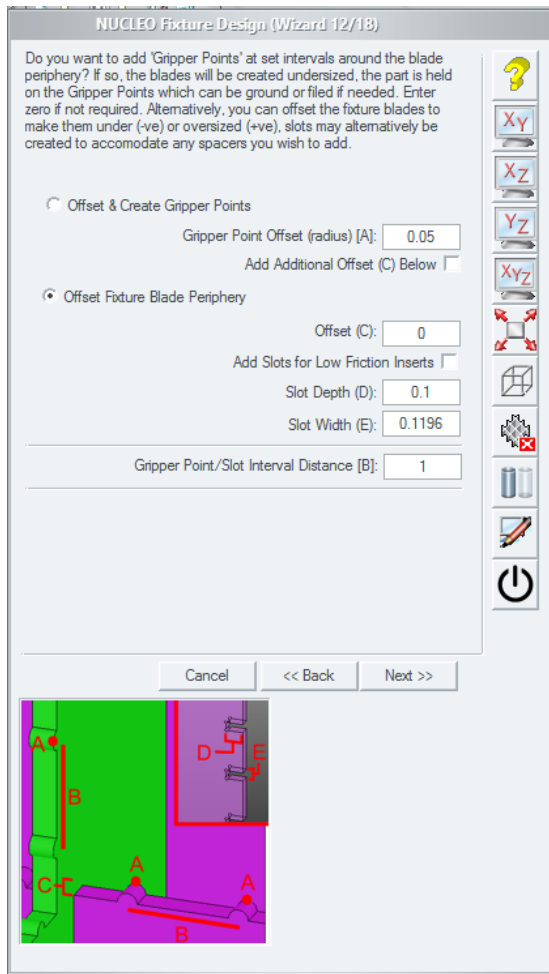
This page allows for the definition of "top-side" fixture clamps. These provide a mechanism to locate components from the plus Z side in a downwards direction.

This is an advanced option, for the benefit of explaining the process in a more concise manner, the details for this option are outlined in another document.



Next page: Weld Position Assignment.

This page is reserved for future use. It is planned to support offline programming for robotic tool path along with an associated kinematic simulation application and finally a post-processing solution.



example) that can be weaved into the slots across the blades. This is useful to prevent scratching parts that are soft and don't need painting.

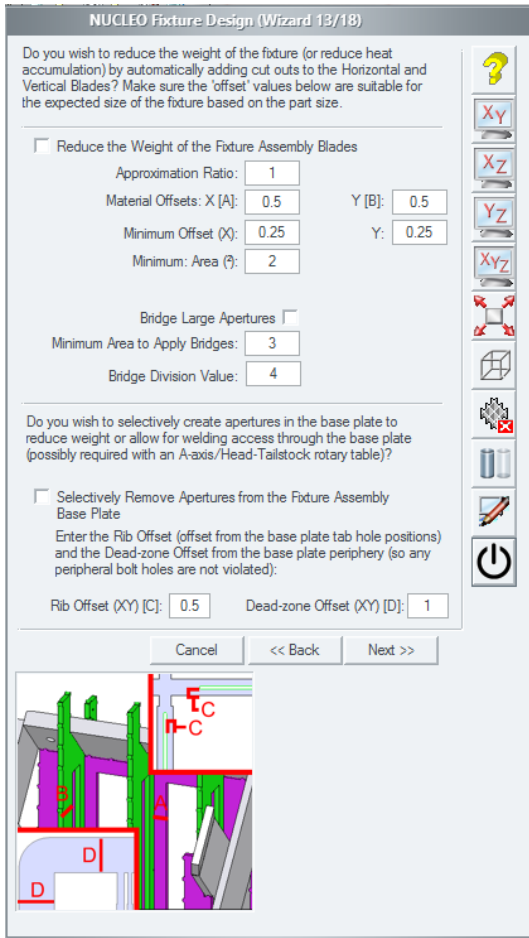
The "Gripper Points/Offset" page:

For some situations it is desirable to offset the fixture from the part geometry to allow for variance between the real-world part and the computer model of the part. In this case, the "Gripper Points" dialog allows an offset to be entered by which the horizontal and vertical blades are offset – to make the fixture oversized. Then, every "x" distance, a radius "gripper point" is added to the profile which matches the offset. The part then makes contact with these gripper points rather than the whole blade profile, therefore allowing for manual modification of the blades where needed by virtue of grinding or filing the gripper points down.

If "Gripper Points" are not required, a zero value is entered for the Gripper Point radius or the "**Offset Fixture Blade Periphery**" option is selected.

The "**Offset Fixture Blade Periphery**" simply allows an offset to be added to the fixture blade edge – normally to give extra clearance.

In addition to this function, it is possible to create "Low Friction Inserts". The purpose of these slots is to accommodate some low friction material (copper, for

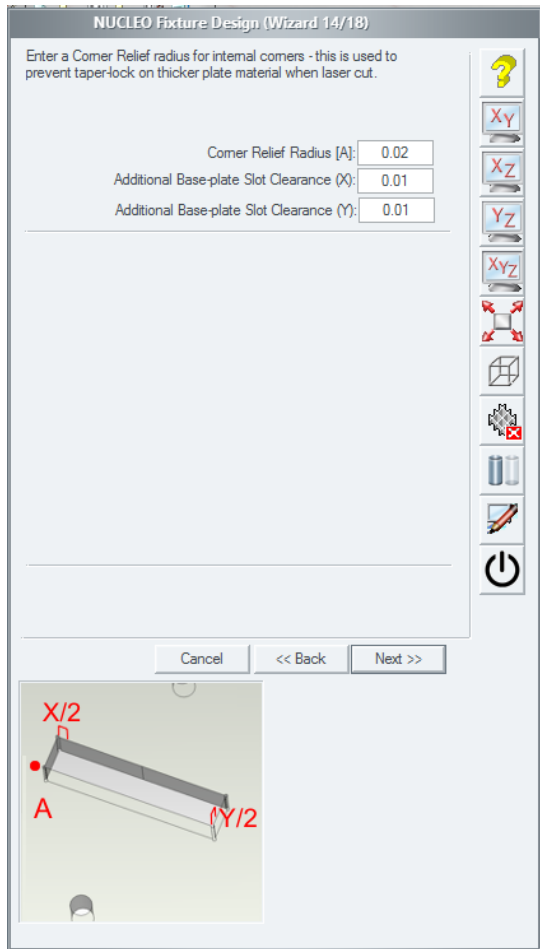


Next page, Weight Reduction:

This provides options to add cut-out features automatically to the horizontal and vertical blades and/or selectively in the base plate.

An offset for X and Y is needed for the horizontal and vertical blades respectively. These are also used for the base plate, if required.

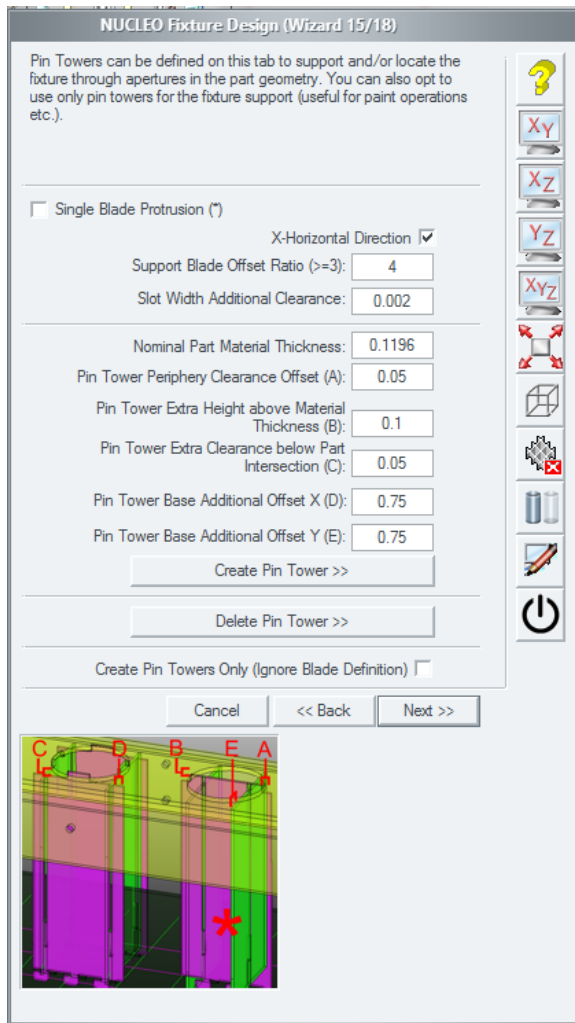
The “**Bridge Large Apertures**” option is used for very large fixtures where a strengthening rib is needed to add rigidity to the fixture, across large cut-outs. Note this option takes more computational time.



The “Corner Relief” page:

This dialog enables the specification of “corner relief” (“Mickey Mouse ears”) to the base plate tab feet cut-outs and blade tab feet.

In addition to the radius value, an oversize value for X and Y is also provided to give clearance.



Next, the “Pin Tower Definition” page:

This dialog enables the creation of what could be considered “location or pin towers”. Using a special CAD function to select the edge of a feature and along with several parameters, a pin tower can be automatically generated at the necessary location to additionally support the fixture.

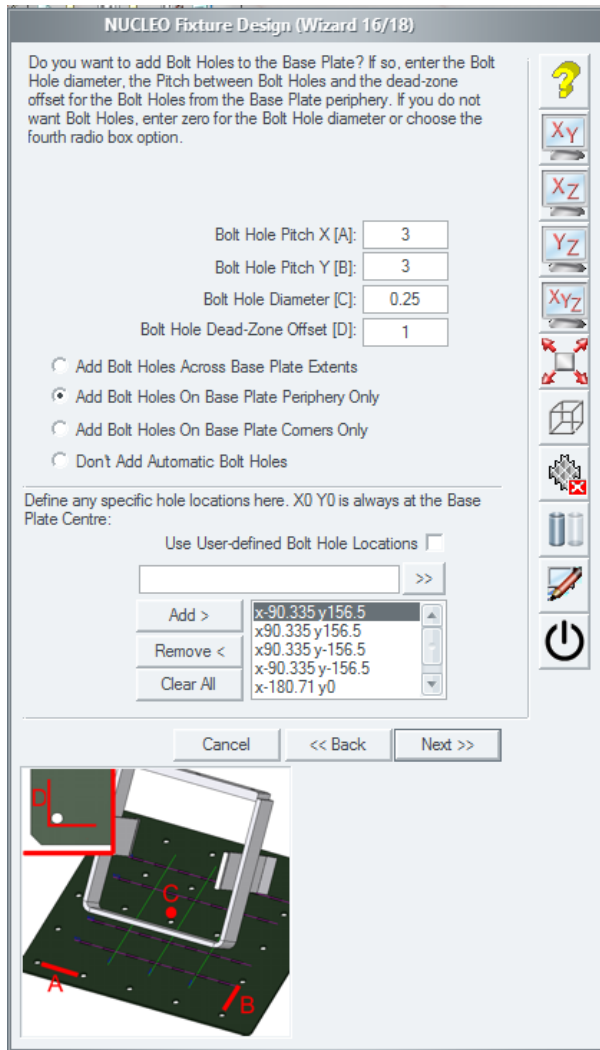
Two variants are available (as shown in the image):

- 1) Four Point pin tower – this is where four blades are created to intersect with the feature periphery, giving four contact points (good for large holes/features).
- 2) Single Blade Protrusion pin tower – this uses a single protruding blade edge to locate the part (in an either the horizontal or vertical direction when viewed in plan view). This is better for smaller features/holes.

Note that the hole/feature may be any shape.

It is also possible to only create the pin tower blade items – this is

accomplished by checking the option “Create Pin Towers Only (Ignore Blade Definition)” option. In this case, only the pin towers are created. This is useful for operations such as QA, paint preparation etc. Instead of the part being encapsulated by all the fixture blades, it can be supported on the pin towers allowing better access for the appropriate process.

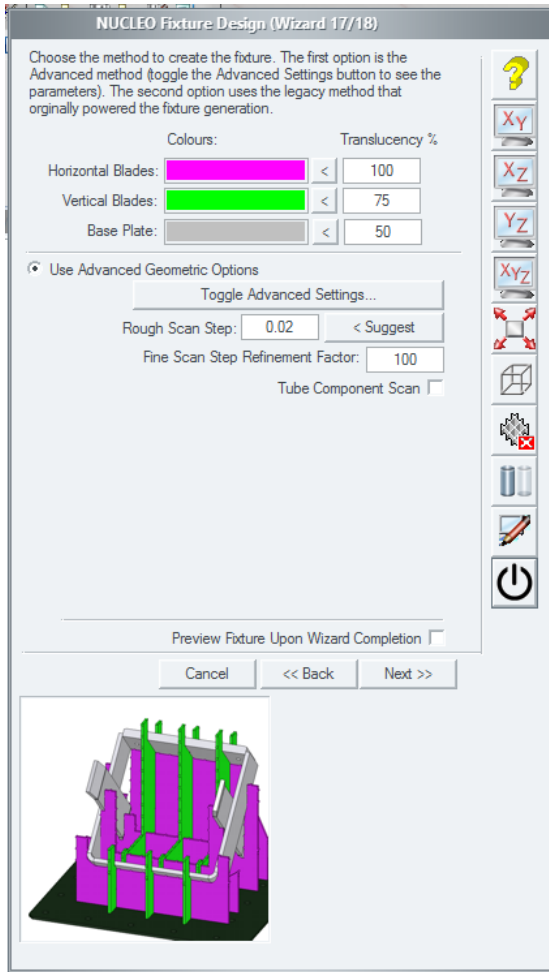


The “Bolt Hole” page:

This dialog defines any optional bolt holes for the base plate. The pitch in X and Y is specified along with the bolt hole radius and an offset from the base plate periphery. When created, the system will place bolt holes throughout the base plate wherever there is no interference with blade tab feet locations.

To provide an adequate offset from the horizontal and vertical blades to the bolt hole pattern, it is recommended that the base plate be created oversized using the parameters in the fourth Wizard dialog – “Base Plate Extension” settings.

Note it is also possible to define user coordinates for bolt holes – sometimes needed for Rotary Axis solutions.



The “Geometric Results” page:

This sets the manner in which the fixture assembly is produced.

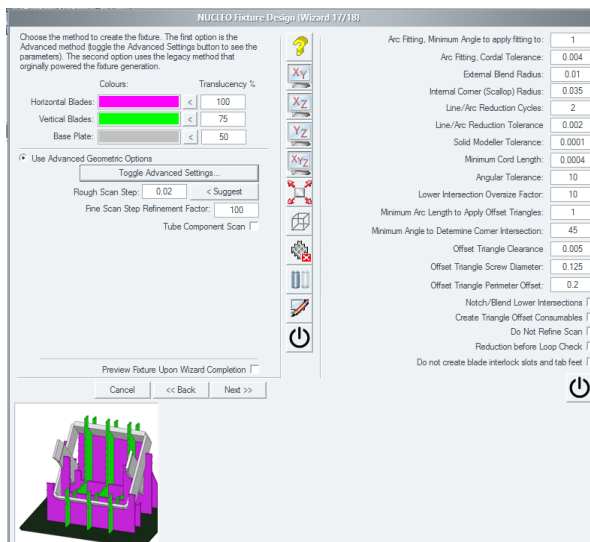
Fundamentally, the **NUCLEO/a** algorithm works like this:

- It scans along the fixture line from left to right (horizontal lines) and bottom to top (vertical lines). Every “x” distance (as defined by the “**Rough Scan Step**” in the Wizard page), the algorithm plots a point where it finds a Z intersection with a solid body. With this “point cloud”, the rough section is computed, taking the lowest Z intersection point across the width of the blade material.
- Based on the value of the “**Fine Scan Step Refinement Factor**”, and the options set in the “**Toggle Advanced Settings**” section, the algorithm repeats the process but only at corner intersections. So, if the rough scan is 0.02”, the fine scan will look at corners at 100 times smaller than this value (0.0002”)

- Depending on user-preference and part topology, a larger scan step (maybe 0.03”) and a smaller factor (say 50) is also appropriate. Of course, the tighter the rough scan and the bigger the factor, the slower the result.

The “<Suggest” button takes the “**nominal material thickness**” value - optionally input in the second page of the Wizard (used to define the thinnest material section) and computes a good default step and factor accordingly, if set.

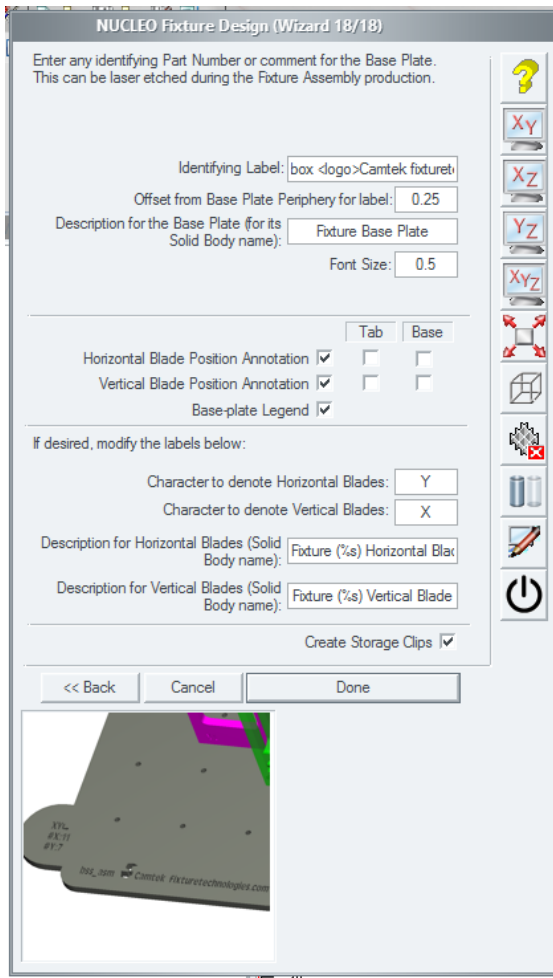
The “**Tube Component Scan**” option does additional checks to attempt to add triangular offsets into the blade periphery, if desired.



Otherwise, the solid fixture component parts can have their colour and translucency set in this page. For other assistance on the many advanced options, float the mouse cursor over the pertinent item in the dialog to obtain a tool-tip.

If any values are changed for experimental purposes, the system defaults may be reset by clicking the button in both sections of the page:



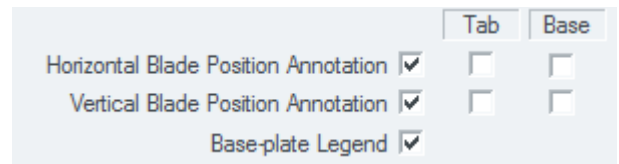


The final Wizard page:

This page allows the entry of an identifying label for the Base Plate and some miscellaneous parameters (these will not be changed very often).

The tool-tips for the relevant options explain and give hints on additional parameters that may be set for annotation purposes.

The section below deserves a mention:

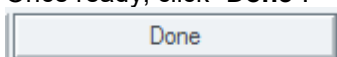


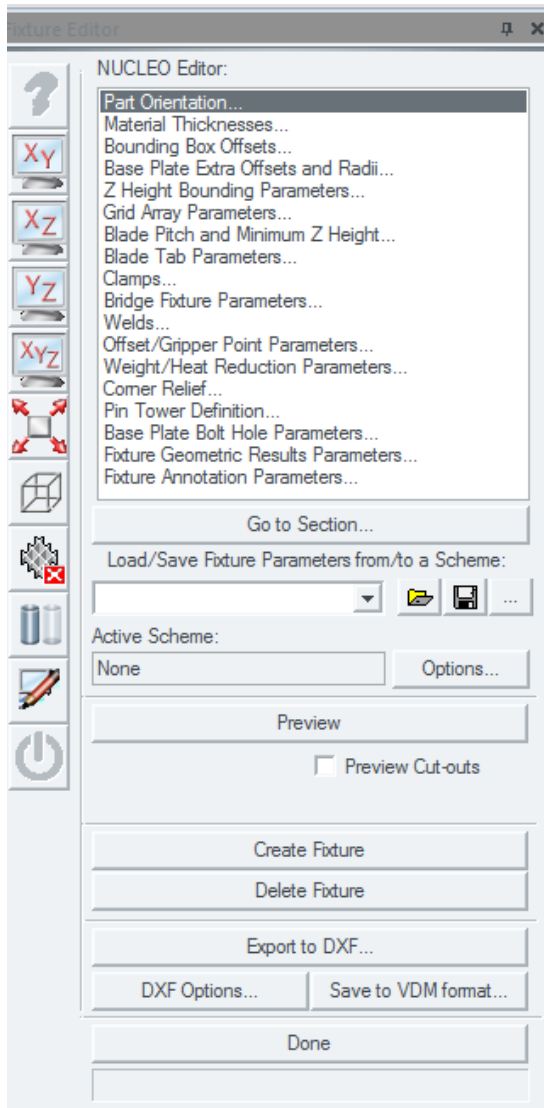
Each blade will have its reference number cut in to the solid fixture blade. If the “**Tab**” check box is checked, the annotation will be added to a “balloon” to the edge of the blade. If the “**Base**” option is set, the annotation will be cut through the base plate near the position for each respective blade.

Care must be taken with the “**Base**” option to avoid blade positions that are close or overlapping, so that one blade

direction’s annotation does not obliterate another alternate direction’s base plate annotation. Likewise, if a blade fails to create properly, it may be that the length of the blade position line needs to be extended to create suitable geometry, if the “**tab**” option is used *and* the blade is intersected at the tab position by an alternate-direction blade.

Once ready, click “**Done**”:

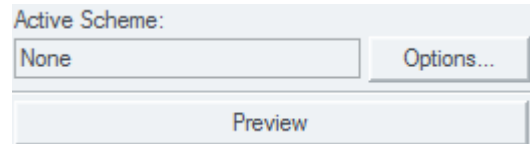




The “**NUCLEO Editor**” dialog is the central interface to modifying and creating fixtures and is executed at the end of the Wizard.

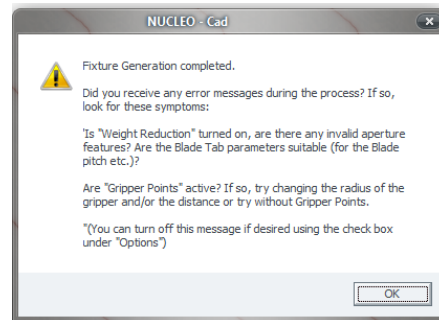
In here, you can go back to any step on the Wizard or choose to create previews or final version of your fixture design.

To see a quick preview of the fixture, the “**Preview**” button is clicked:

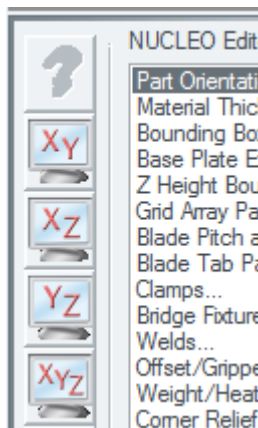


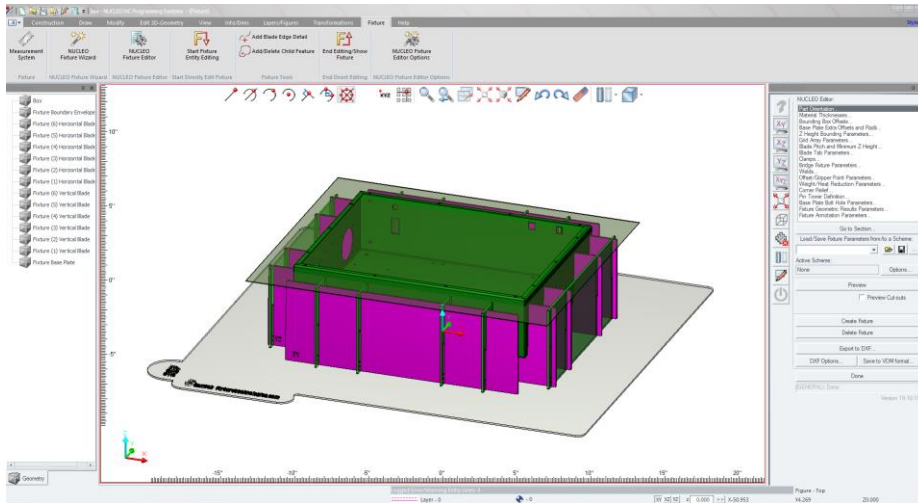
A backup of your fixture to-date is created (and is cycled through five revisions by default).

Once created, you will see a fixture and part model and a message box giving you hints – should there be any issues. You can turn this message off by setting the parameter via the “**Options...**” button once you understand the assistance it is offering and don’t need to see it again.

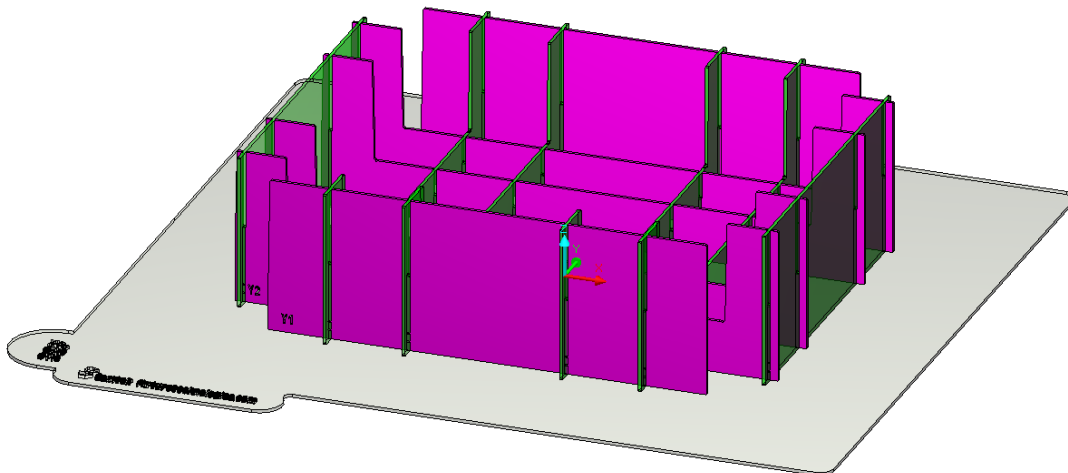


Click the “**XYZ**” button:

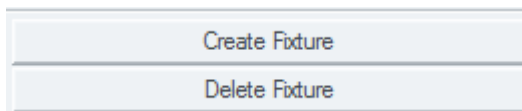




To see the fixture without the part or boundary, click the **“Toggle Part”** icon:



If content with the result, click the **“Create Fixture”** button:

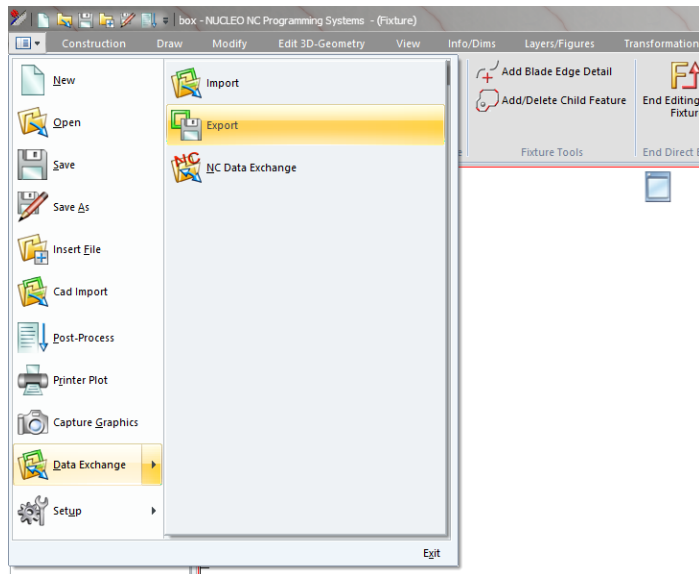


The process – depending on the geometric options – will take longer than the preview. In the end, you will end up with a finer, more accurate fixture.

To export your model to a DXF file, used the **“Export to DXF...”** button option. Note that further DXF options exist under the **“DXF Options...”** dialog.

When complete, click **“Done”** to close the editor.

Lastly, save your part document, then if you so wish, export the model to either Parasolid X_T or 3D IGES format (if you want to open it in your favourite CAD system):

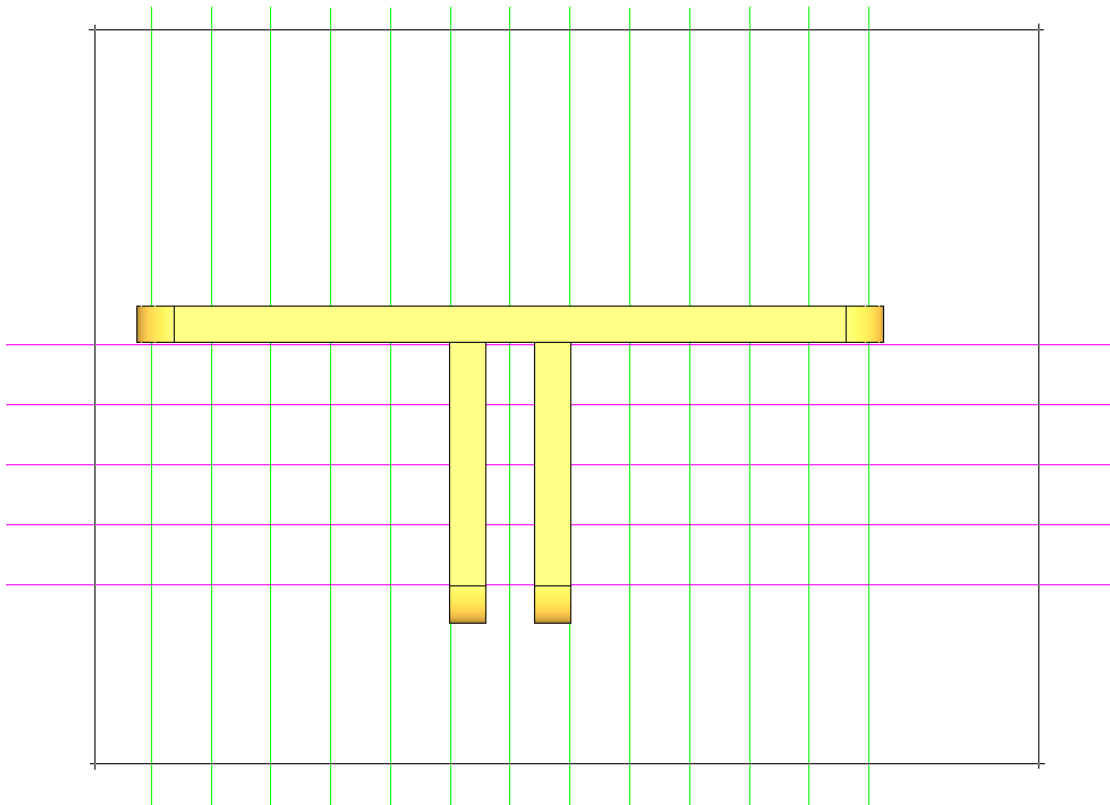


Congratulations! You're done!

Notes on selecting multiple entities in NUCLEO Fixture

Sometimes, you may want to delete all items using a “window” and the mouse. For example, if we wanted to delete multiple fixture lines we could do so like this:

When the system command prompt is active, select all the lines by windowing them with the mouse; click in an area top right where there is no geometry, then drag down and left so you get a selecting window similar to below:



(The corners of the selection box will cross over, this mean “whatever is touched rather than enclosed by the selection box” is selected). When all lines are highlighted, press **Enter**.

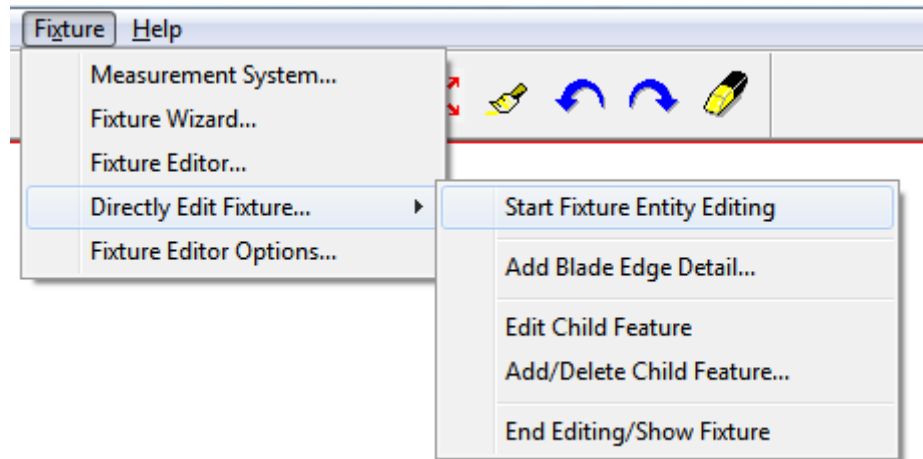


Directly Modifying a Fixture

Note: This process is outlined using OPTCAM Classic V7. The basic sequence has not changed in NUCLEO Fixture.

It is possible to modify the fixture base or blades directly, after it has been created. **Note, however, that if the Fixture Editor is executed again, the changes made here will be deleted.**

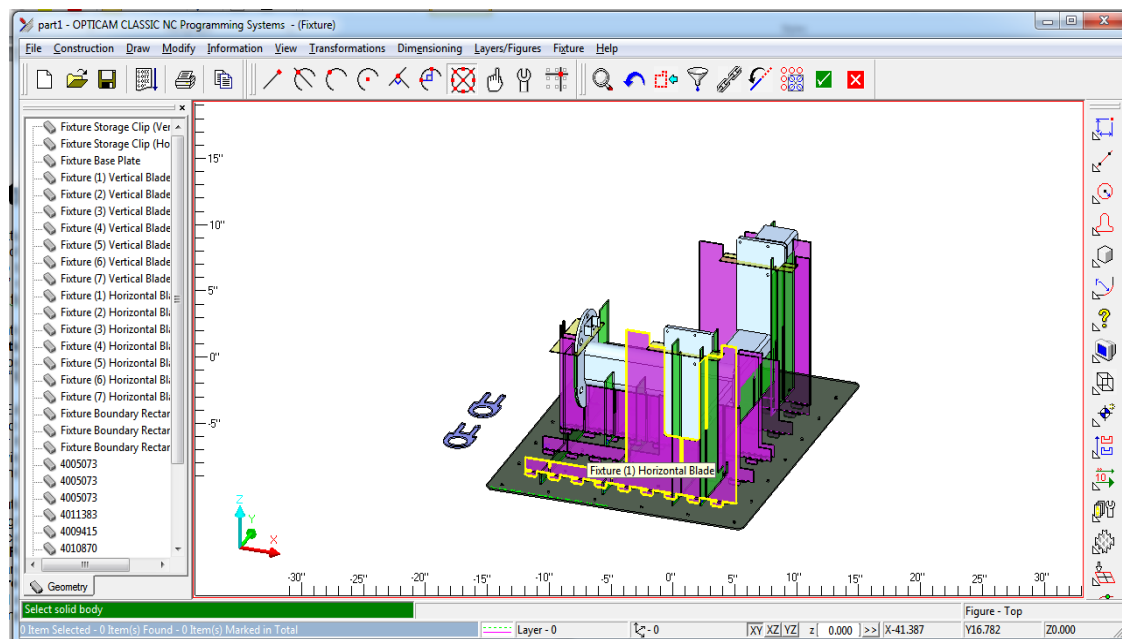
Accessing the Direct Editing options is done via the sub-menu in the main Fixture menu:



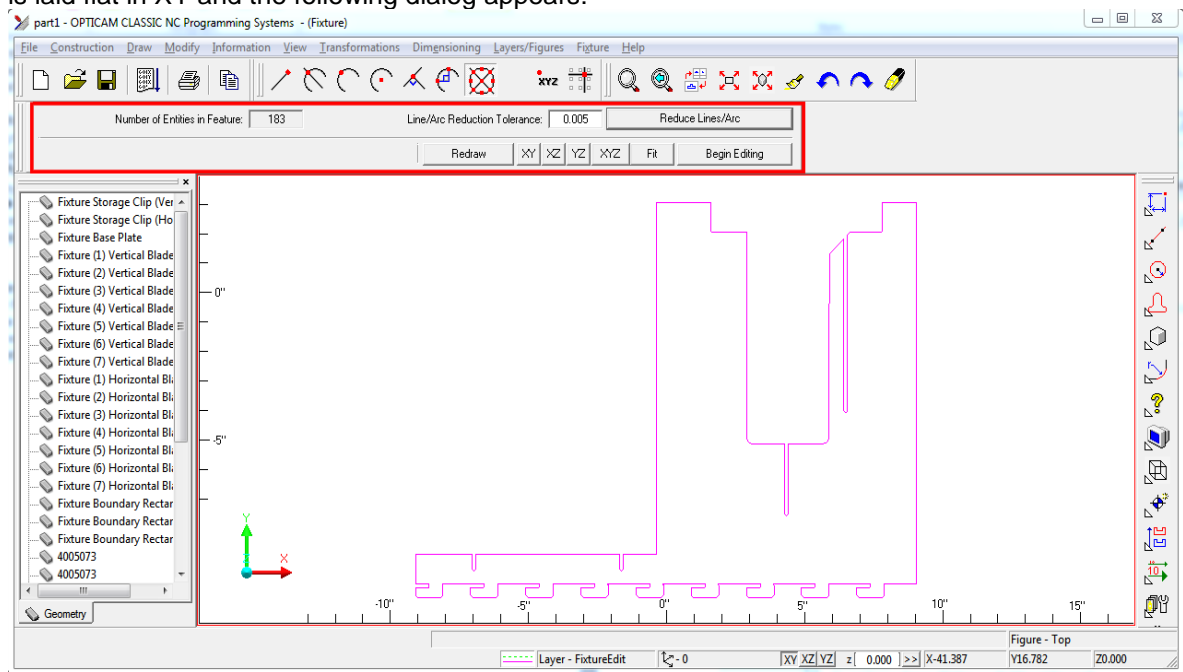
“Start Fixture Entity Editing”

This option puts the system into a special “Figure Edit” mode whereby the user can directly edit the fixture blades or base plate using 2D CAD functions.

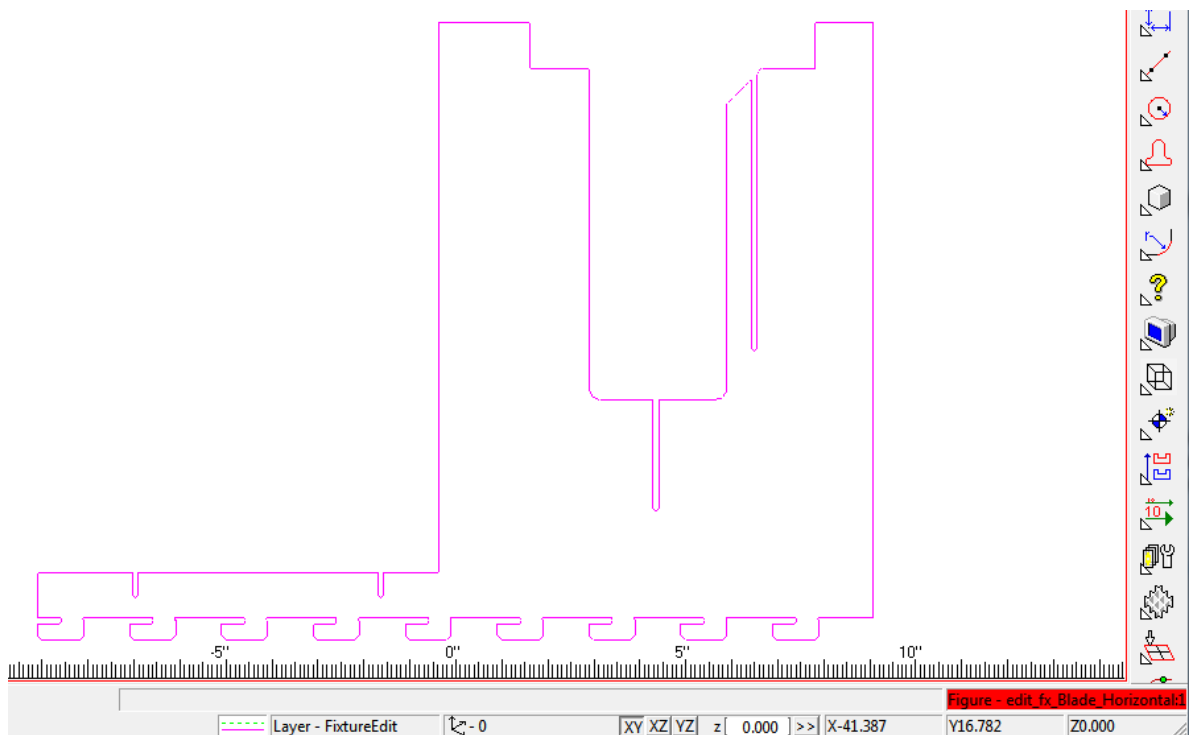
When this option is selected, the user is prompted to **select a solid body** (in this case, a fixture entity):



Once selected (highlighted, then the green tick icon pressed or [ENTER]), the selected blade is laid flat in XY and the following dialog appears:

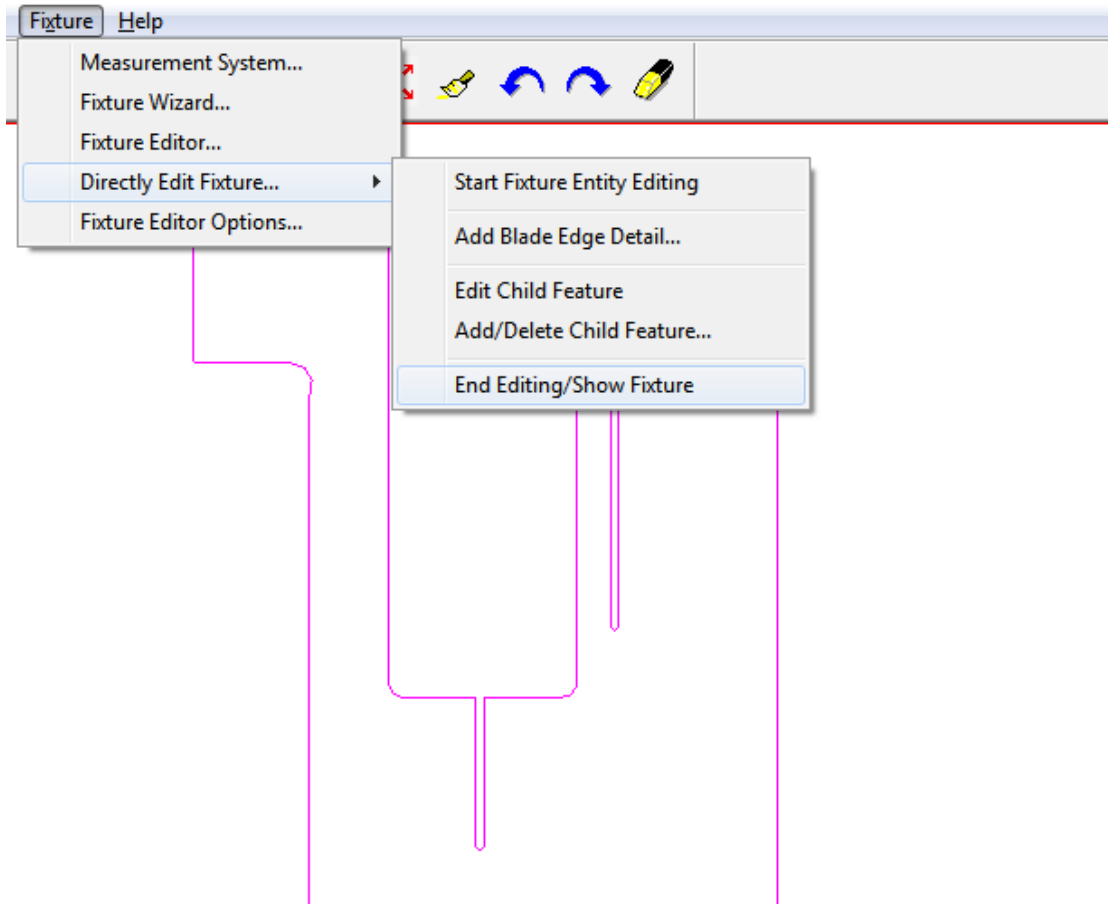


From this dialog, the process or direct editing can be started straight away, or before editing, the option to run a line/arc reduction on the blade is available. Once the **“Begin Editing”** button is clicked, the “Figure Edit” mode is started (this is apparent by the red box in the bottom right of the NUCLEO Fixture window):

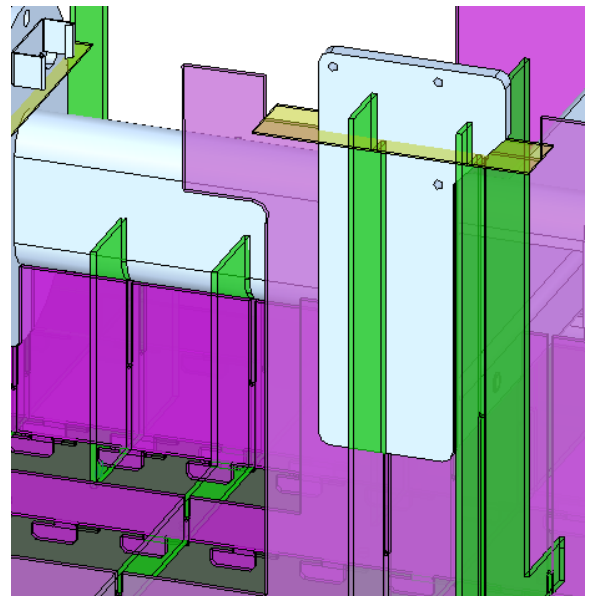


From this point, the user can modify the blade edge geometry in any way, using the standard CAD functions. Once the modifications are complete, the fixture is automatically updated upon selecting either:

- 1) The menu option **“End Editing/Show Fixture”** or:
- 2) Selecting the **“Fixture Editor”** menu option.



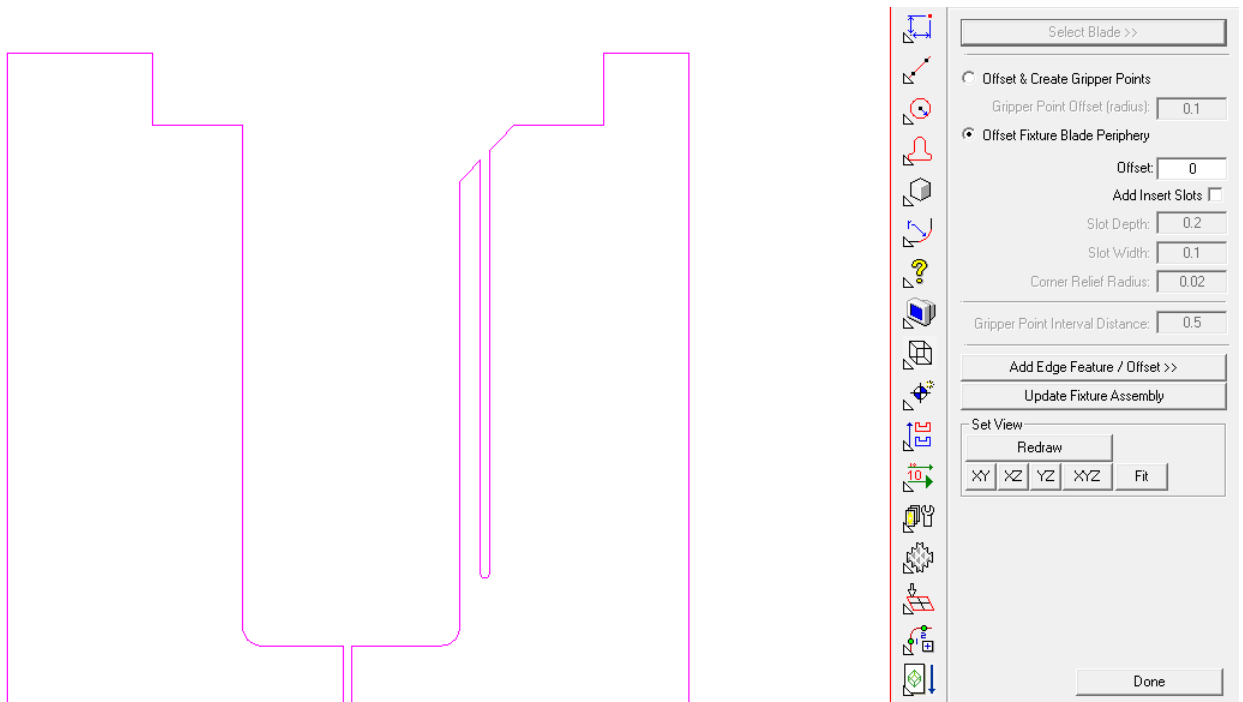
Once the fixture is updated, the blade will be regenerated with the changes (as long as the blade geometry is still a closed shape). The blade will be displayed in a slightly different hue to show it has been modified.



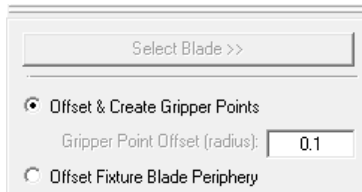
“Add Blade Edge Detail”

This option assists in adding edge detail to Blades, such as “Gripper Points”, “Low Friction Insert Slots” or simply a partial blade offset. In some cases, it is desirable to only have these edge features along parts of the blade.

When the option is selected from the menu, the following dialog appears. The blade to edit is selected using the “**Select Blade**” button – again the user is prompted at the bottom left of the NUCLEO Fixture screen – to select a **solid body**. Once selected, the blade is laid flat in XY and the other option buttons are enabled:

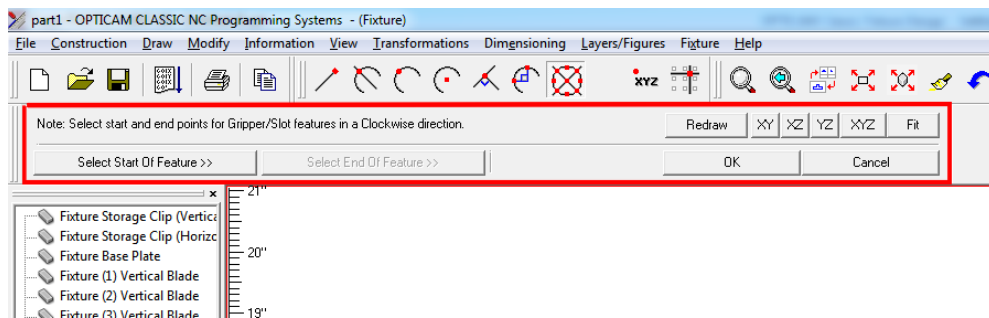


The type of edge detail is now set – for example – “Gripper Points”:



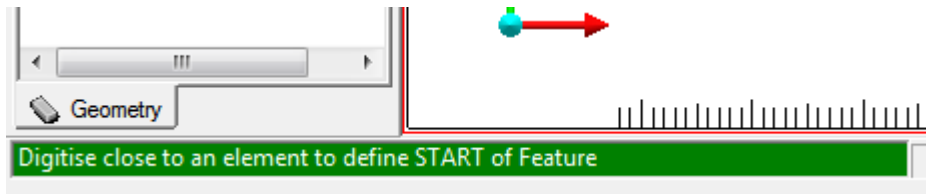
... then the “**Add Edge Feature / Offset**” button is clicked:

Another dialog appears:

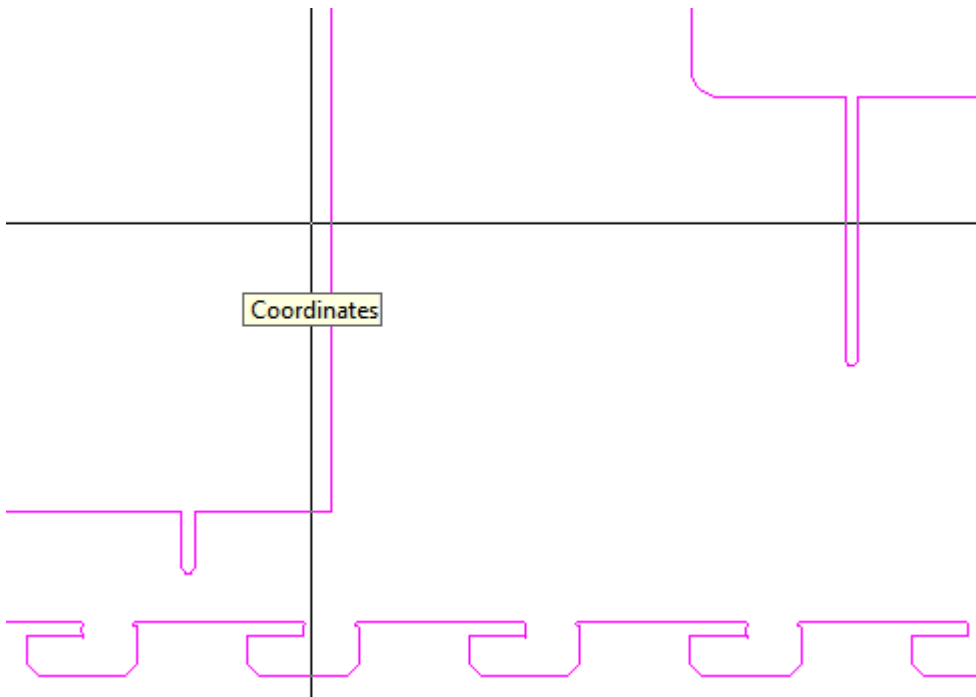


Note that all Blades are defined clockwise so ensure to pick the start and end point correctly.

From this dialog, the “**Select Start of Feature**” button is pressed and the system prompts:

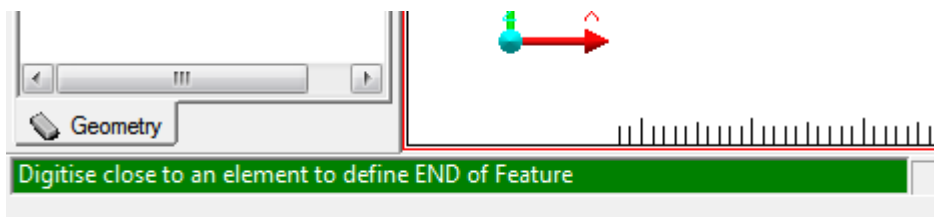


Using the cursor, the position to start point is digitised:

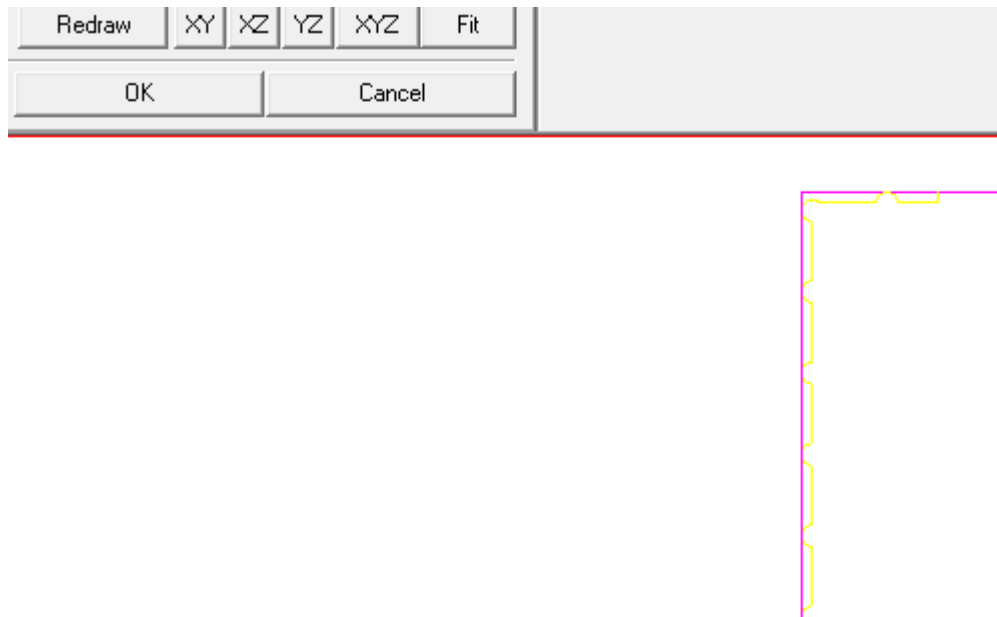


Note it is better to digitise “Close” to the necessary element than on the element – for this purpose the “Coordinate” snap mode is defaulted.

Once the start point is selected, the end point is prompted:

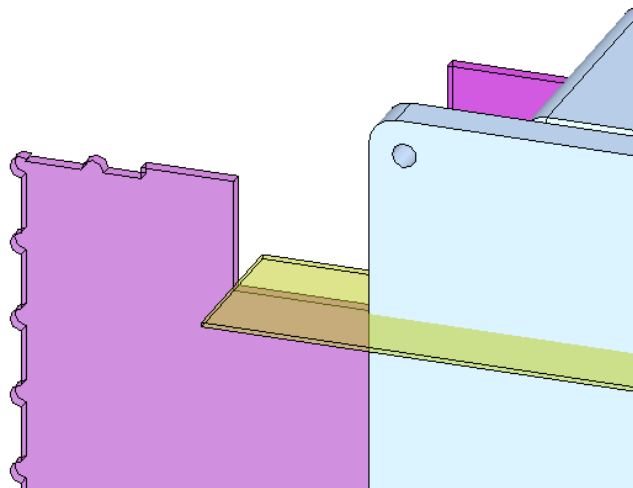


Once complete, the edge detail is shown in yellow on-screen:



If the detail looks correct, the “OK” button is clicked and the user is returned to the previous dialog. The changes to the blade will be made in the fixture by either:

- 1) Clicking the “**Update Fixture Assembly**” button or:
- 2) Clicking “**Done**”.



“Edit Child Feature”

This option is used exclusively when in “Figure Edit” mode – i.e., when directly editing a blade or the base plate. It allows “Child” features in a blade or the base plate to be directly edited using the 2D CAD functions. “Child” features are - for Blades - apertures for reducing weight or to allow better weld access and for the base plate, the bolt holes and tab feet apertures.

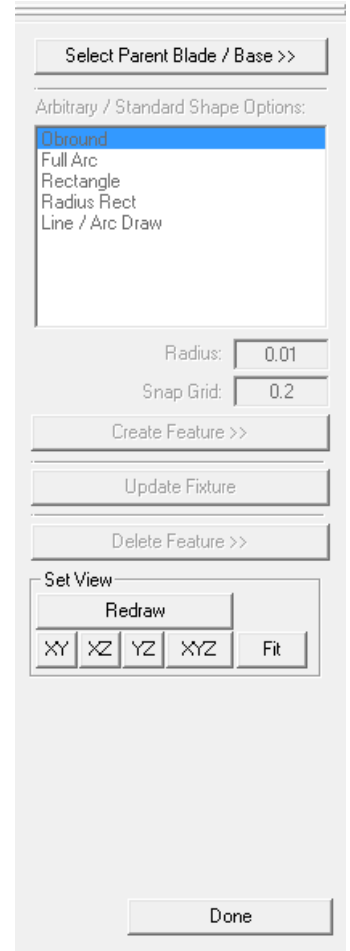
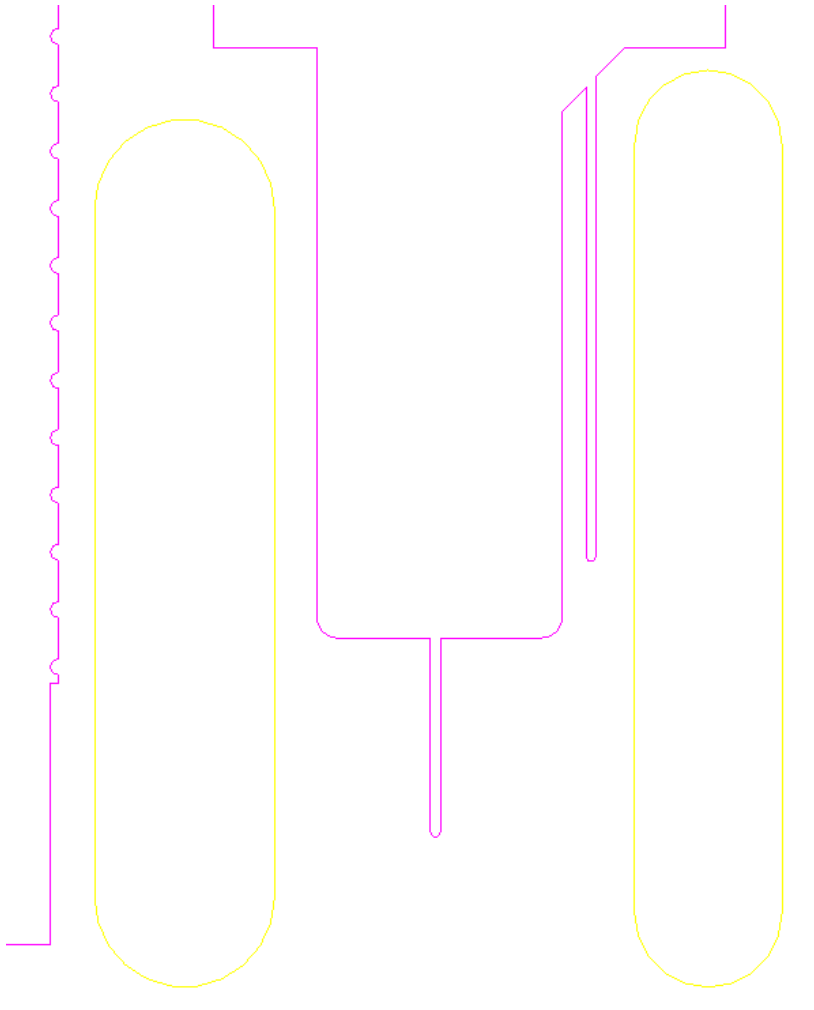
Once editing is complete, the changes are updated upon selecting:

- 1) The menu option “**End Editing/Show Fixture**” or:
- 2) Selecting the “**Fixture Editor**” menu option.

“Add/Delete Child Feature”

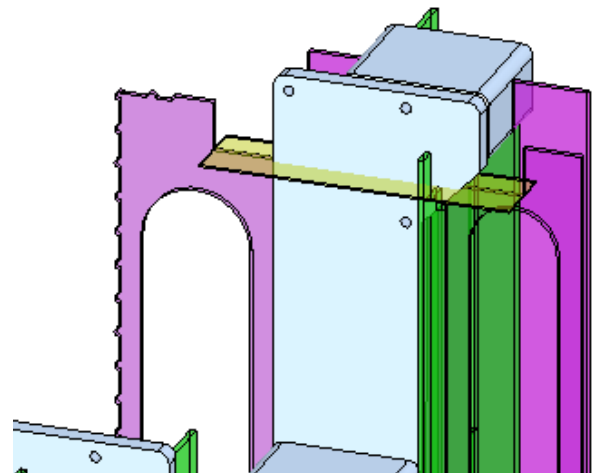
This option enables the definition (or deletion) of aperture features for the base plate or any of the blades. Upon selecting the “**Select Parent Blade / Base**” button, the user is prompted to **select the solid body**.

Once selected, the blade or base plate is laid flat in XY. In conjunction with the “**Arbitrary / Standard Shape**” selection list box, The “**Create Feature**” button allows standard or user defined profiles within the boundary of the blade/base plate:

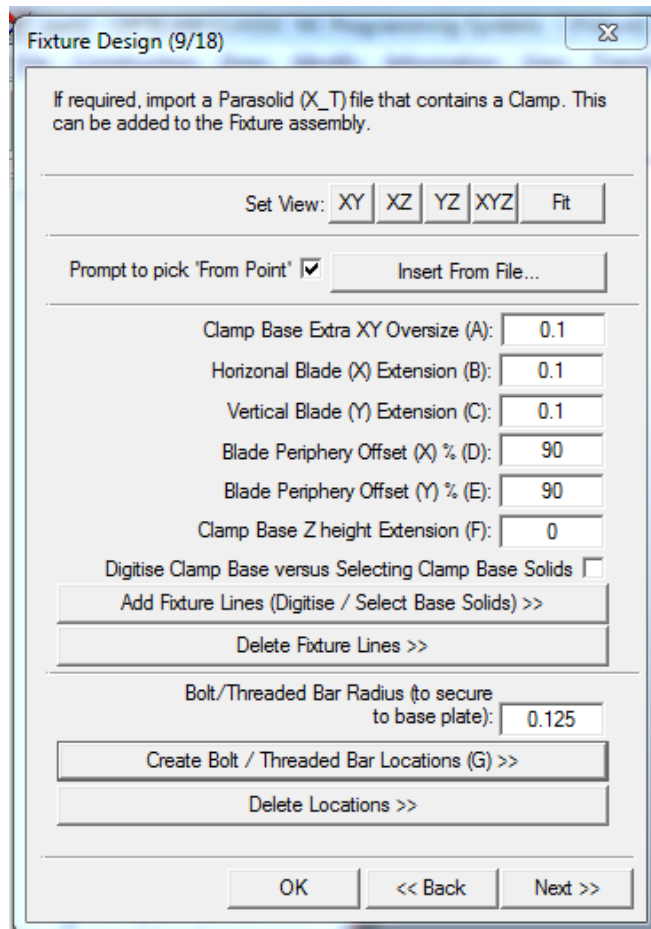


Multiple features can be added and of mixed types – they may also overlap each other to create a custom or quick “cut out” profile.

Once the features are defined, the command is ended to create them (normally, [ENTER] or by clicking the Green Check icon). To update the fixture, the “**Update Fixture**” button is clicked or “**Done**” (any changes will result in a prompt to update the fixture).



Using Clamp CAD Files

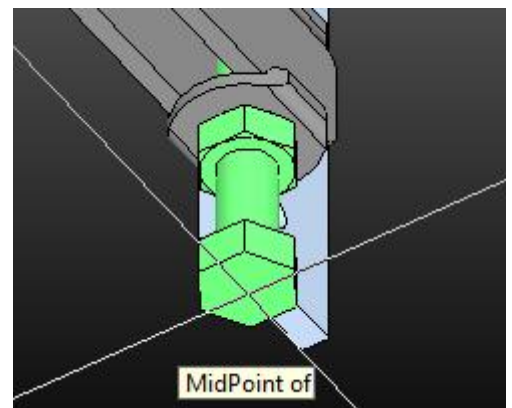


Note: This process is outlined using OPTCAM Classic V7. The basic sequence has not changed in NUCLEO Fixture.

Popular Clamp models are available online from major suppliers as X_T (Parasolid) CAD files. With a little preparatory work, it is very easy to include such models into the fixture assembly. It is recommended that the CAD files are:

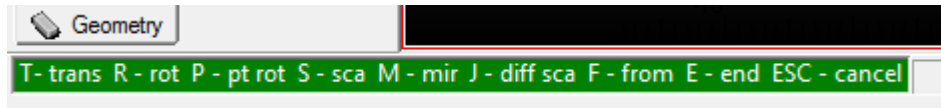
- 1) Saved with the Clamp located upright with the base XY-planar, and:
- 2) With the clamp-to-part contact point set at X0 Y0 Z0 within the CAD File to avoid an extra step upon importation into the fixture assembly.

To insert a clamp, the “**Insert From File**” button is clicked. If the checkbox “**Prompt to pick ‘From Point’**” is checked, it is assumed that the insertion point for the fixture CAD model is set to X0 Y0 Z0 at clamp-to-part contact point (for example, mid point of the green cylinder/bolt in the picture to the right).



If this checkbox **is** checked, this assumption is not made and an extra few steps are necessary during the clamp model import:

- 1) The model is imported and the view set to a suitable angle so to make the selection of the clamp-to-part contact point easily selected. As in the previous image, an example would be the snap point of mid-point of a suitable face.
- 2) After selecting the snap point, the command line prompts:



By pressing “T” or by selecting the first icon in this toolbar selection, the clamp is set to “translate” mode:

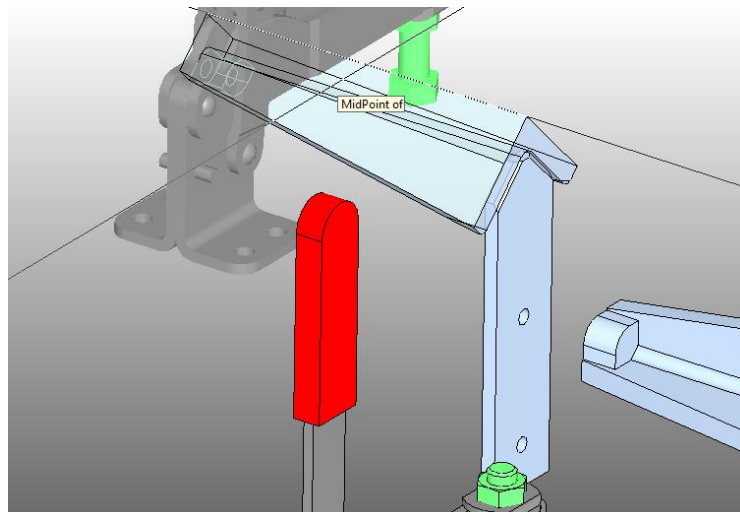


If the “**Prompt to pick ‘From Point’**” option **was not** checked, the system will skip these two previous steps. The next prompt is:



- ➔ If the “**Prompt to pick ‘From Point’**” **was** checked, it is necessary to rotate the view from the “underside” view to a more normalised isometric view – so that the clamp-to-part contact position can be selected. To do this, the **SHIFT** key is depressed whilst the left mouse is used to manipulate the view.

Next, a suitable clamp-to-part contact point is selected on the model:



(the clamp will be “ghosted” as it is moved about – it may not appear to be at the intended location at times, but assuming the “from” position is correct initially, it will translate to the correct point after a suitable snap position is selected).

Once positioned, the clamp may be rotated or repositioned. The prompt line will display short help, with keyboard shortcuts such as “**T**” for Translate, “**R**” for Rotate etc. Pressing the **T** key will allow the clamp to be further translated, and this is often necessary if the “snapped point” chosen is the middle of a circle or edge. In this case, it is common to want to move the clamp *incrementally* from the last snapped point.

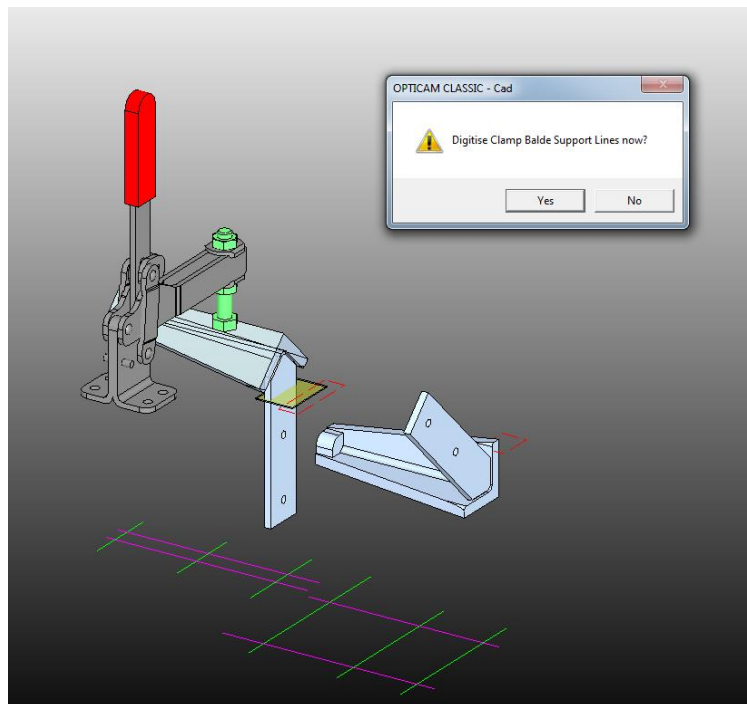
To do this, when prompted for the translation position, the “**D**” (delta) keyword is used. For example, typing **DX1 DY-1** followed by **[Enter]** will move the clamp positively in X and negatively in Y by 1 inch/millimeter.

This can be repeated as many times as needed, simply by translating again (“**T**”) then entering the DX, DY and any DZ values plus **[Enter]**.

Likewise, when prompted for a translate-to position, an explicit X YZ value can be typed in, if known, for example **X0 Y6 [Enter]**. If any of the axes are not stated, the current value will remain, so to just change the X and the Z of a clamp, it is possible to enter **X-12 Z6**, for example.

Using the same methodology, it is also possible to return a position from two snap points; simply type “**X**” then pick a snap point, then type “**Y**” and pick another, and “**Z**” if needed – followed by **[Enter]**. This will compute the position from multiple snaps.

Once all translation/rotations are complete, press **[Enter]** to exit the insertion mode. Next, the following prompt appears:



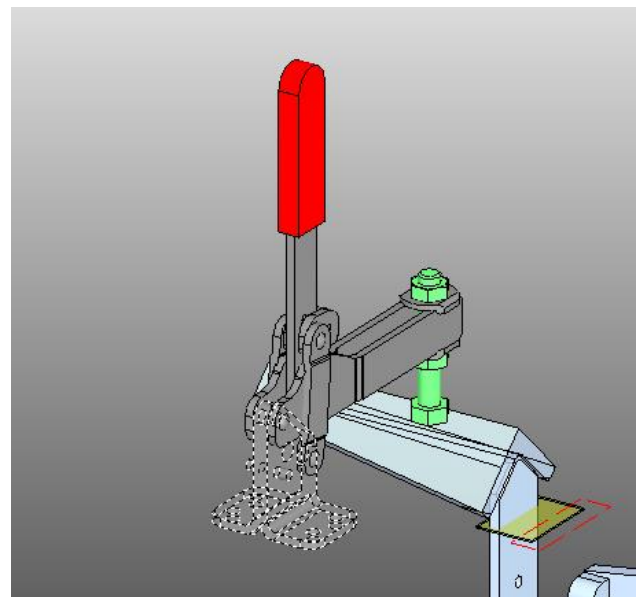
This simply automates the next step in the clamping process – so it is normal to click “Yes”.

The command prompt changes to:



Upon selecting the correct clamp base geometry, the selection on-screen is dotted:

Pressing **[ENTER]** or the usual “green tick” icon ends the selection process.



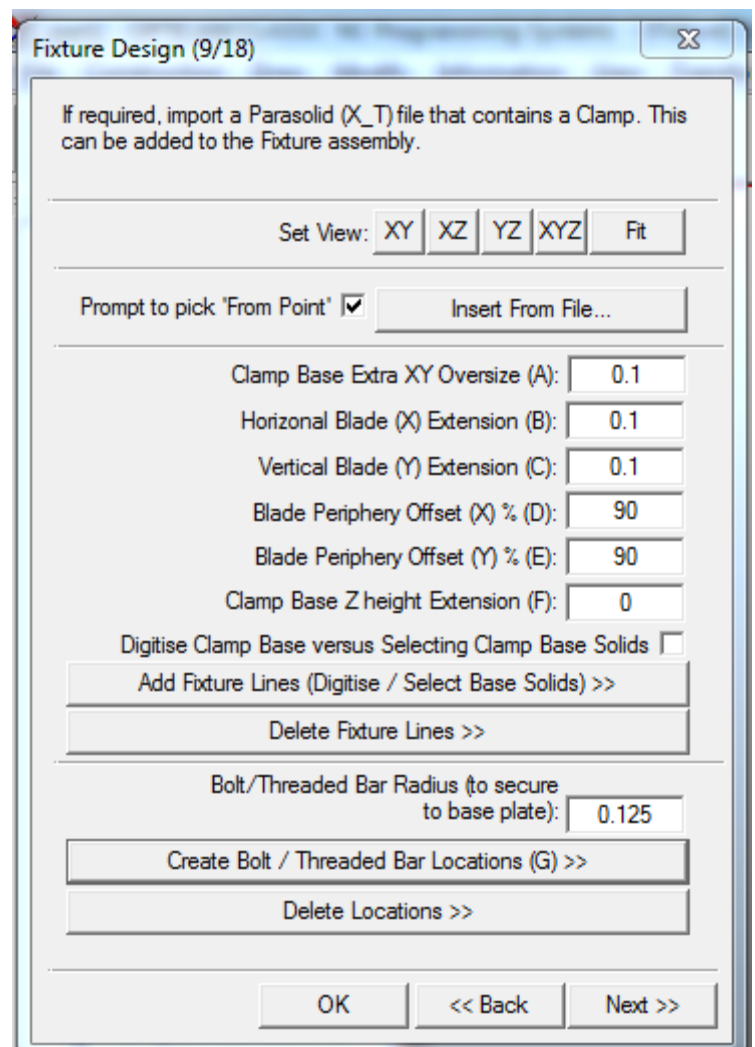
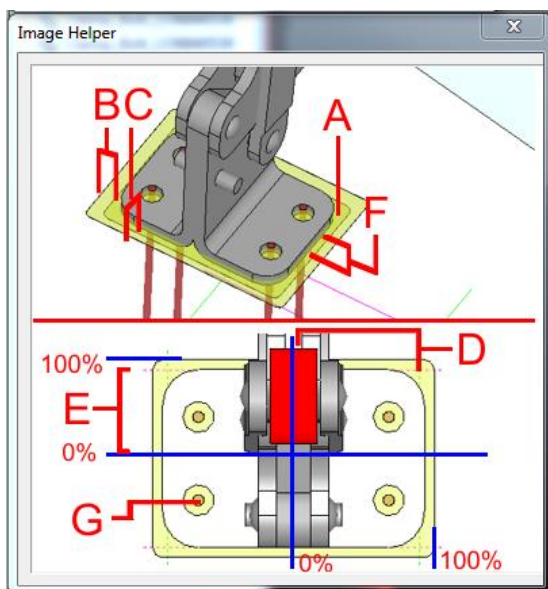
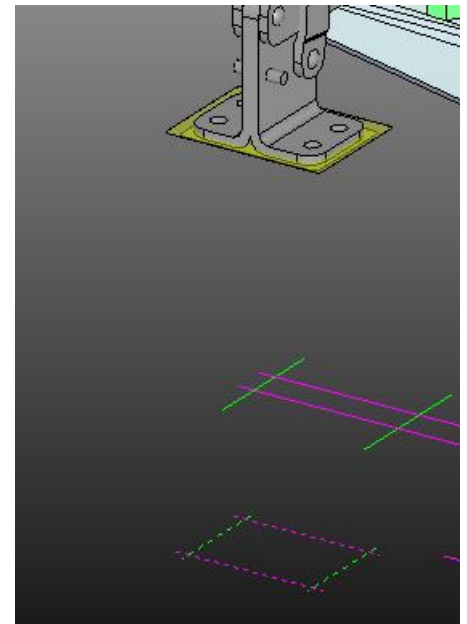
Based on the default entries in the Clamp Import Wizard page, some necessary boundary solids are created, along with the fixture blade position lines for the clamp.

The clamp position is now complete – additional clamps may be added or deleted as necessary. When the fixture assembly is generated, it will include the “clamp tower” blades also.

A description of the entries on the “Clamp Import” Wizard follows:

As usual, the image helper should assist in describing the values.

It is normal to add a small oversize for the clamp base (some clearance) – this is added in field **(A)**.



Fields **(B)** and **(C)** specify how much larger the fixture lines (to define the “clamp tower”) should be, so to correctly encapsulate the clamp base. This is added to the oversize **(A)**.

The “**Base Periphery Offset**” fields specify how much distance – as a percentage from the centre of the clamp base – the blade fixture lines are drawn. “Zero” is at the centre point of the base, 100% being respectively to $-X$, $+X$, $-Y$ and $+Y$ from this point.

Field **(C)** determines the depth of the recess into which the clamp base sits.

Finally, it is necessary to bolt the clamp and “clamp tower” to the base plate. Using the options below, the necessary locations are digitised:



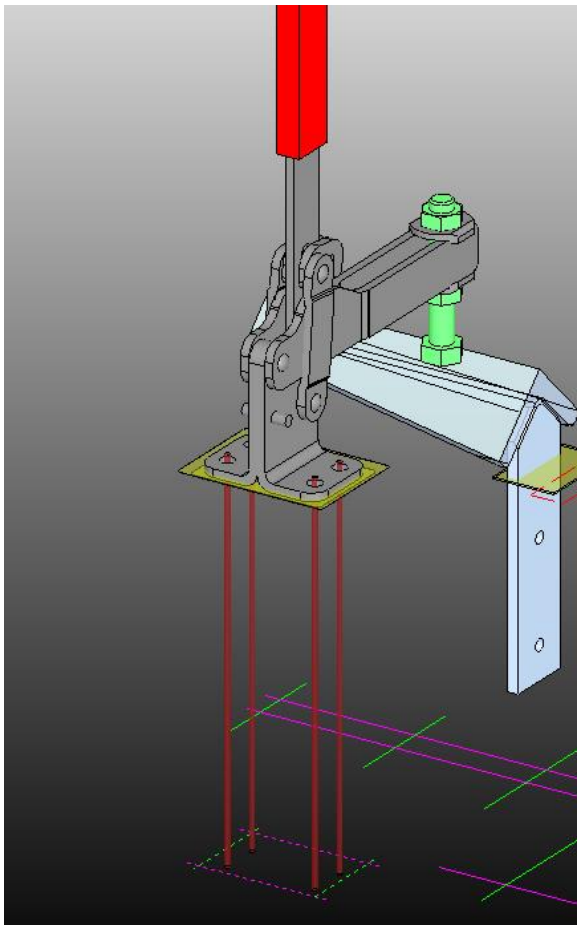
Upon clicking the “**Create Bolt / Threaded Bare Locations (G) >>**” button, the system defaults to “centre-of” snap mode:



Once a suitable centre point is selected, a representation of the bolt/bar is added to the assembly.

The process is repeated until the command is ended.

When the base plate is exported with the rest of the fixture assembly, it will include bolt hole locations at the intersection of the bolt solids and the base plate.



These can be optionally tapped or simply used at through holes.

