

ProNest[®] CNC

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Introduction

EDGE Connect CNC operators can use ProNest CNC in Phoenix to decrease scrap and improve cut quality. ProNest CNC nests parts efficiently on the plate (also called a sheet or workpiece) and applies the best cutting parameters for the job.

Base features and optional modules

Base module: ProNest CNC

The base module of ProNest CNC is pre-installed on every Hypertherm EDGE Connect CNC. The base module lets you add one CAD or Shape Library part to the part list and then create a nest that has one or more copies of that part.

Optional module: True Shape Nesting

With the True Shape Nesting module, you can:

- Build a part list that contains many CAD files and nest those different parts together on the sheet.
- Nest parts on a remnant that you traced in Phoenix.



To learn more about the True Shape Nesting module, contact your table manufacturer for a free 30-day trial.

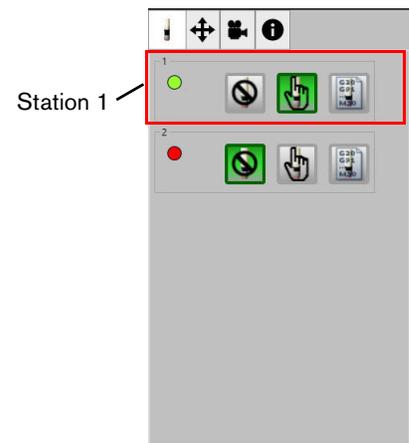
Before you begin

Before you use ProNest CNC, it is important to know these guidelines:

- You can only create one nest at a time.
- ProNest CNC only supports station 1.

If you have questions when you do the steps in the next section, refer to:

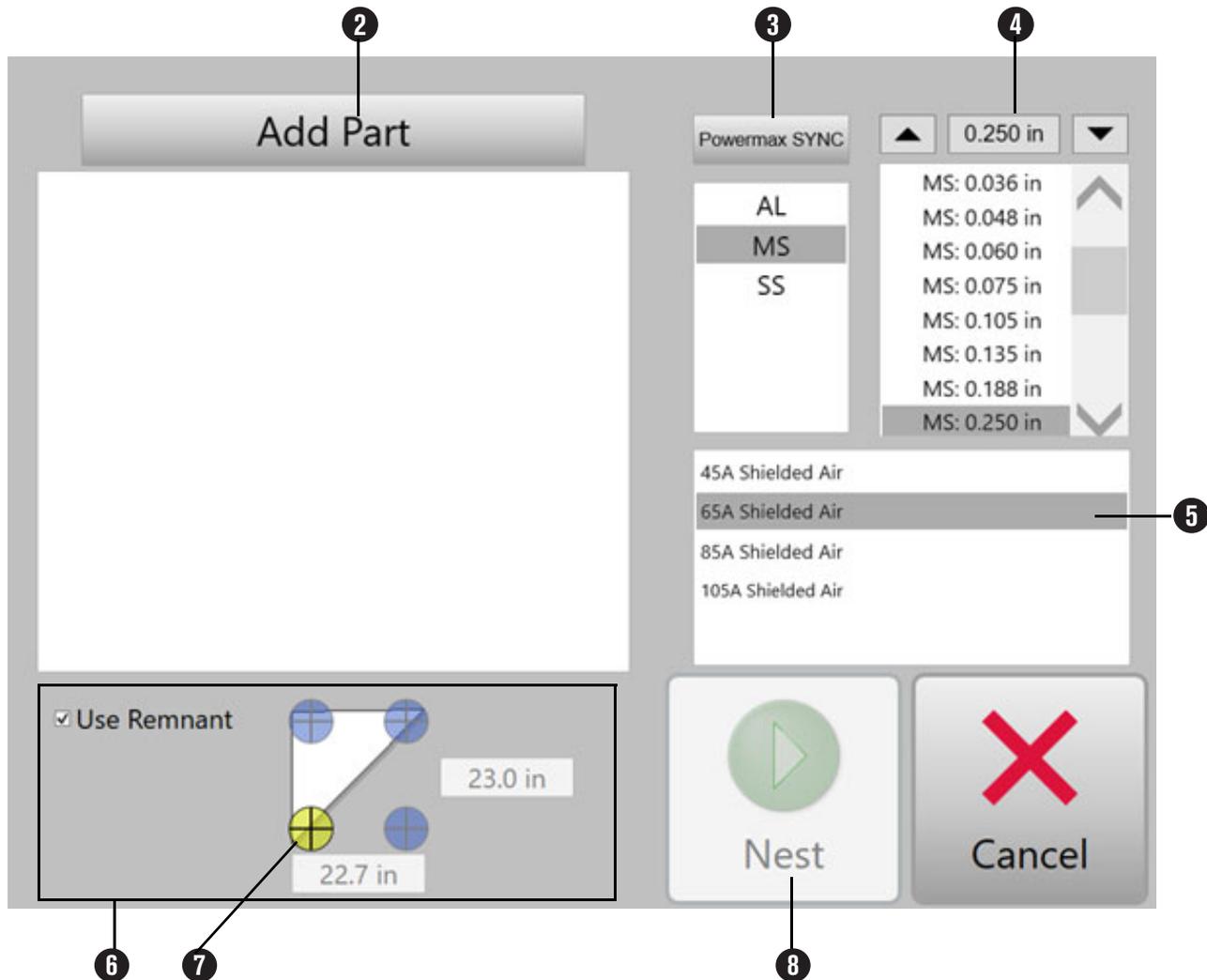
- [Frequently asked questions](#) on page 15.
- [Errors and troubleshooting](#) on page 15.



Basic workflow

ProNest CNC creates one nest at a time. You can add up to the maximum number of parts to fit on a plate, nest the parts, create a part program for that nest in Phoenix, and then cut it.

Figure 1 – Overview of steps to use ProNest CNC



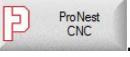
Step	Summary	For detailed steps, refer to:
1	Open ProNest CNC. Not shown in Figure 1 .	page 5
2	Add a CAD file or Shape Library part.	page 5
3	Select the machine.	page 7
4	Select the material type and thickness.	page 8
5	Select a class.	page 9
6	Select what to nest on.	page 12
7	Set the home position.	page 13
8	Create the nest.	page 13
9	Cut or save a nest. Not shown in Figure 1 .	page 14

Step 1: Open ProNest CNC

In Phoenix, you can open ProNest CNC with or without a Shape Library part.

Without a Shape Library part

If you only want to add CAD part files to your nest:

1. On the Phoenix Main screen, select the **Shape Manager** soft key.
2. Select the **ProNest CNC** soft key .

With a Shape Library part

If you want to add a Shape Library part to your nest, select the simple shape first and then open ProNest CNC.

1. On the Phoenix Main screen, select the **Shape Manager** soft key.
2. Select a library shape.
3. Enter properties for the shape.
4. Select **OK** .

ProNest CNC opens with the shape already in the part list.

Step 2: Add parts

ProNest CNC lets you nest on external CAD files and one Shape Library part at a time.



With the True Shape Nesting module, you can add many CAD files and only one Shape Library part to your nest.

CAD files

You can load new CAD files into Phoenix if the files are saved:

- On a USB memory stick that is plugged into the CNC
- In a network folder (if your CNC has an Ethernet or wireless connection)

Supported CAD file formats

You can nest these CAD file formats with ProNest CNC.

- *.dxf
- *.dwg
- *.dgn
- *.cam

Rules for CAD files

1. Only include cut geometry.

When you add a CAD file in ProNest CNC, all geometry found in the file is cut. Even if there is a designated layer for marking, the geometry on that layer is still cut. Make sure that your drawing only has cut geometry.

2. Multi-part CAD files are not supported.

If a CAD file contains separate parts, that file is not supported by ProNest CNC. Only single-part CAD files are supported. You cannot add CAD files that contain entire nests of parts in ProNest CNC.

3. Be sure file units of the CAD file match the units used in Phoenix.

- ❑ When the CNC is set in **English** units, use CAD files that are drawn in inches. When the CNC is set in **Metric** units, use CAD files that are drawn in millimeters.
- ❑ To view the units used by the CNC, select **Setups** on the Main screen in Phoenix.
- ❑ If units are different, your parts will be too big or too small. For instance, if ProNest CNC is set to use **English** units and you import a part that was drawn in millimeters, the part will be too big.

If you have CAD files with incorrect units, correct the problem in the CAD software instead of adjusting the CNC's units.

To add CAD files to the part list:

- 1.** In ProNest CNC, select the **Add Part** button. Refer to [Figure 1](#) on page 4.
- 2.** Browse to the folder location where your CAD files are saved. This can be a USB memory stick or a network folder.
- 3.** Select a CAD file and then select **Open**. The CAD file is added to the part list.
- 4.** Use the **+/-** buttons to adjust the quantity of the part. You can also select the number to the left of the part and type in a quantity.

If you have the True Shape Nesting module, you can do these steps again to add more CAD files.

Shape Library parts



You can only add one Shape Library part to the part list, even if you have the True Shape Nesting module.

To add a Shape Library part to the parts list:

1. Do the steps on [page 5](#) to open ProNest CNC with a Shape Library part.
2. When ProNest CNC opens:
 - ❑ If you are nesting on a remnant, make sure that the **Use Remnant** checkbox is selected.
 - ❑ Use the +/- buttons to adjust the quantity of the library part.



You can also select the number to the left of the part and type in a quantity

You can also add CAD files to the part list if you have the True Shape Nesting module.

Step 3: Change the machine

ProNest CNC has a current machine which controls the materials and performance applications that are available. This machine should be the same as the cutting system that will be used to cut the nest.



In most cases, ProNest CNC's current machine will be the same as your cutting machine hardware and you will not need to make changes.
ProNest CNC only supports Station 1.

It may be necessary to change the current machine for applications where:

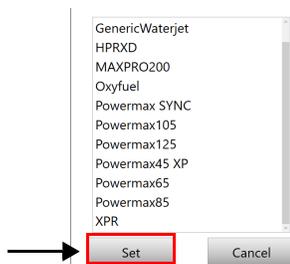
- There are two or more cut tools (for instance, plasma and oxyfuel).

AND

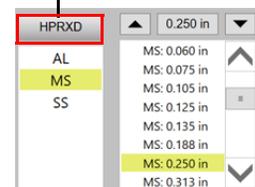
- You want to create a nest for a specific machine process.

To change the current machine:

1. In ProNest CNC, select the **Machine** soft key.
2. Select a machine from the list, then select **Set**.



Machine soft key



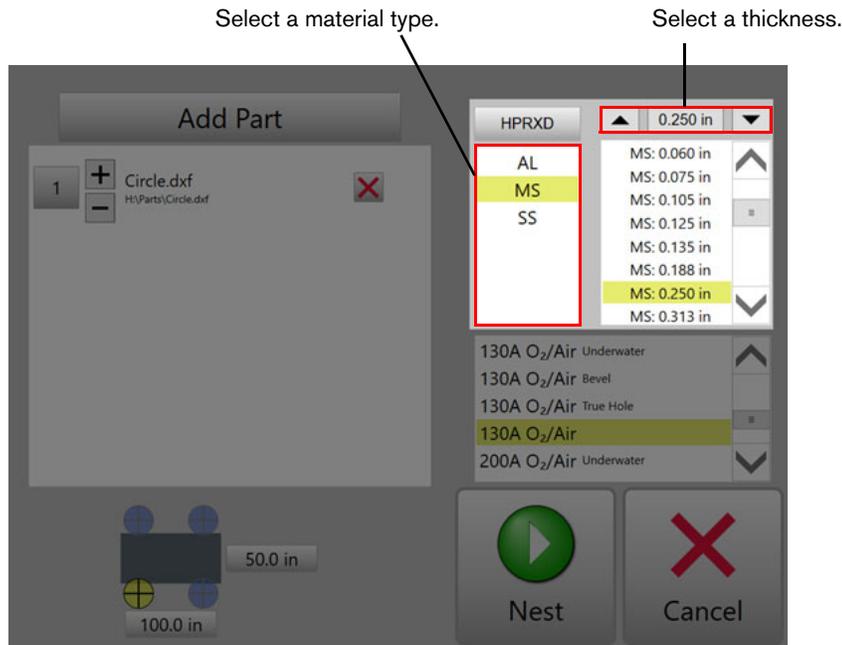
Step 4: Select a material type and thickness

ProNest CNC assigns different SureCut™ parameters (such as feedrate, kerf, leads, and separation distances) on the nest for different materials.

Select the correct material type and thickness for the nest. Refer to the image below.

The materials that you can select include:

- MS = Mild steel
- SS = Stainless steel
- AL = Aluminum



After you select a material type, select the thickness of your plate.

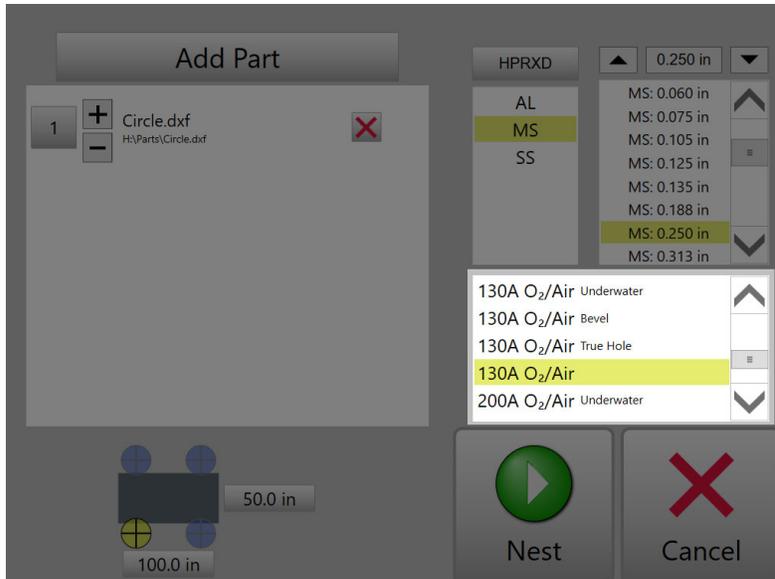
- Use the arrows or scroll bar to page through the list in the top right.
- or-
- Select the thickness box at the top of the pane and type in a thickness.



If you are cutting with an oxyfuel tool or generic plasma, then material type and class are not available and it is only necessary to select a thickness.

Step 5: Select a class

In ProNest CNC, class is used for plasma applications to apply special cutting parameters for the consumables that will be used to cut a nest. Class is also used to apply performance applications such as True Hole® and other technologies to parts on the nest.



The list of available classes changes for different machines (HPR, XPR, etc.), material types, and thicknesses.

For plasma systems, you can select the amperage and cutting gas/shield gas in ProNest CNC with the class. When you select an amperage and gas, you specify which consumables (electrode, nozzle, etc.) are used to cut the parts on the nest.

Here are things to know when you select a class:

- It is usually faster to cut with a higher amperage than with a lower amperage.
- A lower amperage with a slower cut speed usually gives a better quality cut than a higher amperage.

Refer to [Table 1](#) on page 10 to learn more about the pros and cons of different classes.

Table 1

Class	Description	Pros	Cons
True Hole®	<ul style="list-style-type: none"> Gives bolt hole quality with virtually no hole taper and minimal ding on mild steel. Applies automatically to holes with eligible ratios of material thickness to hole diameter. 	<ul style="list-style-type: none"> Automatic "bolt-hole" quality Removal of hole taper Decreased ding that is biased to the outside of the hole 	<ul style="list-style-type: none"> Slower than a standard cut Slightly longer cut length (because of adjusted leads) Only holes with eligible thickness to diameter ratios are supported.
Fine Feature	<p>Helps to give a better finish on the cut surface, a sharper top edge, and tighter control over angle deviation.</p> <p>Note: XPR does not have a Fine Feature class.*</p>	<ul style="list-style-type: none"> Best finish on the cut surface Sharp top edge quality Tight control over angle deviation 	<ul style="list-style-type: none"> Slower than a standard cut
HyDefinition inox (HDi) 60	<p>Supplies high quality cuts with minimal dross.</p> <p>Note: XPR does not have the HDi class.*</p>	<ul style="list-style-type: none"> Sharp top edge quality Shiny surface finish Superior angularity with reduced angle variation Less dross 	<ul style="list-style-type: none"> Only for thin material Only available at 60 A F5/N2 gas is necessary
Bevel and True Bevel	<p>Identifies to ProNest CNC that Hypertherm bevel consumables (shield, nozzle, electrode, and so on) are in use.</p> <p>Note: Bevel cuts (angled non-perpendicular edges on parts) are not possible with ProNest CNC parts. Selecting a class with Bevel or True Bevel only identifies that Hypertherm bevel consumables will be used to cut the nest.**</p>	<ul style="list-style-type: none"> Better consumable life when cutting non-bevel (perpendicular) 	<ul style="list-style-type: none"> Only available at some amperages A separate consumable set is necessary
Moving Pierce	<p>Uses PowerPierce® technology to extend stainless steel pierce capability for the HPR800XD to 4 inches (100 mm) and for the HPR400XD to 3 inches (75 mm).</p> <p>Note: XPR does not have a Moving Pierce class.*</p>	<ul style="list-style-type: none"> Eliminates the need for pre-piercing Can pierce thick stainless without an edge pierce Can pierce interior profiles with plasma (as long as the hole is sufficiently large and has a linear lead-in) Keeps the torch away from molten material 	<ul style="list-style-type: none"> Linear lead-ins are necessary Causes molten slag ("rooster tail") that can spatter onto adjacent parts High amperages are necessary

* XPR cut charts already give superior cutting results on mild steel, stainless steel, and aluminum throughout all thin and thick processes.

** For XPR users, True Bevel is shown but not supported with ProNest CNC. This is because all XPR consumables are bevel consumables.

Class	Description	Pros	Cons
Thick Pierce	Uses special piercing commands in EDGE Connect for the HPRXD Thick Pierce. Note: XPR does not have a Thick Pierce class.*	<ul style="list-style-type: none"> Can pierce thick material without an edge pierce 	<ul style="list-style-type: none"> Not available for Stainless Steel or Aluminum
Underwater	Significantly decreases the level of noise and smoke generated by normal plasma cutting, as well as the glare of the plasma arc. Hypertherm has developed underwater cut charts for mild steel at 80 A, 130 A, 200 A, 260 A, and 400 A. Select this class if the nest will be cut with underwater plasma.	<ul style="list-style-type: none"> Decreased noise, smoke and glare compared to normal plasma cutting Decreased heat distortion and warping Parts can be moved immediately after they are cut Available at many amperages 	<ul style="list-style-type: none"> Slower than a standard cut Cut charts do not support Stainless Steel or Aluminum Not compatible with True Hole Can cause dross at the bottom of the piece as the molten metal cools down quickly and stays attached to the part
Edge Pierce Only	Indicates that when cutting the selected thickness with the specified amperage and assist gas, you must use edge piercing for parts. This class is not usually used with ProNest CNC. Note: For thicker materials, edge pierces may be necessary when cutting with some amperages and gases. This is because a traditional pierce would not cut through the plate.		
Shielded / Unshielded	For Powermax cutting systems, identifies if the consumable set in use is shielded (with a shield) or unshielded (with a deflector). Kerf compensation and other cutting parameters can change with different consumables. You must select a class that is the same as the consumables in use on your Powermax system to make sure that ProNest CNC uses the correct parameters. Refer to the Powermax Operator Manual. Technical documentation is available at www.hypertherm.com/docs .		
<p>* XPR cut charts already give superior cutting results on mild steel, stainless steel, and aluminum throughout all thin and thick processes.</p> <p>** For XPR users, True Bevel is shown but not supported with ProNest CNC. This is because all XPR consumables are bevel consumables.</p>			

Step 6: Select what to nest on

In Phoenix, there is a default plate (also called a sheet or workpiece) size that is set on the **Setups > Cutting** page. If you change the default plate size in Phoenix, you also change the default plate size in ProNest CNC.

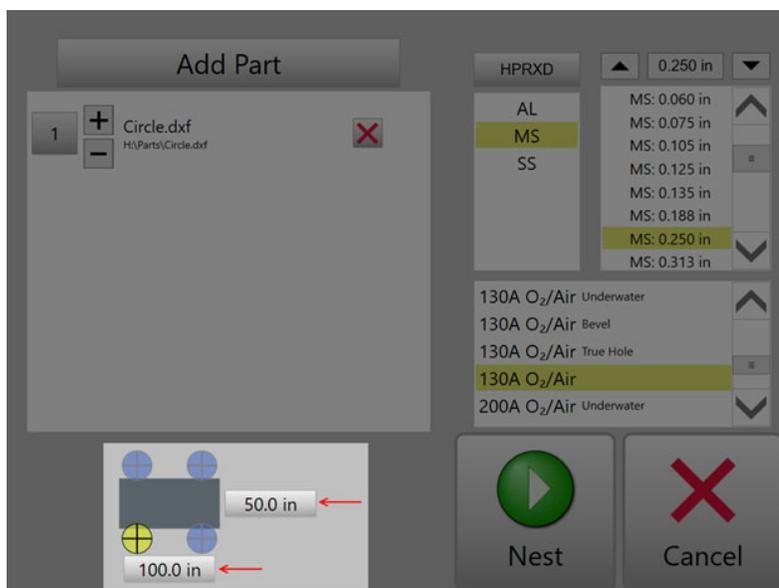
If your plate is not the default size, then you have two options in ProNest CNC:

- For rectangular plates, enter the dimensions of the cutting area.
- For irregularly shaped plates, you can nest parts on the last remnant that you traced in Phoenix if you have the True Shape automatic nesting module.

Rectangular plates

To edit the cutting area for a rectangular plate in ProNest CNC:

1. Select either dimension field in the bottom left.
2. Enter a new value.



3. To set the home position, refer to [page 13](#).

Remnant (irregular) plates

If you have the True Shape automatic nesting module, you can use ProNest CNC to nest parts on the last remnant that you traced in Phoenix.

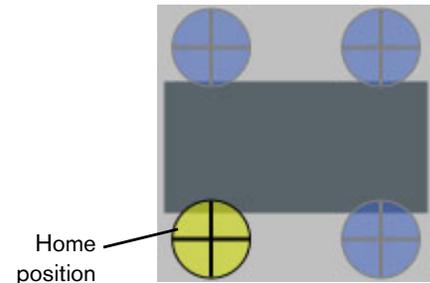
 If you have not yet traced the remnant, refer to [page 17](#) for instructions.

To nest parts on the last remnant that you traced, make sure that the **Use Remnant** checkbox is selected. Refer to [Figure 1](#) on page 4.

Step 7: Set the home position

To change the home position that is used for a nest in ProNest CNC, select one of the four home position indicators on the plate image.

When you change the home position for a given nest, the corner you select becomes the point of origin for automatic nesting. Nesting will begin from this point.



Step 8: Create a nest

You can create one nest at a time in ProNest CNC.



If you select **Cancel** in ProNest CNC before you do this step, you can still view and cut the part that is selected. However, the part will not be processed through ProNest CNC.

When you nest parts, ProNest CNC:

- Processes parts in the part list, which includes adding lead-ins/lead-outs.
- Automatically nests parts on the sheet.
- Applies SureCut™ technology to optimize cut quality.
- Creates the NC program for the nest.

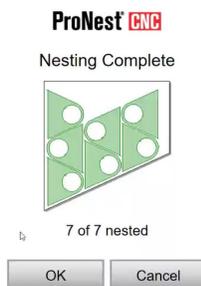
To nest parts:

1. After you complete the steps on the previous pages, select **Nest**



2. Look at the preview of the nested parts on the dialog.

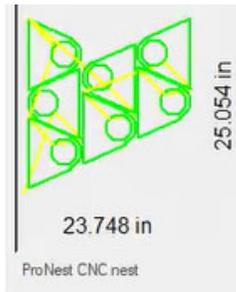
Example of the nest preview dialog



If you want to change the quantity of parts in the nest, select **Cancel** to go back to ProNest CNC.

3. Click **OK** on the dialog to go back to the Main screen in Phoenix.

Example of parts that are ready to be cut on a remnant



The orientation of a simple shape does not always look the same as the preview. ProNest CNC uses the home location you selected and the geometry of the part to adjust the orientation of the shape.



If you selected a cut area that is larger than the plate size in Phoenix, you will see a "Part is Larger than the Plate" warning message.

Step 9: Cut or save a nest

After you create a nest in ProNest CNC, the nested parts become your current part in Phoenix. You can:

- **Cut the nest.** You can interact with this nest just as you would any other part in Phoenix.
- **Save the nest.** To save nests for future use, select **Files > Save to Disk** from the Main screen. Select the **Help** soft key on the Save to Disk screen for more information.



When a remnant is saved to disk, it is no longer recognized by Phoenix as a remnant that you can nest parts on.

- **Copy the nest.** To copy a nest, you must make the same nest in ProNest CNC again. You can only create one nest at a time.

Errors and troubleshooting

Table 2 below has a list of common ProNest CNC errors and possible causes for those errors.

Table 2

Problem	Description	Possible cause
Not all parts are nested	After you select Nest, a message indicates that not all parts in the part list were nested (for example, "2 of 3 parts nested").	<ul style="list-style-type: none"> ▪ Not all parts in the part list will fit on the plate. ▪ Single part is too big to fit on the plate. ▪ Some parts in the part list are invalid, empty, or contain multiple exteriors (see below).
Part is Larger than the Plate	This error/warning can occur after you select Nest .	Cut area selected in ProNest CNC is larger than the plate size in Phoenix.
Empty Drawing File!	This error/warning can occur after you select Nest .	Invalid or empty CAD files were added to the part list.
No parts nested - please check your part	This error can occur after you select Nest . It indicates that none of the parts in the part list are acceptable for nesting. A nest was not created.	<ul style="list-style-type: none"> ▪ Invalid or empty CAD files were added to the part list. ▪ A single part is too big to fit on the plate. ▪ A CAD file containing multiple parts was added.
Parts on the nest are very over or undersized	Check the units used in the CAD file.	See "Rules for CAD files" for more information.

Frequently asked questions

■ How do I view the NC code of a part/nest created with ProNest CNC?

Use Phoenix Shape Wizard or a text editor to review the program for a nest that you made with ProNest CNC.

■ I made nests in the full desktop version of ProNest. How do I load them on my CNC?

You can load the nest programs on the CNC. However, you cannot add nest programs to the ProNest CNC Part List.

■ Can I bring in a 3D part file (such as SOLIDWORKS or Inventor parts) and nest that using ProNest CNC?

No. Only *.dxf, *.dwg, *.dgn, and *.cam file formats are supported.

- **Is True Hole supported? How can I identify if a hole has True Hole once parts have been nested?**

Yes. ProNest CNC applies True Hole to qualifying holes if you select a material class with "True Hole" before you select **Nest**.

If True Hole exists on the current part/nest in Phoenix, a "True Hole technology in use" status message shows on the Main screen.

- **Can nests be saved for later use?**

Refer to [Step 9: Cut or save a nest](#) on page 14.

- **Can I bevel parts with ProNest CNC?**

Beveling (applying non-perpendicular cut angles to part edges) is not supported with ProNest CNC. The term "Bevel" in the ProNest CNC material / class list refers to Hypertherm Bevel consumables. For XPR systems, standard consumables (which are all bevel-capable) are used for all classes, including True Bevel.

- **Can I add multiple Shape Library parts to the part list and nest them together?**

Only one Shape Library part can be added to the part list at a time, even if you have the True Shape nesting module.

- **Does ProNest CNC support marking?**

No. All geometry found in CAD files is cut. Even if there is a designated layer for marking, the geometry on that layer will still be cut.

- **Is Collision Avoidance available with ProNest CNC?**

At this time, ProNest's Collision Avoidance feature is not applied to any nests made with ProNest CNC. There are other aspects of Rapid Part™ technology that may be applied when using an EDGE Connect with a Sensor THC, including:

- Automatic crossover height calibration
- Preflow during initial height sense (IHS)
- Skip IHS

- **Can I use a setup file (*.pn or *.pnca) from the full desktop version of ProNest on ProNest CNC?**

No. ProNest CNC has its own unique configuration archives.

- **Is waterjet or laser supported on ProNest CNC?**

Only plasma, oxyfuel, and waterjet generic cutting are supported.

Trace a remnant in Phoenix

In ProNest CNC, you can nest parts on a rectangular part or on the last remnant that you traced in Phoenix. Refer to [page 12](#).

In Phoenix, you can trace remnants with straight lines only.

To trace a remnant in Phoenix:

1. From the Main screen in Phoenix, select **Shape Manager > Teach Trace**.

2. On the Remnant Window, jog to the location where you want the trace to start. Use the joystick or jog keys to move the torch over the plate.

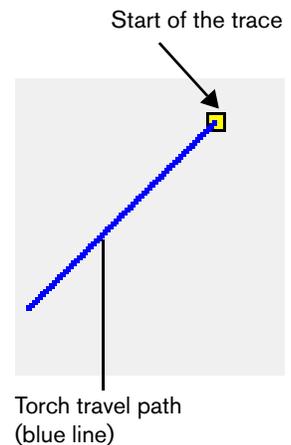


A blue line in the Remnant Window shows where the torch has traveled.

3. Select **Add Point**. A yellow rectangle shows the location where the trace starts. Refer to the example on the right.



If you added a point in the wrong location, select **Delete Last Point**.



4. Jog the torch to the next corner point on the remnant and then select **Add Point**. Phoenix traces a straight line between points that you add one after the other.

5. Do step 4 again until you have traced the remnant. The **white area shows the shape of the remnant** that has been traced.

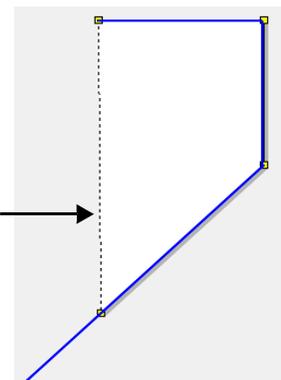


You must add at least three points.



It is not necessary to jog the torch back to the start to close the remnant. **The dotted line shows where Phoenix automatically adds a line to close the shape** if you do not add another point.

Dotted line →



6. When the trace is complete, you can nest a Shape Library part or CAD file parts on the remnant.



Make sure that the **Use Remnant** checkbox in ProNest CNC is selected.

- To nest a CAD part file on the remnant, select **ProNest CNC** and then refer to [page 5](#).
- To nest a Shape Library part on the remnant, select **OK** to return to the Main screen and then refer to [page 5](#).