



Version 6.1 Release Summary

September 2003

Overview

Version 6.1 introduces a major new feature called Feature-Based Machining. In addition to Feature-Based Machining, this release contains new machining strategies and new features along with enhancements to existing features. As with every release, many corrective maintenance issues have been addressed as well. This release summary will briefly review some of the new features and changes to Prospector. Choose What's New from the Help menu for a more thorough explanation of the changes.

Important News about Your Software Licenses

When you install your software, your hardware key (dongle) will automatically be updated to allow you to use the Version 6.1 release. The automatic update of the hardware key occurs during the final step of installation. This means that it is necessary that your hardware key be connected to the parallel port of the workstation you are updating to ensure the key is properly re-initialized. If you forget to attach the key, you can reinstall the software or run LiveUpdate and choose the license update file:

<Install Path>\DongleUpdate61.txt



If your software is not covered by a software support contract, your hardware key will not be properly updated and you will not be able to run the new release.

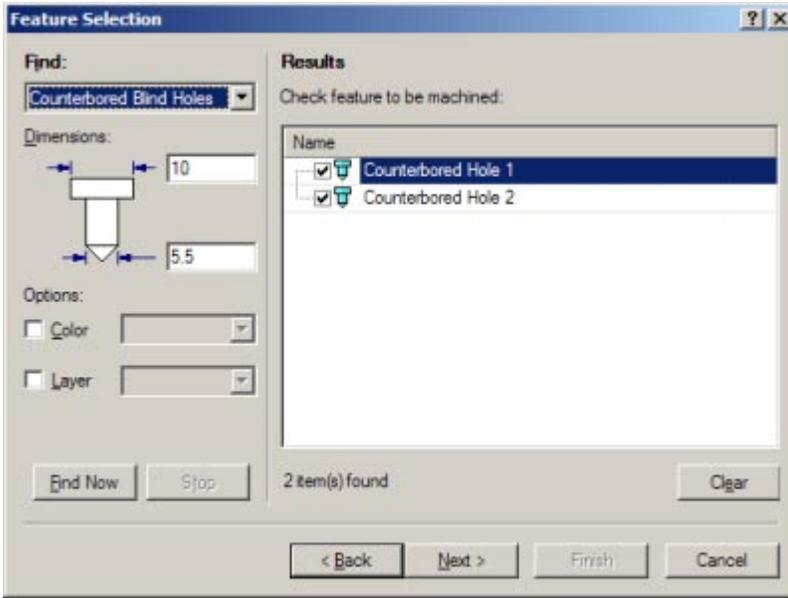
If you are not sure of the status of your software support or would like to check with us before loading the update, call us at **888-294-9450**.

An Important Note about Installing Version 6.1

Make a backup copy of your PowerSource databases before you install and run Prospector. This release will automatically update your PowerSource databases and seed it with default rules and settings for the new features. This update process was part of all previous releases and occurs the first time you run the software. Version 6.1 is different than previous releases in that once your databases are updated, they can not be used with a previous release of Prospector. Should it be necessary to install and run an older version of Prospector, these updated database file will not work properly with the older release. Make sure you create a copy of your databases first and store these in a safe place just in case you ever need to use an older version of the software. If you have any questions about the procedures for updating your system, please call our HelpDesk at **(800) 280-0240**.

Feature-Based Machining

Feature-based machining is the ability of Prospector to identify 2D or 3D features of the job based on geometric shape and size which are in turn processed by PowerSource to execute a predefined set of rules to generate a sequence of programs to fully machine the feature. The first application of this new technology is being applied to hole making operations.



The new Feature Selection page of the new program wizard automatically finds specific hole types.

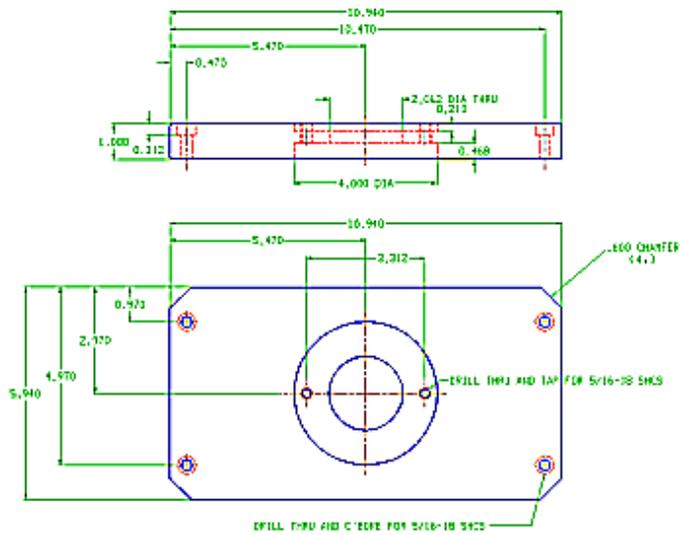
In this example, all Counterbored Blind Holes of a specific size are located.

Other options such as Color and Layer let you refine the search even further to choose only those features with specific characteristics.

Feature recognition works for both 2D detail drawings and solid models:



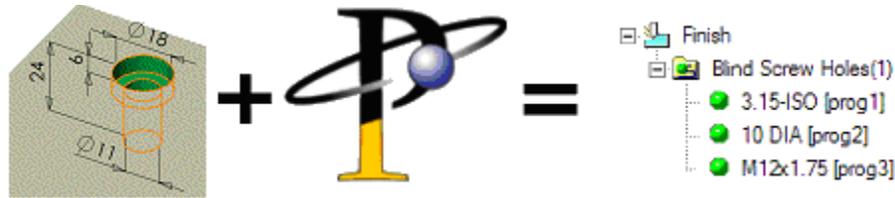
A 3D model provides the added benefit of automatic calculation of depths for machining operations.



2D detail drawings like this one from ExpertCAD recognize features based on standard drafting practices. The new block description feature for 2D projects means that through hole depths are automatically calculated.

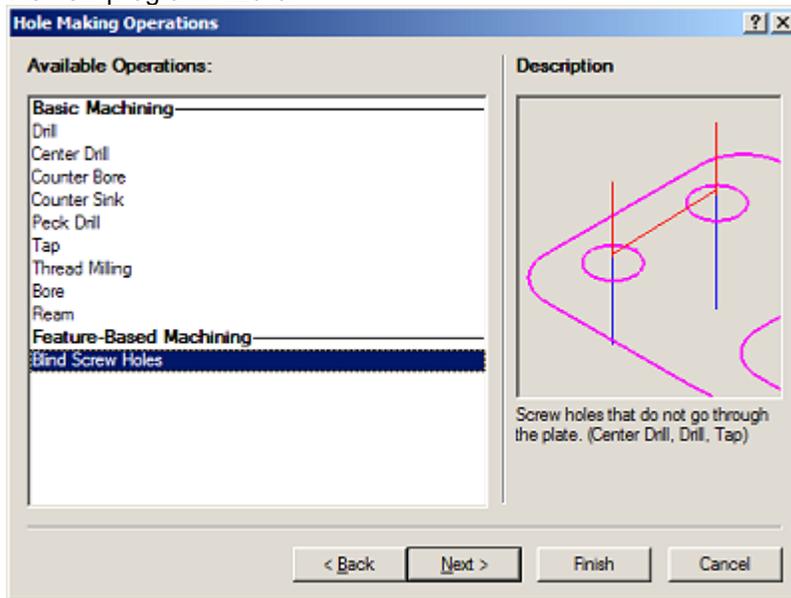
A wide array of hole types including countersinks and counterbores in either blind or through configurations can be detected. Feature recognition saves an enormous amount of time over previous methods to identify hole locations. Another significant benefit is that it helps eliminate the possibility of human error that could occur when manual methods are used (e.g. accidentally choosing a 1/4" hole location to include in the set of 1/2" holes to drill).

Once the feature to machine has been identified, a *Feature-Based Machining Sequence* can be applied to generate all the programs to machine that feature. A Feature-Based Machining Sequence is a discrete set of rules in your PowerSource database that are executed when you machine a feature in Prospector that will create all the programs and program parameters needed.



Conceptually the definition of the feature is fed as an input to PowerSource which runs a set of rules that results in the generation of a machining sequence. A *Machining Sequence* which is the series of separate programs needed to machine the entire feature is the end result.

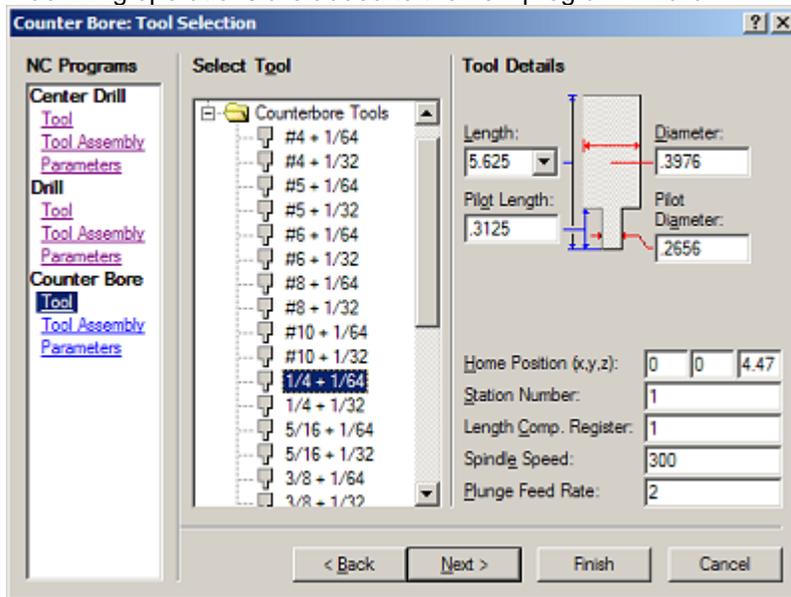
A Feature-Based Machining Sequence can be chosen on the Hole Making Operations page of the new program wizard:



Basic Machining is a category of discrete machining operations like Drill and Tap. Choosing a basic machining operation will create one program to perform that operation.

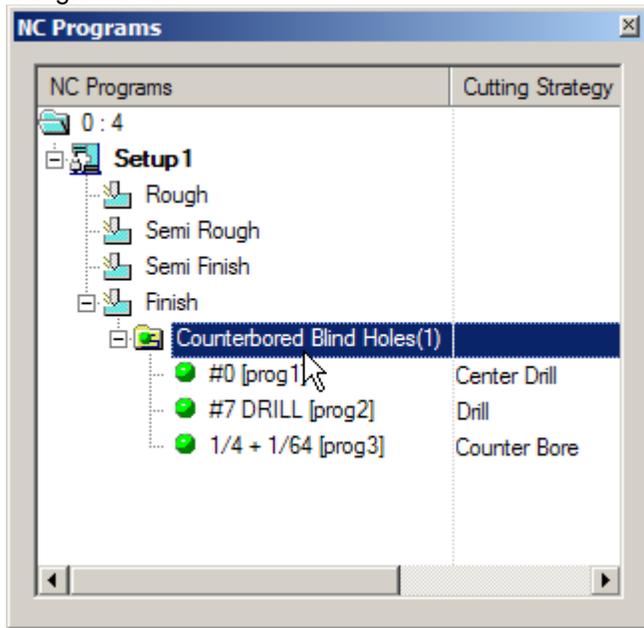
Feature-Based Machining is a series of operations defined in your PowerSource database to perform all machining operations for the feature. Multiple programs are generated when this type of operation is chosen.

When working with a Feature-Based Machining operation, all the required pages for the different machining operations are added to the new program wizard:



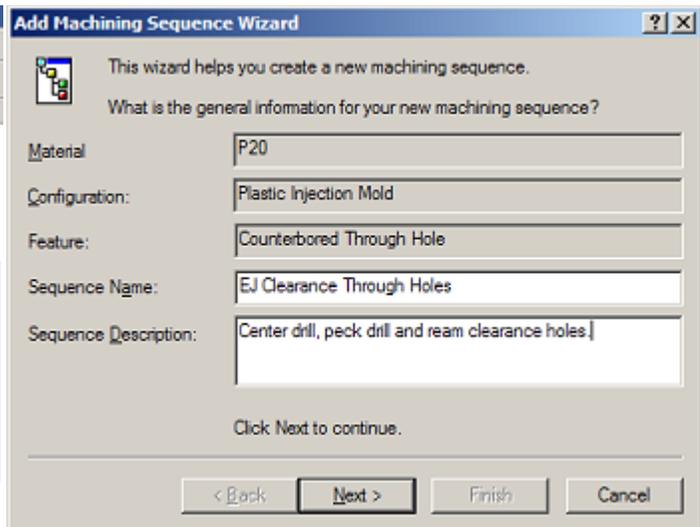
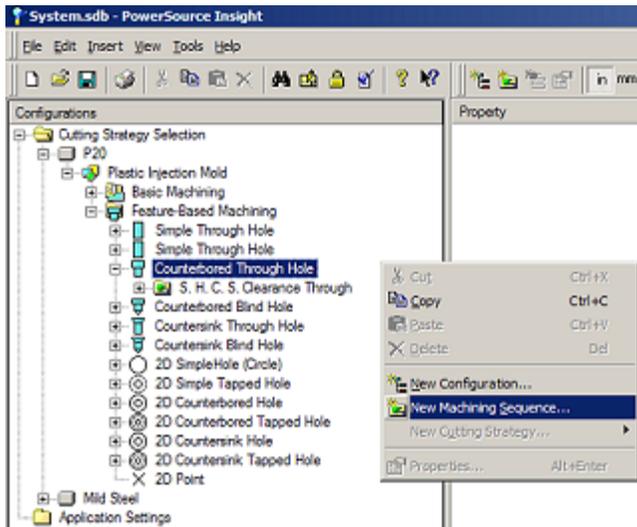
On the left side of each page of the wizard is a *road map* control that shows the pages of the wizard and which page you are on. In this example, 3 programs are being generated to machine this feature. You can choose to walk sequentially through the wizard or jump to a particular page by clicking on it in the road map control. Of course you can click Finish at any time because the rules for machining the feature have already determined the proper program parameters.

Using a Feature-Based Machining operation produces a machining sequence which is added to the NC Programs list:



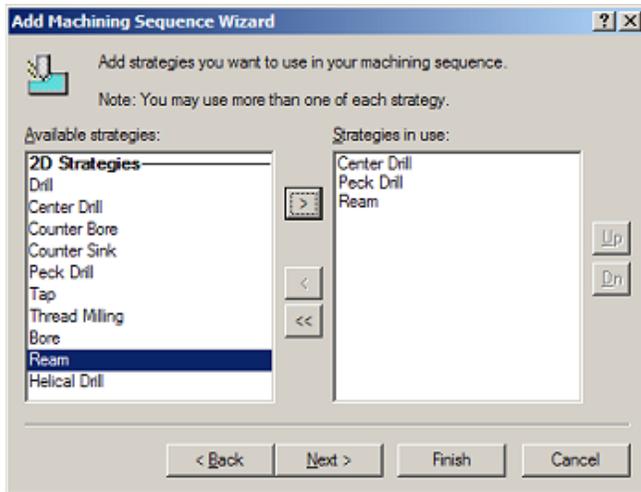
Just like an individual program, you can work with a machining sequence in many of the same ways. For example, you can use features like Update or Properties to change parameters or Send to Control to post process all the programs in the sequence.

Feature-Based Machining Sequences are defined using PowerSource Insight and stored as part of your database.

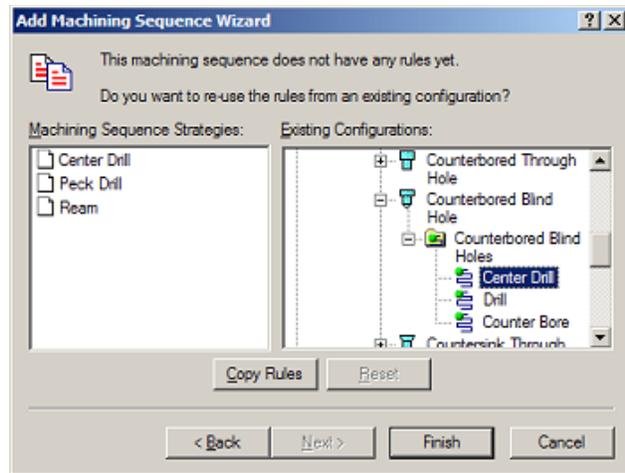


Any number of Machining Sequences can be easily defined for a given feature by choosing *New Machining Sequence*.

The easy to use wizard walks you through the process of creating a new machining sequence in PowerSource Insight. On this page, assign a name to your new sequence and provide a description for it. This description will appear in the new program wizard.



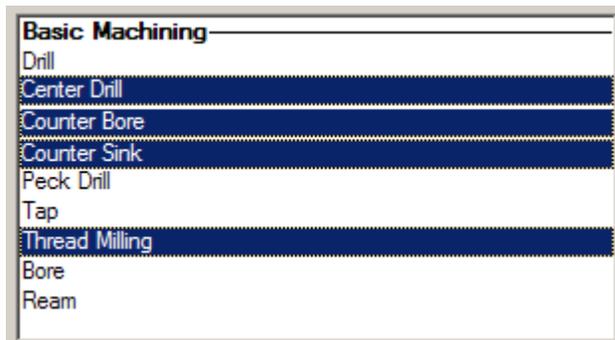
Indicate and order the sequence of operations for the new machining sequence.



Newly created sequences can inherit rules from other sequences. This makes it easy to clone existing sequences and with few modifications, fine tune the new sequence to address the specific application.

New 2D Machining Strategies

New hole making machining strategies have been added for Version 6.1:



Center Drill – formerly integrated with Drill. It is now a separate strategy to more clearly define the intended operation.

Counter Bore – this is a new strategy which includes the new counterbore tool type.

Counter Sink – a new strategy which includes new countersink tool types.

Thread Milling – a new machining strategy that offers significant advantages to tapping in certain circumstances.

In addition to these new machining strategies for hole making, other options specific to each strategy have been added to fine-tune the output. For example, dwell time can be specified for a center drilling operation and rigid tapping is now an option for tapping holes. To take full advantage of these options, your post processors may need modifications. If you haven't done so already, we recommend you take advantage of our **Pro/Post Service** to update your post processors for this release and all future releases. To learn more about Pro/Post Service, visit our web site at www.softtech.com or call our HelpDesk at **(800) 280-0640**.

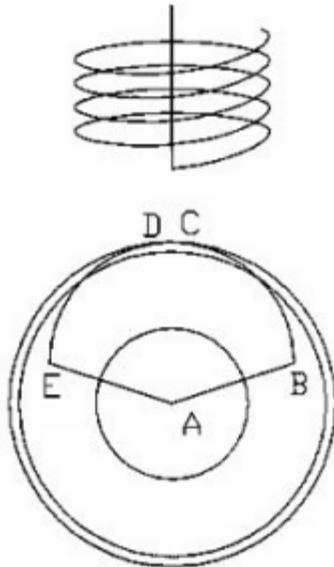
Thread Milling

Thread milling is an alternative method to tapping for machining threaded holes. It can be particularly useful in cases where larger diameter holes are involved or when working with difficult materials. Other advantages include:

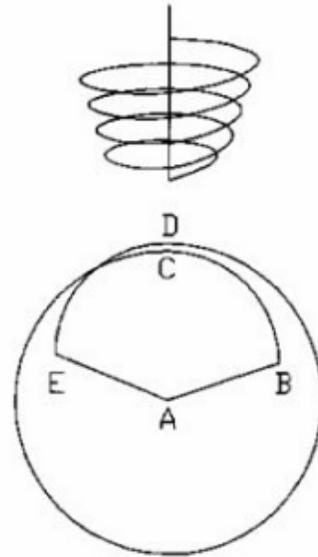
- High cutting Speeds and Feeds
- Produces threads with excellent form, finish, and dimensional accuracy
- Eliminates the possibilities and consequences of tap breakage
- Precise thread depth control
- One tool for through or blind holes

- One thread mill can produce varying thread diameters of the same pitch
- Smaller machines can produce larger threads due to less spindle torque
- No reversal of the spindle required

The thread milling machining operation produces a helical cutter path for straight threads and a spiral cutter path for tapered (NPT) threads.

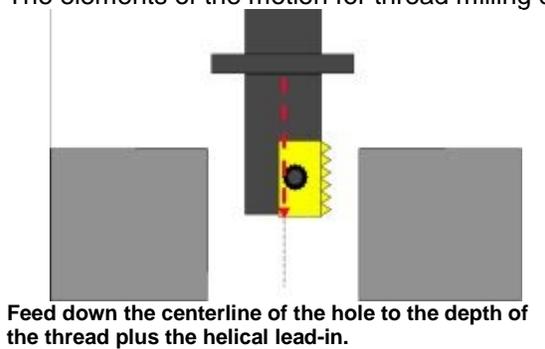


A helical cutter path as seen from above produces straight threads.

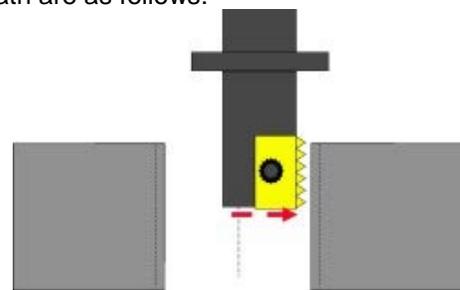


NPT threads are produced by a spiral cutter path.

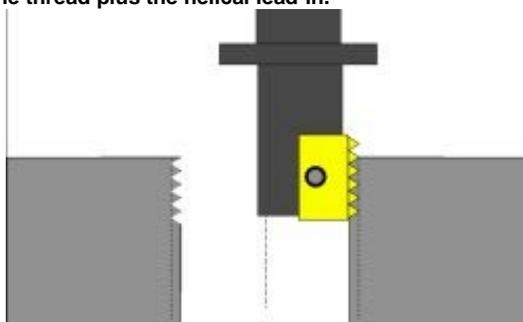
The elements of the motion for thread milling cutter path are as follows:



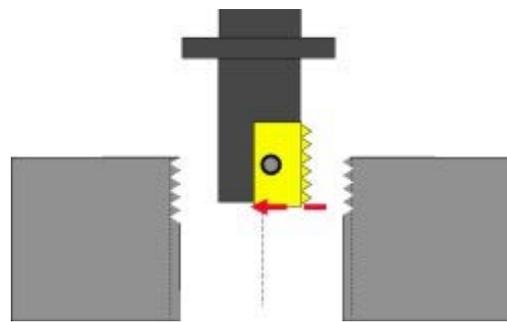
Feed down the centerline of the hole to the depth of the thread plus the helical lead-in.



Helical lead-in to the start of the thread.

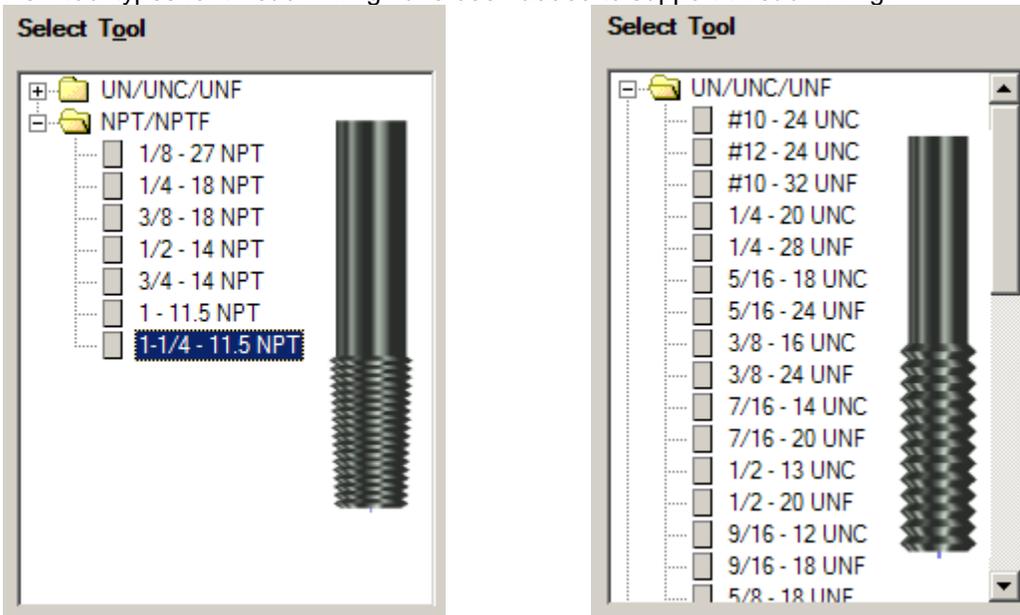


Helical cut in +Z to form the threads.



Helical lead-out to the centerline of the hole.

New tool types for thread milling have been added to support thread milling:



Thread milling tools for NPT/NPTF standard and UN/UNC/UNF standard threads are included in the PowerSource database.

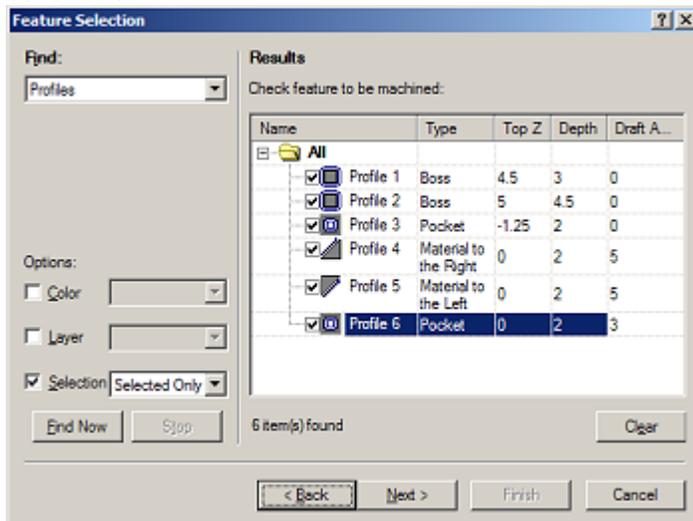


Tooling, feed rates and spindle speed data as well as the default rules for thread milling parameters provided by SECO/Carboloy.

We recommend SECO/Carboloy tooling for use with all Prospector machining strategies.

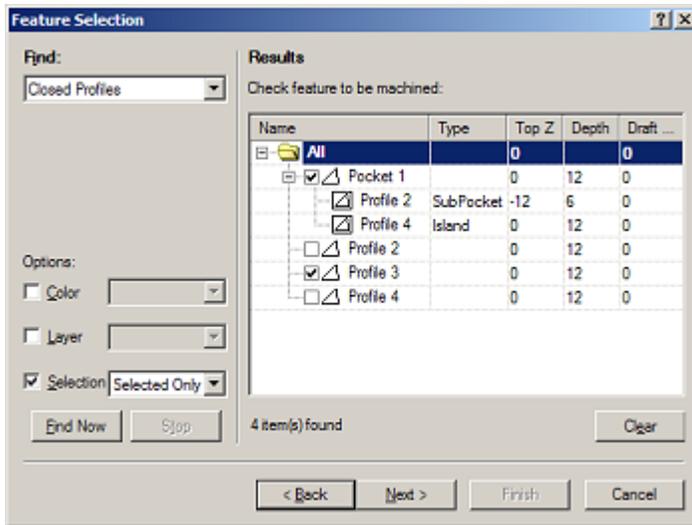
Enhanced Pocketing and Profiling

The revised new program wizard allows for much better definition of the features to be machined. This added definition greatly assists Prospector in choosing the best program parameters for the conditions.



Once you choose Profiling on the first page of the new program wizard, the new *Feature Selection* page lets you indicate which features to profile and assign proper height, depth and draft to each feature independently or as a group.

Options to assist in identifying profiles to machine include by selection, by color and by layer.

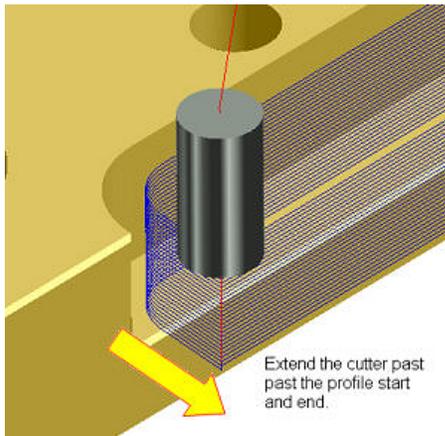


The Feature Selection page for pocketing offers the same features as that for profiling.

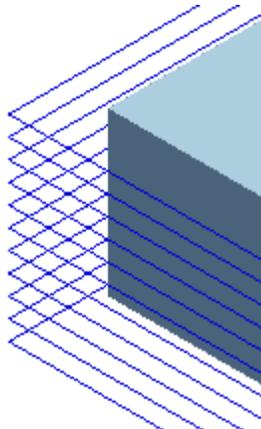
Expanded pocket definitions let you identify *sub-pockets* and *islands* as distinct features within a pocket.

Z level, depth and draft for every feature can be set independently or as a group.

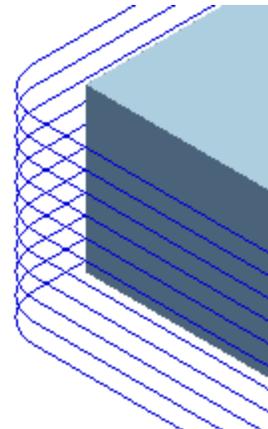
Several new options for 2D profiling and pocketing have been added:



The new *Extend Open Distance* option for profiling allows a cutter path to be extended at the start and past the end of an open profile. This helps ensure you make a clean cut for these conditions.

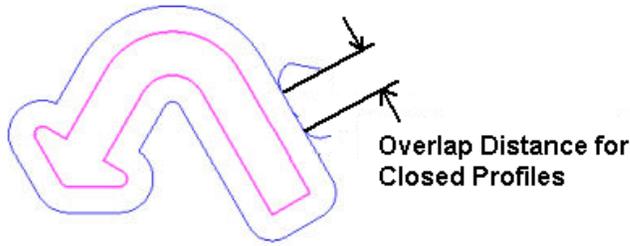


Sharp Corners



Sharp Corners

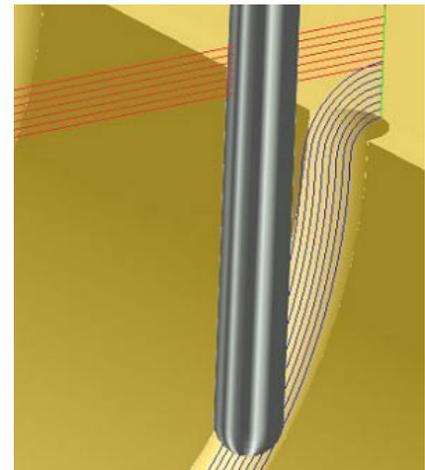
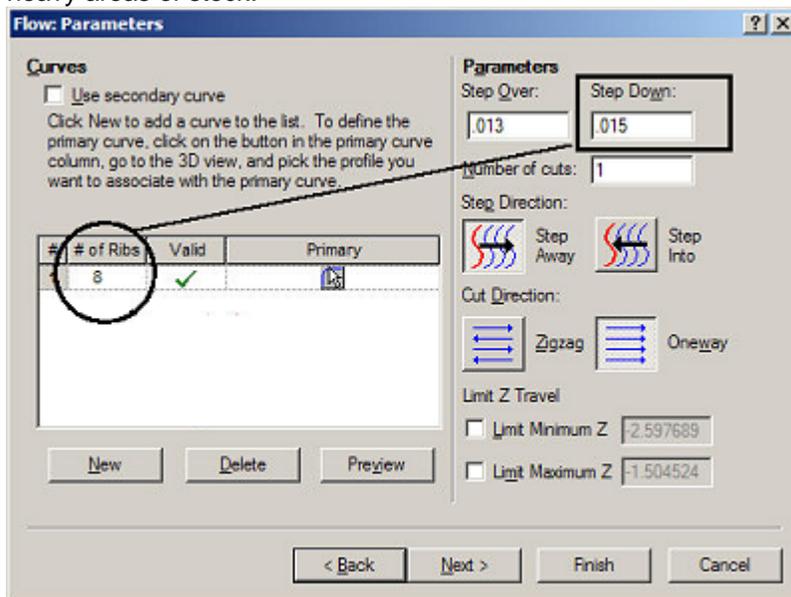
The new *Sharp Corners* option lets you choose whether or not to roll the cutter around corners.



Overlap Closed Distance extends the cutter path of a closed profile to overlap the start and end point by the distance specified. This prevents the possibility of leaving a small scallop on the part. This property is available for both 2D profiling and 3D Z-Planar No Clear programs.

Stepping Down Flow Cuts

A new feature for Flow Machining allows for multiple passes to walk the tool down in Z to remove heavy areas of stock.



The # of Ribs specifies the number of passes for the flow cut. Each pass will step down in Z by the Step Down distance you specify.

Machining Flats with Z-Planar With Clear

New options are available for machining flat areas of the job when using the Z-Planar With Clear machining strategy. The *Machine Floors* property has 3 settings:

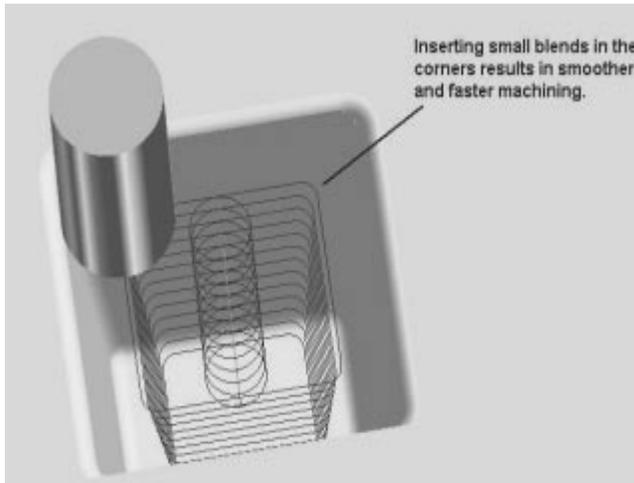
Property	Value
Zig Zag Cut Angle	0
Quick Start	No
Quick Start Levels	6
Contour Transition Angle	89
Quick Rough	No
Optimize For Performance	Yes
Machine Floors	Off
Lift Between Islands	No
Fast Feed Rate	35
Home Position	
Tool	
Tool Setup	
High Speed Machining	

Off – a fixed step down is used to determine the location of each Z-level.

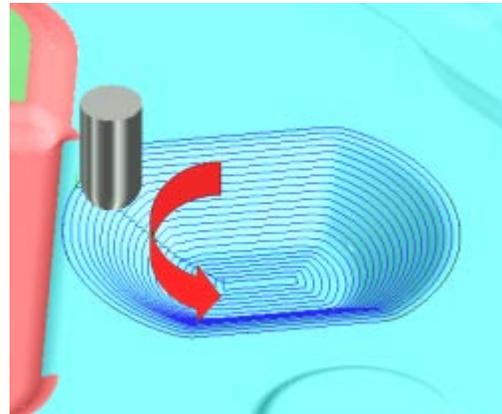
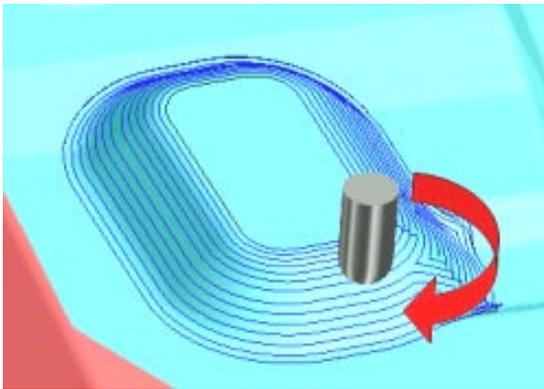
Insert Levels At Floors – this setting will insert additional levels at each flat in the event the programmed step down does not fall exactly at the same level of the flat area. Only the flat areas are machined; not the entire inserted level.

Floors Only – levels are generated to machine only the flat areas of the job.

3D Machining Enhancements



A new *High Speed Machining* option has been added for Z-Planar No Clear programs to replace sharp interior corners of the cutter path with blended corners. This can be a very good technique to use for roughing, semi-roughing or even semi-finish because at these stages of the machining process there can be high concentrations of stock left in corners of the job. By inserting blends, you can smooth the transition into the heavier stock and eliminate the abrupt change in direction. Machine and tool wear as well as machine time can benefit using this technique.



New options for Contour Machining lets you choose the direction of the cut. For generally convex areas, it's usually a good choice to use a clockwise direction to maintain climb cutting. For concave areas, a counter-clockwise direction is the better choice.

Radial machining has been enhanced to distribute cuts more evenly between the primary and secondary curves. This helps produce smoother finishes when machining around tight corners and in areas of high curvature transition

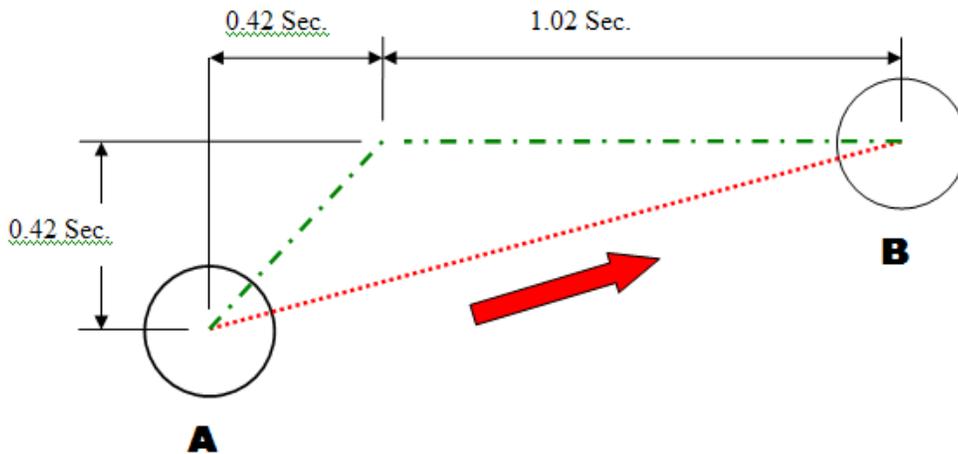
A new *By Level* option for scribe machining lets you choose to cut every curve at a given Z-level before proceeding to the next level.

Machine Tool Simulation for Rapid Traverse Motion

When a machine tool is instructed to move at rapid traverse (typically 'G00' command), the actual motion the machine takes is dependent on how the CNC control instructs the machine to move. There are 2 possible scenarios:

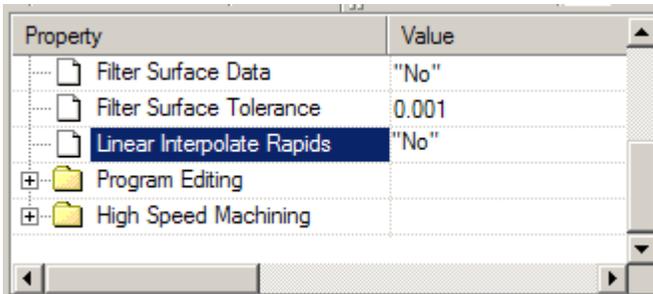
1. Linear interpolation - the tool is positioned to the commanded coordinate so that the X,Y,and Z axes arrive at the same time. This means that the path taken between any 2 points is always a straight line.
2. Non-linear interpolation - all axes of motion move at their respective maximum rate until they achieve their individual commanded coordinate location.

In the second case of non-linear interpolation, unless the programmed path is moving just one axis or is evenly divisible by 45 degrees, the actual path the machine will take is not a straight line.



The commanded path from point A to point B is the red dashed line. The actual path the machine takes will be the same as the red line if your CNC uses linear interpolation for rapid moves. If it does not, the actual path is the green dashed line because both the X and Y axes are commanded to move at their maximum feed rate until they reach their respective end points. In this illustration, the Y axis reaches its end point in 0.42 seconds. At 0.42 seconds when the Y axis has completed its travel, the X axis is still 1.02 seconds away from its end point. Hence the actual motion on the machine is a “hockey stick” shape indicated by the green line.

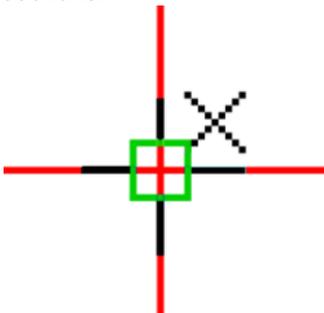
You can choose how you want Prospector to interpret rapid traverse motion by adjusting the Linear Interpolate Rapids setting under Application Settings in PowerSource Insight:



 It's important to set this property based on the behavior of the machine because it affects how Prospector detects gouges, collisions with stock, display of rapids and computation of proper clear planes when traversing from one area of the job to another.

Snapping Point to Intersections

When digitizing points or using the inferencing feature, Prospector will detect and report intersections.

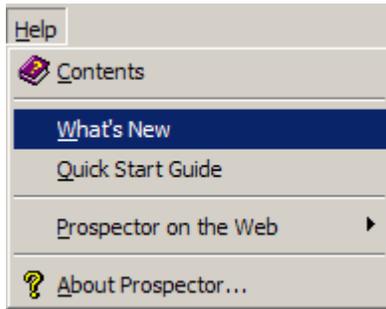


The 'X' beside the cursor indicates that an intersection has been detected.

In the case where 3D lines do not actually intersect, a projection of the theoretical intersection in the 2D view is computed. The location of the point is then projected onto the 3D line closest to your eye.

Customer Closed Track IDs

Track is the electronic database system that records all customer and internally generated requests for corrections and enhancements. When you report a problem or request an enhancement, our customer service representatives will input your request into the Track system and give you a Track ID number. When we complete a release, we set the status on all records in the Track database that have been addressed to closed. For a complete listing of all Track items addressed by this release, please refer to the What's New help in Prospector:



Customer Service - Version 6.1 Update Service Program

As you can see, Version 6.1 introduces a lot of new features particularly for 2D machining. Learning how to take full advantage of all these new software releases can take a serious amount of time. This is especially true if your users are trying to figure it out on their own. Not only is learning by trial and error inefficient and sometimes frustrating, it can also lead to expensive mistakes. Why not make it easy and let our experts train your team to get the most out of this new release? Our Version 6.1 Update Program is designed to get your users up to speed fast to achieve higher levels of productivity using the new software.

The Prospector Version 6.1 Update Service Includes:

- Software installation and database updates for up to 6 workstations.
- Configuration of new PowerSource settings to suit your shop standards and requirements.
- Complete hands-on training on all the new features for up to 4 students.

Your users are trained and every system is up and running the new release to your satisfaction in just one day. This means minimal interruption in your operation. The cost is **\$895** (plus travel and expenses for sites outside 100 mile radius of our Troy, MI office). Don't get left behind! For more information or to schedule a service call, please call our HelpDesk at **(800) 280-0240**.

Pro/Post Service

This is a unique program designed to take care of your post processor requirements perfectly and best of all, permanently. Every post processor provided to you under this program is developed and tailored to your exact shop standards and requirements. The output is always 100% correct and ready to run on the machine. Every post processor is covered with an unlimited warranty provided your Prospector application software is covered by a software support agreement. This means any changes you require for whatever reason are free. Starting at just **\$495** for a typical 3-axis mill, Pro/Post Service is very affordable. Speedy delivery and fast turn-around for any modifications you may require is all part of the service. To learn more or get started today, call our HelpDesk at **(800) 280-0240**.