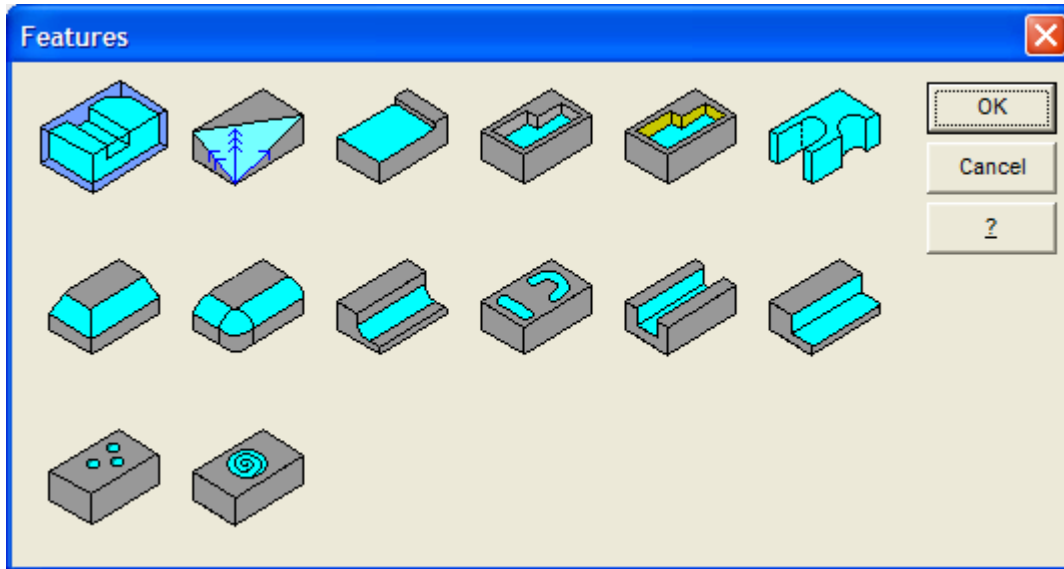


VECTOR CAD/CAM

NC FEATURES, a new product, which can save up to 90% of NC Programming time



An Operator Assisted Manufacturing process Invented and Developed by Centriforce

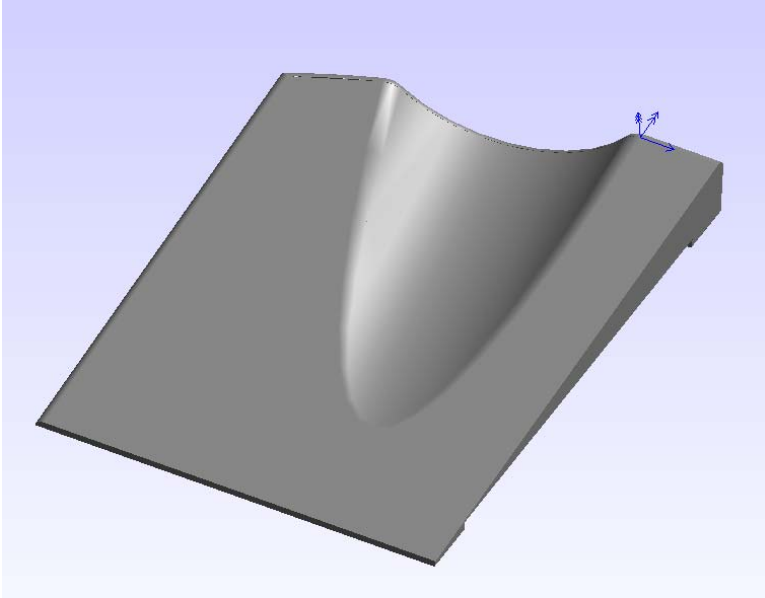
NC-FEATURES is a Trademark of CENTRIFORCE of The Netherlands

Last Updated March 2009 by Fred Smith - IMSservice

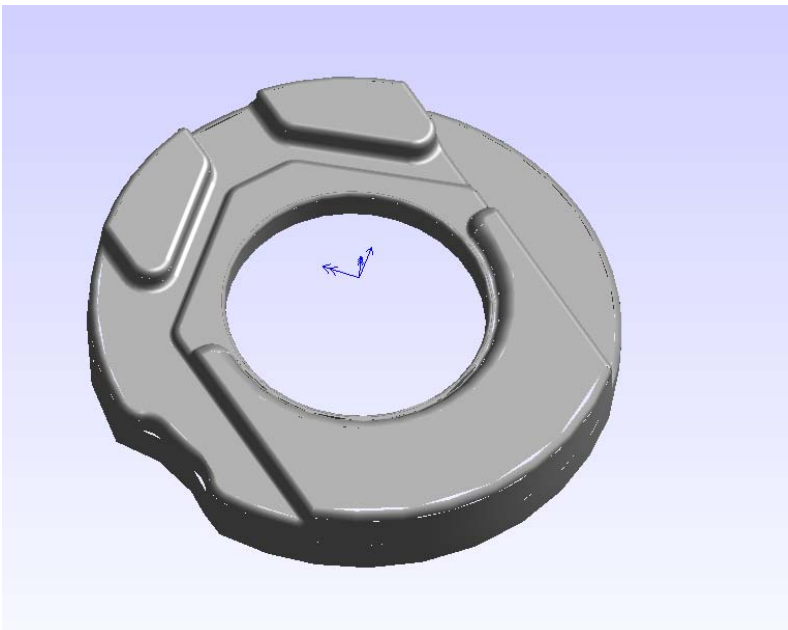
Introduction

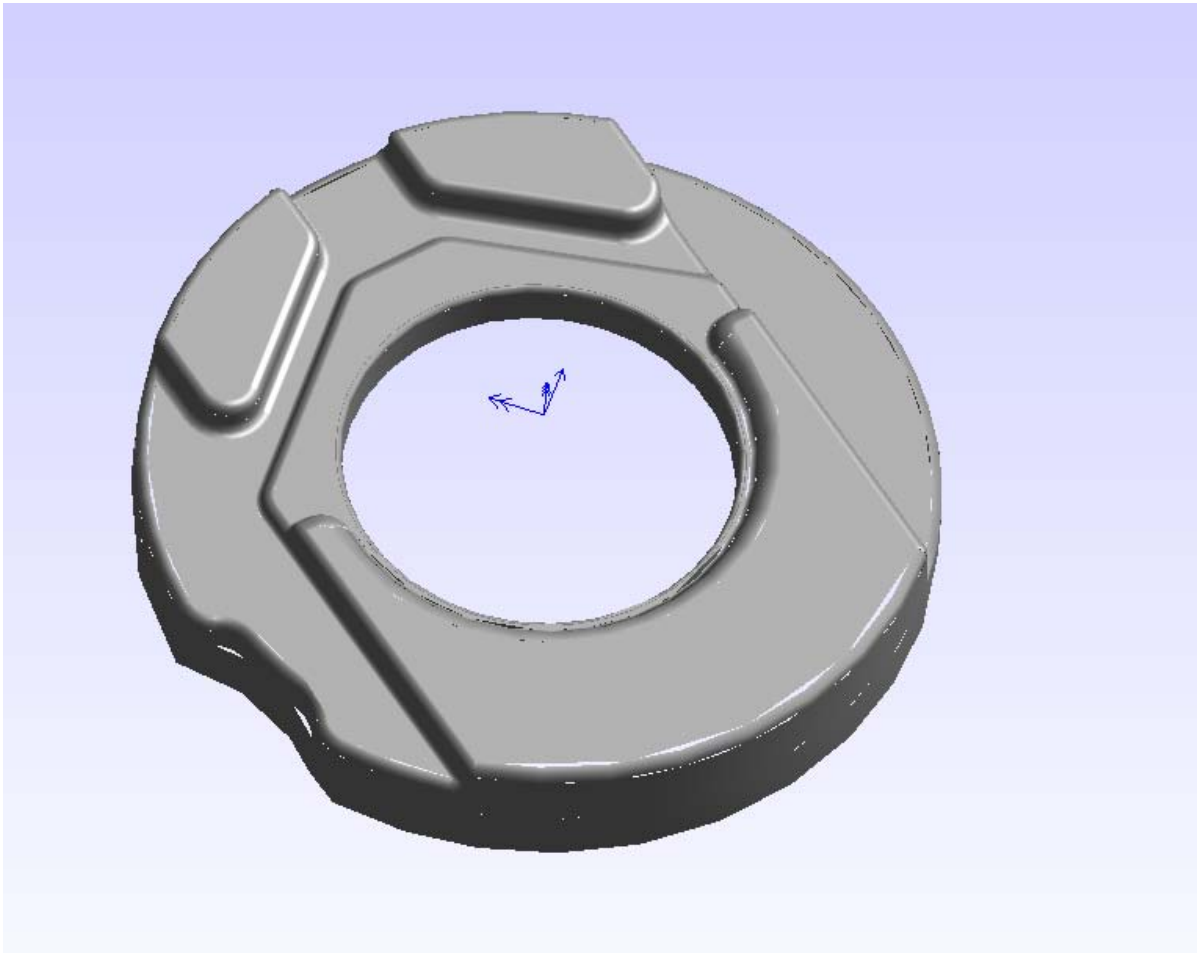
During the past 15 years the development of products has shifted from 2D to 3D. Tens of thousands of small manufacturing companies now receive 3D designs from their customers. However 90% of these products can still be manufactured using 2-Dimensional machining solutions.

This is an example of a 3D Design containing true 3D surfaces. This part requires 3D machining.



This part is an example of a 3D Design containing mostly or only 2D machining processes





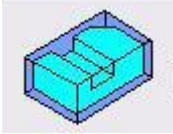
There are almost no software systems in the market, which offer simple machining solutions for the above example. Yet an experienced Operator can see that 100% of the part shown above can be machined using 2-Dimensional processes known as holes, slots, fillets and contours.

NC-Features from Vector CAD/CAM offers a series of processes, which result in simple short CNC programs, which the Operator can select and generate within seconds. The learning curve is less than one hour due to the use of familiar names and strategies. The simple interface produces huge and instant productivity gains for your shop.

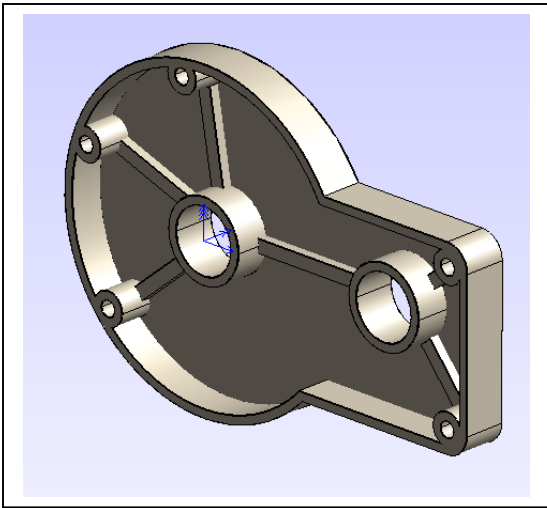
Some Examples of NC Features

FEATURE: DETERMINE STOCK AND TOPSIDE

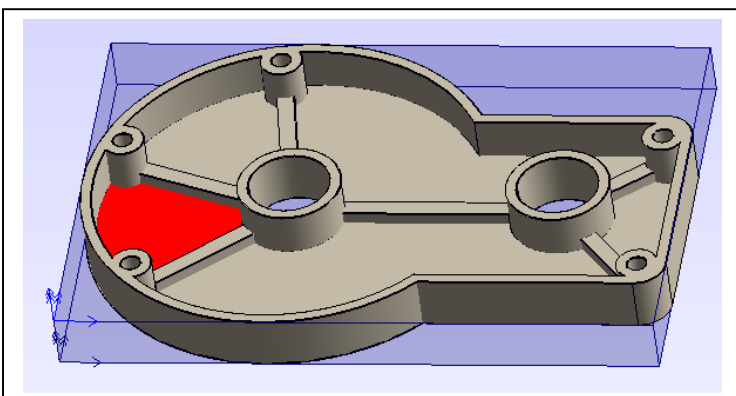
The first thing an operator wants to know: How much raw material is needed and which is the Up-Side



This part was sent from a customer sideways.

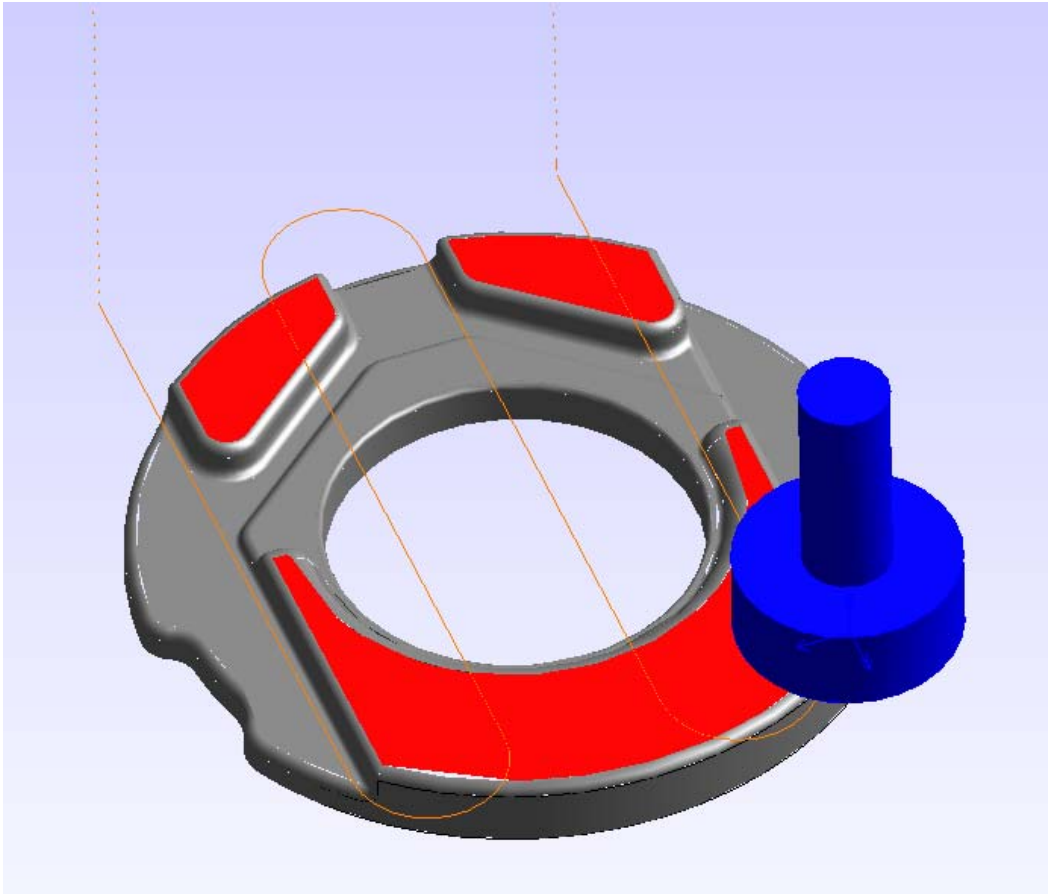
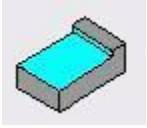


The Orientation of the part is not suited for machining. The true amount of raw material needed is unknown. The Stock Feature solves this in 3 seconds. It generates stock and rotates the part.



FEATURE PLANING

The Stock itself, or some selected surfaces near the top of the part can be “flattened” forming the part reference for other machining operations.

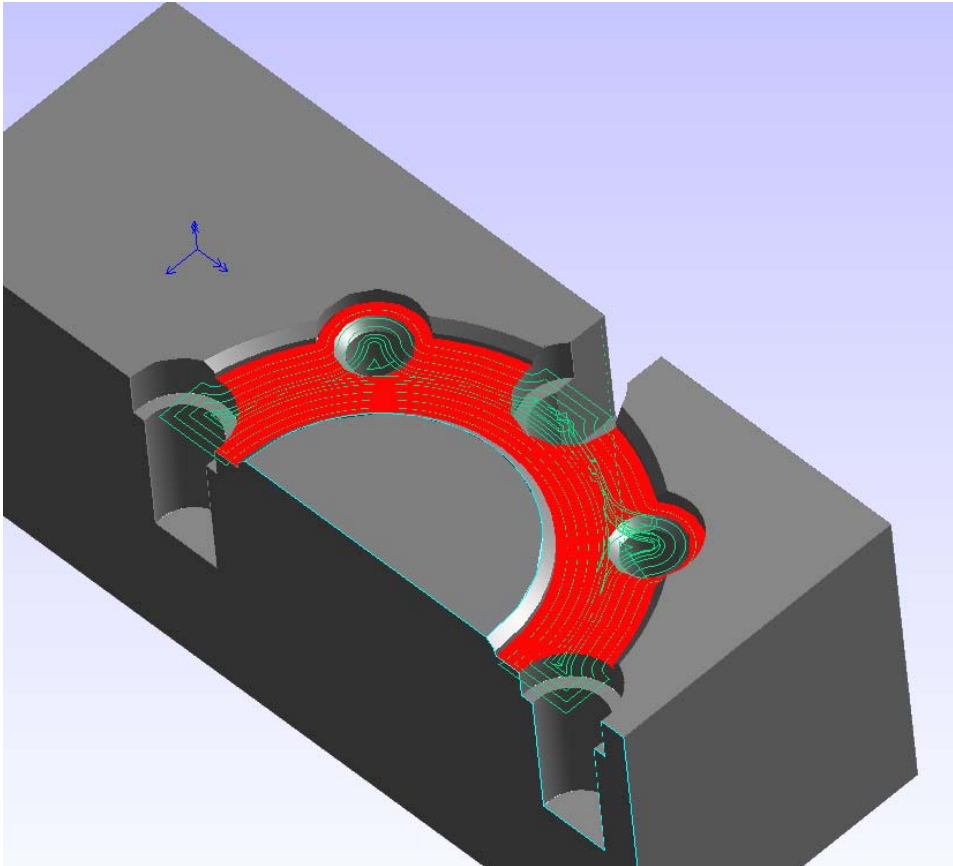


Parts sometimes have obstacles. The Planing Features automatically detects these protruding details and calculates the collision avoidance path automatically.

The Planing Feature calculates multiple surfaces in 3 to 4 seconds

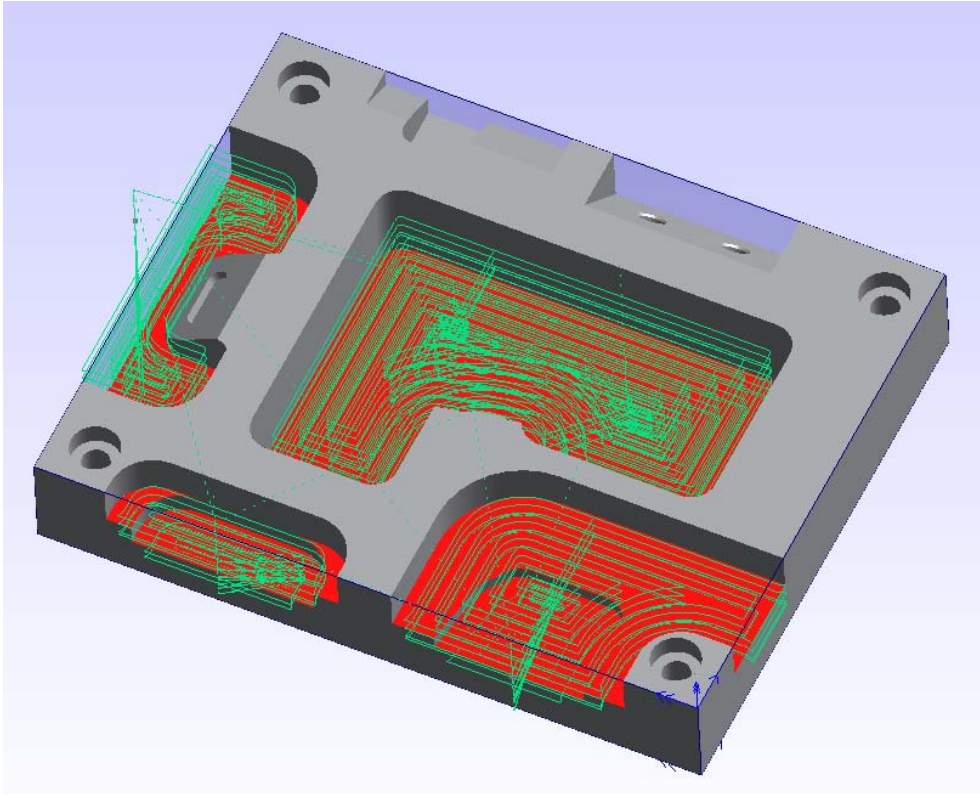
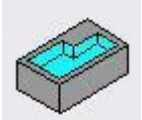
FEATURE: POCKET MILLING

Pockets can be closed cavities with a flat bottom. But a pocket may also extend to the outside of the part. The milling cutter will detect and overlap these openings automatically.



The Pocket Feature facilitates very complex shapes, avoids going down in cavities which should be drilled or subjected to other machining processes.

Use **Features-Extra-Activate Planar surfaces** to quickly select the surfaces needing to be pocketed.



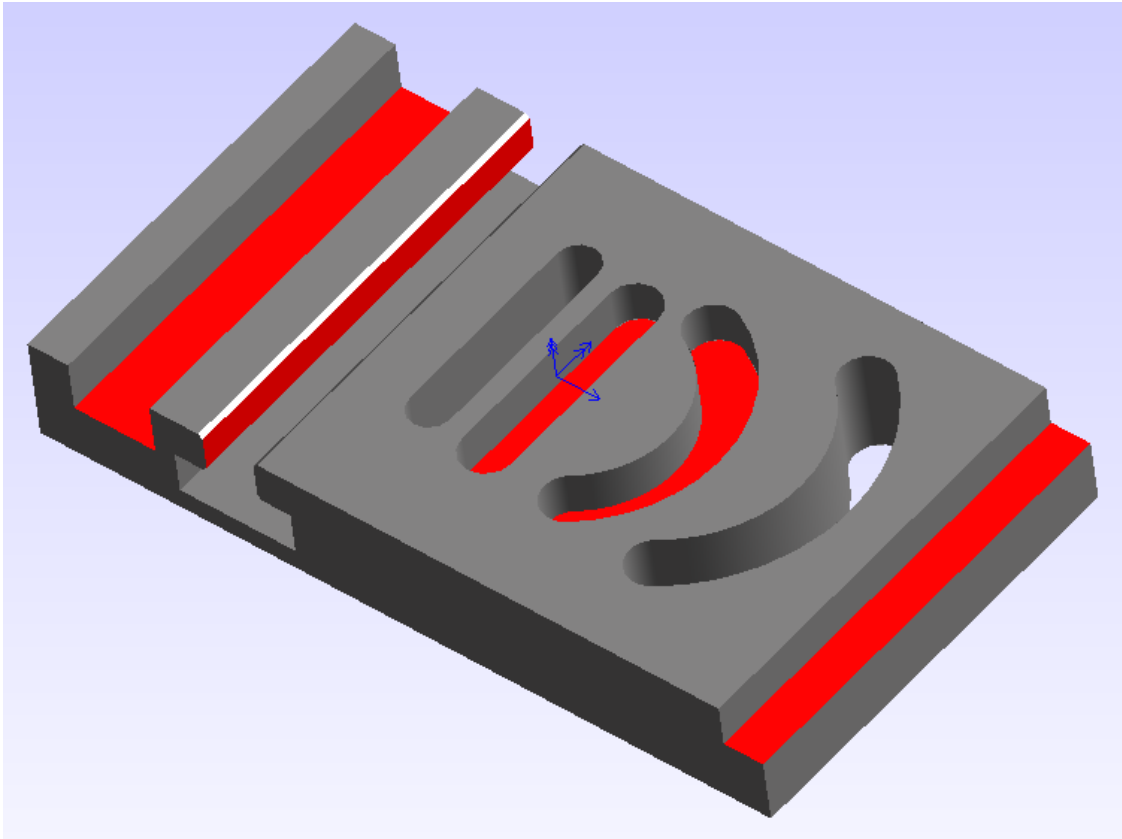
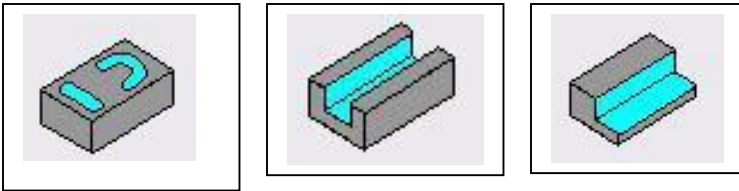
Multiple pockets may be selected at once. The Pocket Feature calculates all surfaces with automatic collision avoidance; it overlaps open sides and connects using safe Z-step-over.

The Pocket Feature calculates a pocket in 2-3 seconds. When multiple surfaces are selected and Collision avoidance is active the calculation time may be extended by a few seconds.

After the pocket is roughed out, the side walls can be finished with the **Pocket finishing function**.

FEATURES: Slot, Groove and Step

The Operator needs to know how wide and how deep these shapes are. The NC Programs must be short using maximum size tools and special machining strategies.

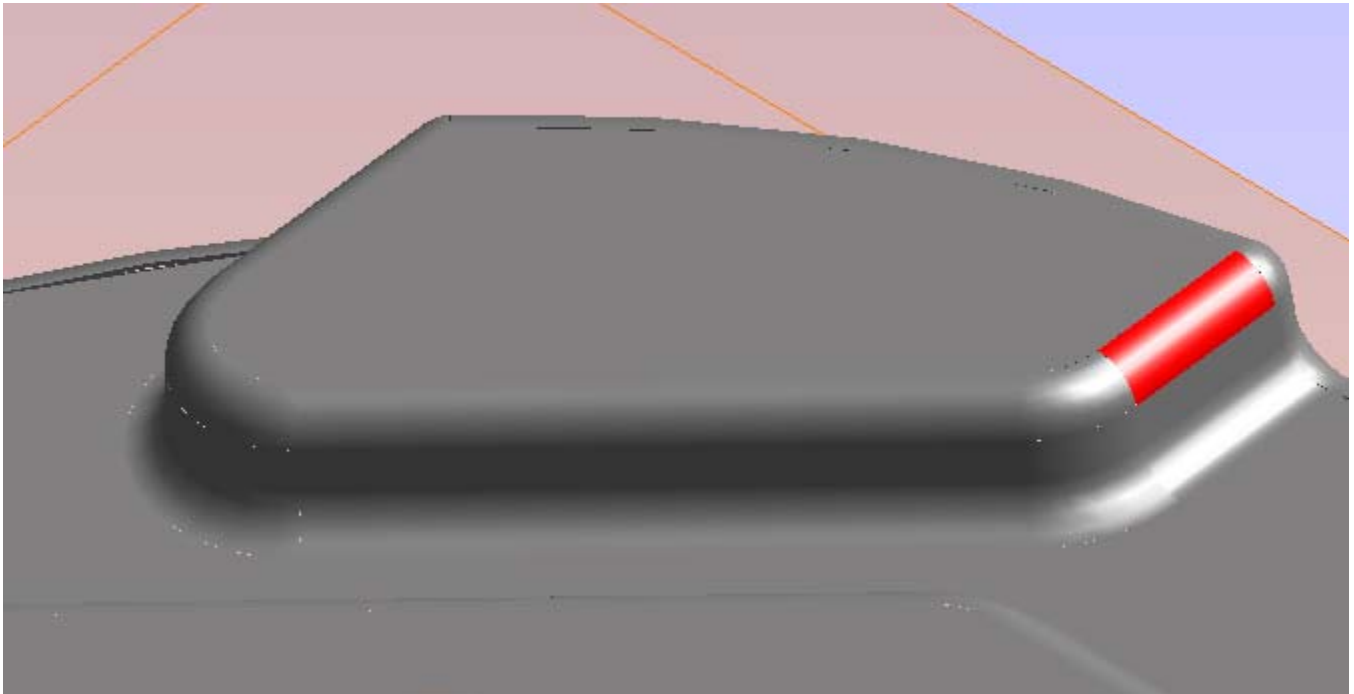


NC Feature Slot and Hole first informs the Operator of the Width and Depth of the part. Often the maximum size tool is desired in order to save time and to create the shortest machine programs.

The Slot ,Groove and Step Features calculate the NC Program in 1-2 seconds offering a variety of strategies.

FEATURE FIND FILLETS and CHAMFERS (This function is not released yet)

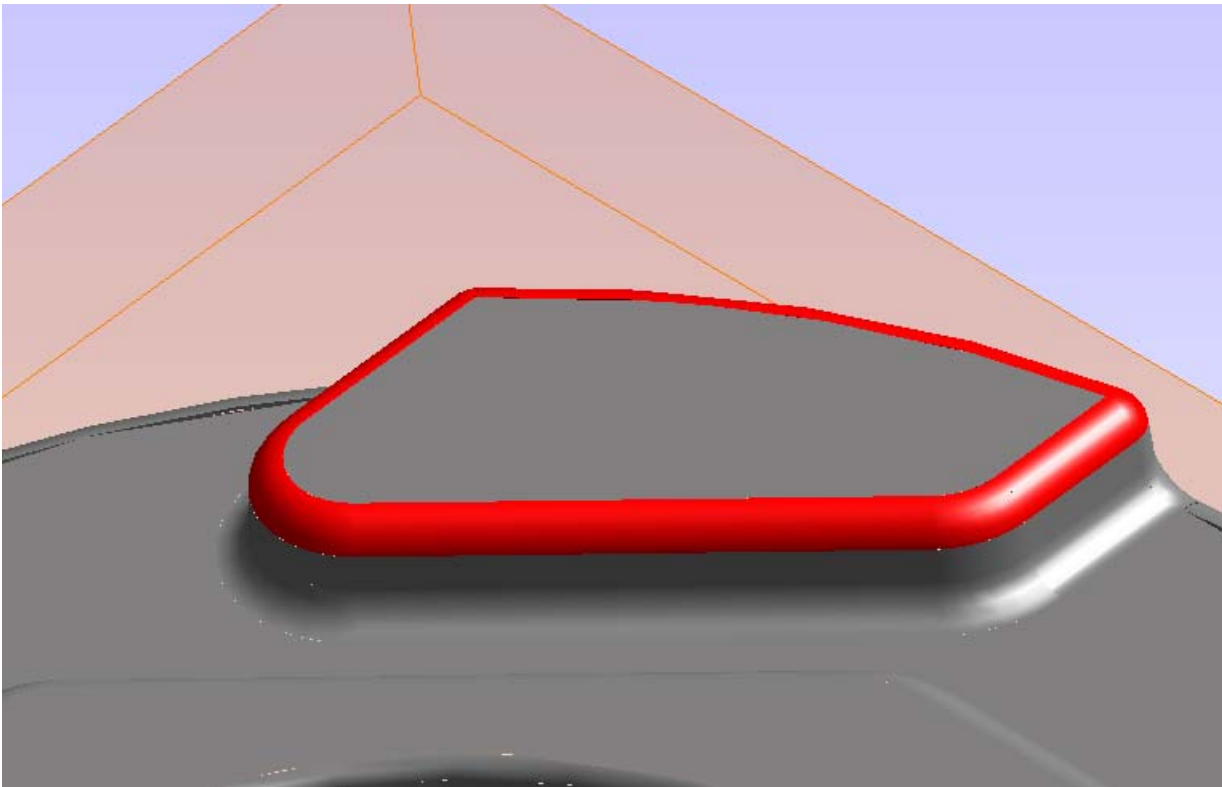
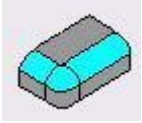
Find the neighbors of a selected area in the same plane in order to use a Rounding or Chamfer milling cutter. This greatly reduces the machining time and program length.



The Find Feature searches for adjacent surfaces in 2-3 seconds.

FEATURE FILLET

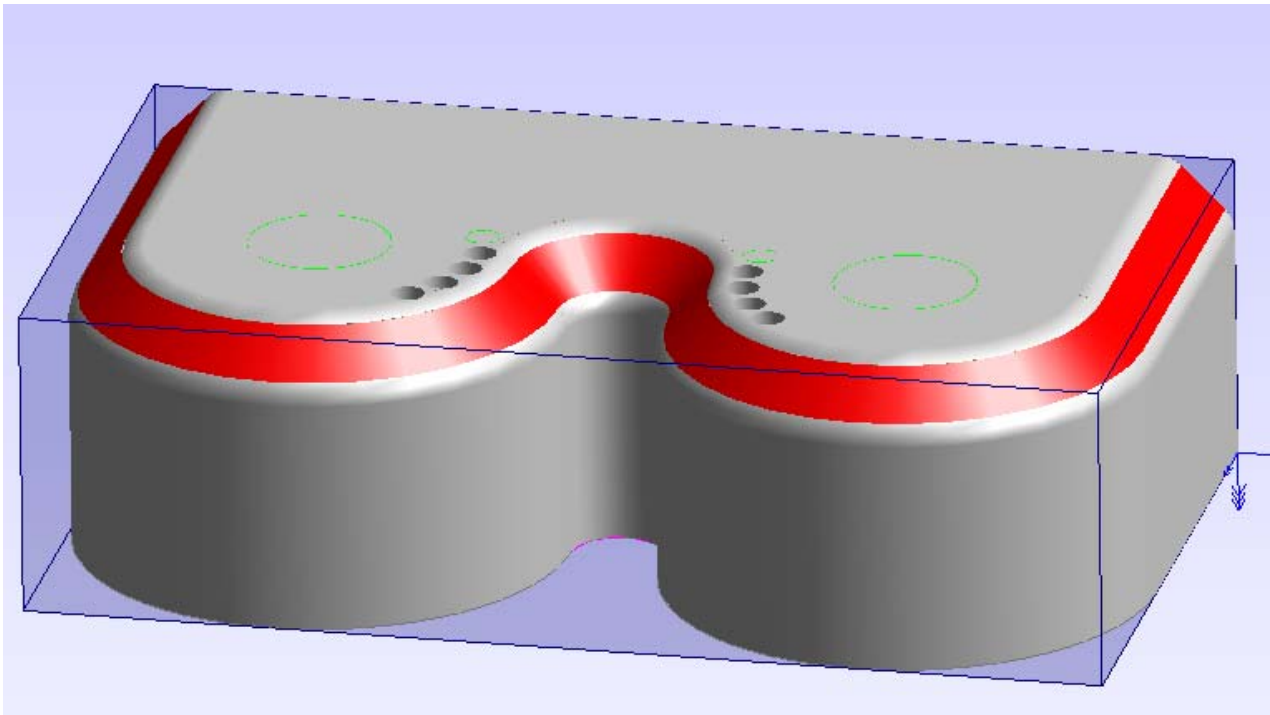
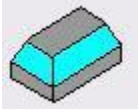
Informs the Operator of the Radius in order that the correct tool can be used producing a very short NC Program.



Automatic overlap is generated. The Fillet Feature generates a program in 3-4 seconds.

FEATURE CHAMFER

Informs the Operator of the Angle and Height in order that the correct Chamfering tool can be used producing a very short NC Program. Multi passes are permitted for larger surfaces.

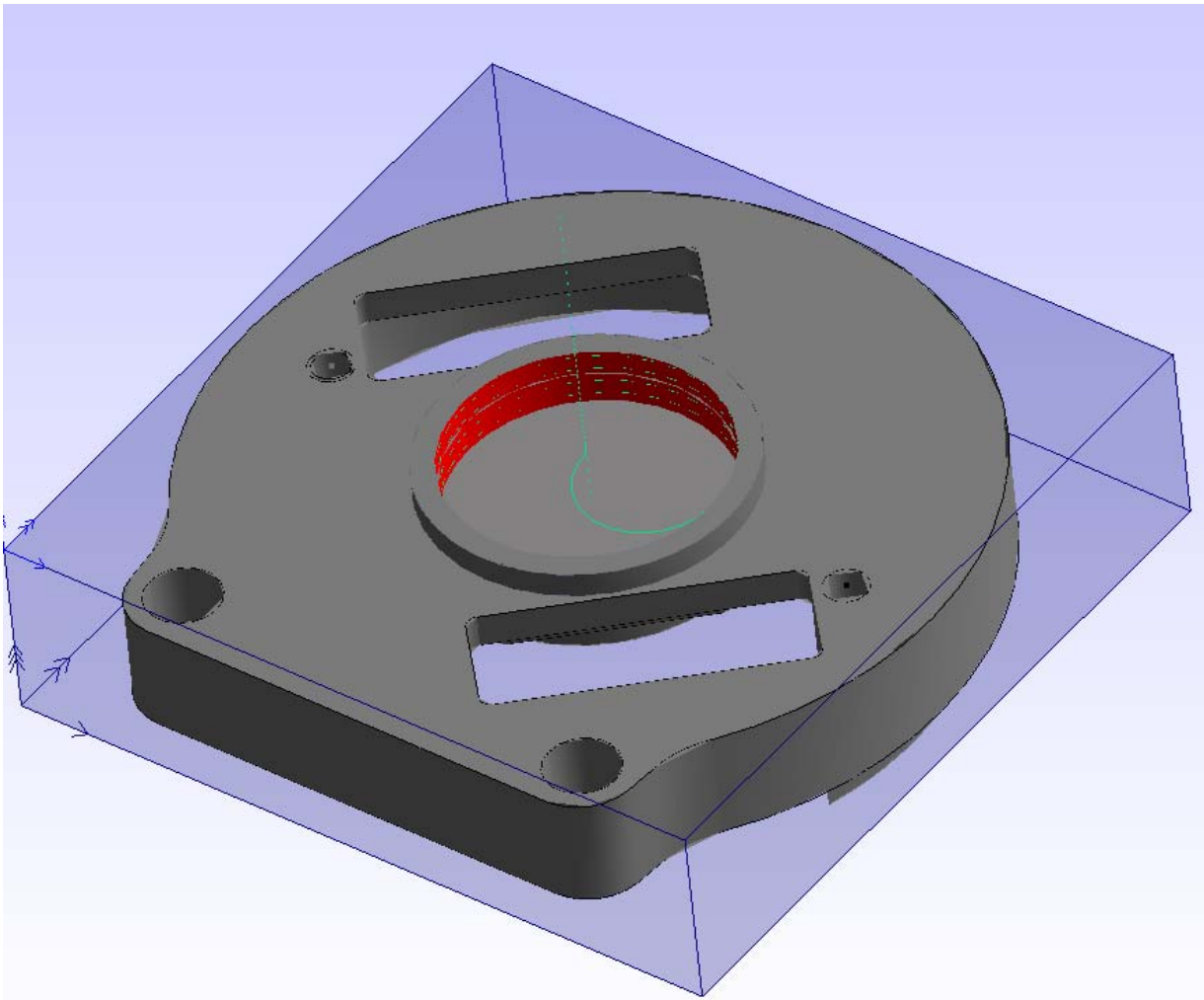


Automatic overlap and safe Z-moves are generated. The Chamfer Feature generates a program in 2-3 seconds.

FEATURE HELIX MILL

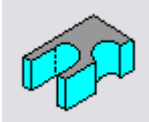


Large Holes, which require a Helix milling process, are calculated using cutter compensation of center of tool. Multiple surfaces may be selected simultaneously. The Software first informs the Operator of the diameters and depths or the selected holes in order that the correct tool can be decided upon.

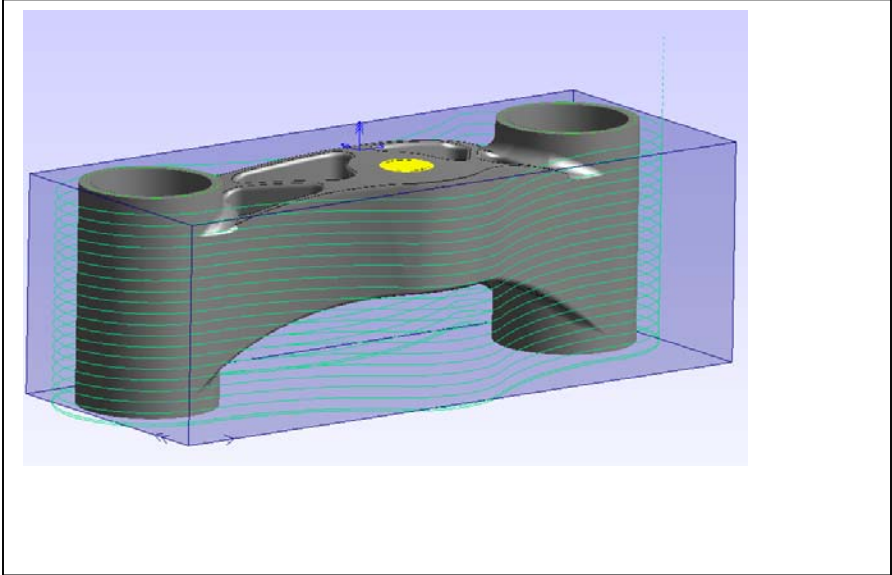


The Helix Milling Feature generates a program in 1 second.

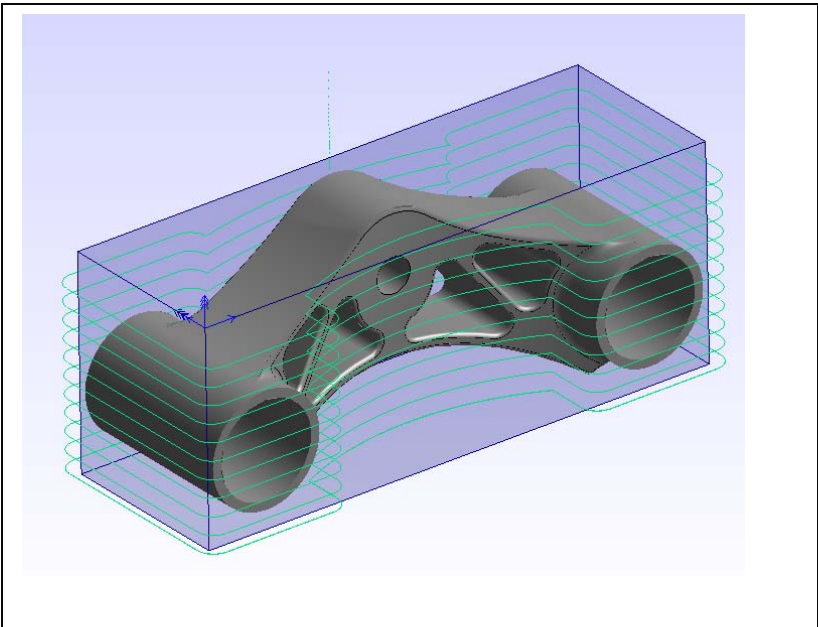
FEATURE SILHOUETTE



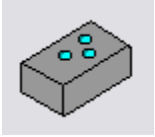
3-Dimensional products can be rough machined regardless of their shape and complexity.



The Silhouette Feature generates a program in 5 seconds. Subject to complexity and number of machining passes this may take a little longer.

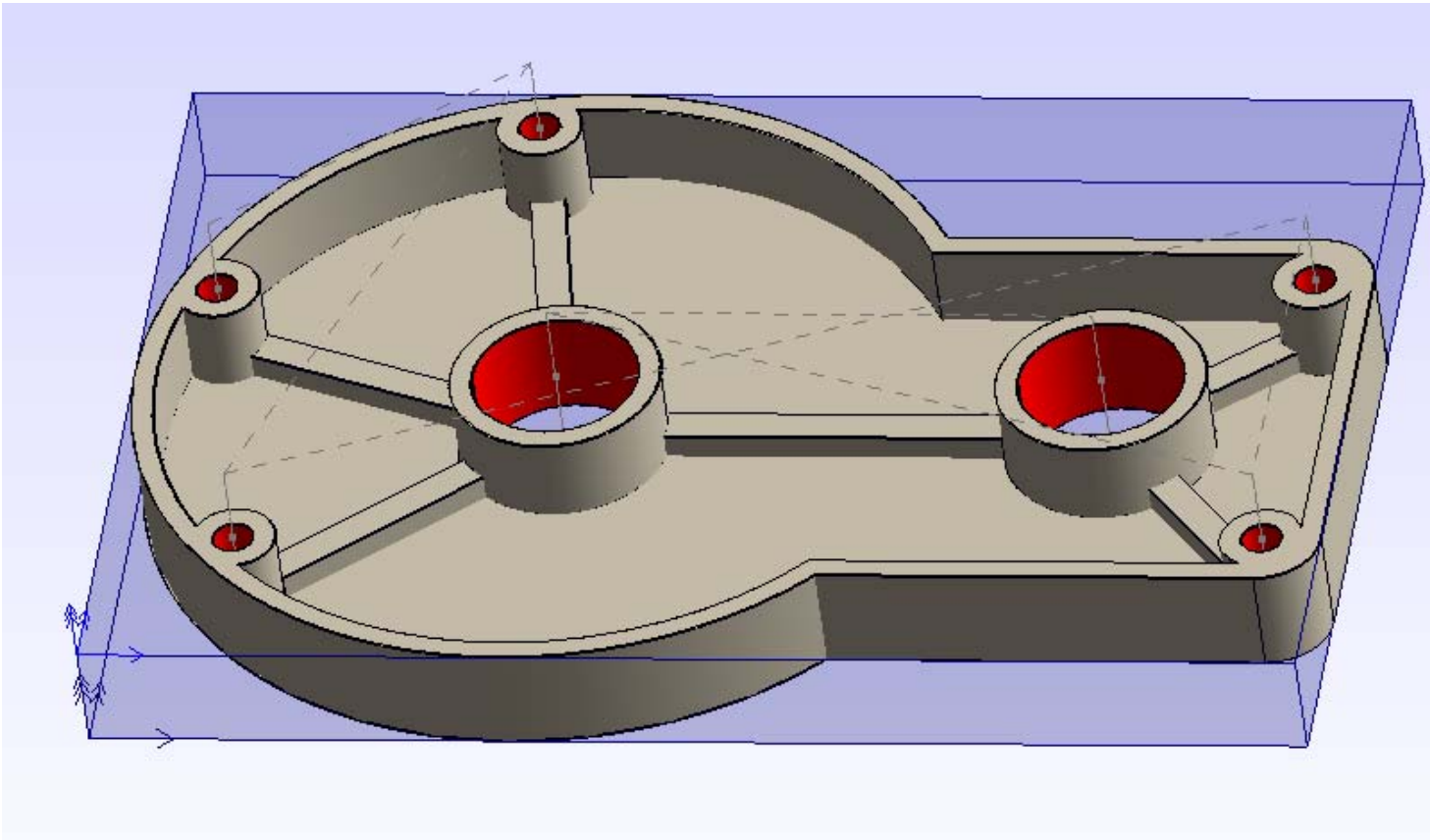


FEATURE DRILL



The critical information for a selection of drilled holes (Cylinders) is the Diameters and Depths.

The Drill Feature informs the Operator with a list of the selected holes. Centering can be started at the push of one button for all holes. Drill operations such as tapped holes can be added with automatic tool changes in the NC Program. The drilled-hole depths can be extended in order to remove sharp burrs. Drill tip angle is automatically added to the drilled-hole depths.



The Drill Feature generates a program for multiple holes in approximately 1 second.