# Genesee Manufacturing Company Inc.

# Product Reference Guide

# FREQUENTLY ASKED QUESTIONS

**What are the tools made from?**

4140 steel hardened to 38-40 Rockwell.

Precision ground. The adjusting ring is hardened, and ring face is ground.

The threads on rings and tool are ground threads.

**What is the adjustment range?**

The inserted blade style tools have a 1/8 diametrical adjustment range accurate to .001".

Gen-Dex indexable tool have a ¼ inch range of diametrical adjustment.

The inserted blade tools are standard with either straight or Morse taper shanks.

**What kind of machines can this be used on?**

They can be used on any machine—automatics or CNC subject to size restrictions.

We've found that approximately 50% are used on CNC and more towards CNC as use of CNC grows.

Additionally, the Mini-Dex line of tools is an ideal fit on Swiss machines.

**What are the insert holders made from?**

Insert Holders are made from 4140 steel hardened to 38-40 Rockwell.

The pockets of the insert holders are machined in hardened conditions.

Finished ground to a tolerance of .0003" of overall length.

**What kind of inserts does this hold?**

The standard tool is designed to hold standard ISO inserts 3/8 or1/4 inch.

Genesee offers the tool with ground C tolerance inserts for precision accuracy on finishing cuts.

**Can I use my own inserts?**

Customers can use other manufacturer's inserts that conform with our standard spec. which is a ground insert with a "C" tolerance.

**What's the largest size standard tool?**

Standard size for 2-inch diameters

We can make larger tools as a special order and do so frequently.

**What's the smallest size standard tool?**

Can produce .011 diameters with inserted blade finishers.

Indexable to .040"

**What would I use a hollow mill for?**

The Hollow mill is a good roughing tool—if you are trying to rapidly reduce or remove material.

The hollow mill is a good finishing tool—it encapsulates the part, has 4 cutters with accuracy. There is concentricity when producing multiple diameters.

**What are frequent special applications for hollow milling?**

Multiple operation tools.
Face Grooving, and Trepanning.
Spherical radius generating
Pointing, Tapers
Preparing pre thread diameters for pipe tapers and/or standard threads.
Steps, multiple diameters.
A greater tool on a CNC machine—interpolate with hollow mill.
Facing, Centering, and/or Chamfering ends of bars.
Tube end Finishing and Deburring.

**Can you chamfer the end of the part?**

Depending on length, you can put the chamfer in the blade. We can also put a chamfering tool in the hollow mill.

With indexable tools, we can put a chamfering tool in or often make a special holder with 2 inserts.

**Do you offer a blade regrinding service?**

To enable our customers to secure more production at lower cost we offer regrinding any set of Genesee hollow mill blades at a cost that will not exceed 50% of the cost of a new set of blades.

**Do you Guarantee your products?**

Any tool we have recommended or designed is guaranteed to perform if used properly in an adequate machine tool and in accordance with our recommendations.

**What should I do if there is a problem?**

Experience has taught us at Genesee manufacturing company that 95% of all problems our customers may encounter in the use of our hollow mills and special production tools are solved once we know the details of the specific problem.

# SETTING A HOLLOW MILL TO SIZE

Genesee adjustable inserted blade hollow mills are most easily set using a plug gauge, finished part, or a piece of stock turned to the required diameter. Once you have one of these items follow the procedure below to set any hollow mill to the desired size.

For Gen-Dex® tools the same procedure is followed, adjusting the "insert holders" in the same manner as you adjust the "blades":

1. Be sure the blades and the tool are clean (no chips or burrs).
2. Slide blades in tools so blade clamp washers are in blade slots.
3. Snug blade clamp screws just tight enough to hold blades down in slot but allow blade to slide in and out of tool (with some resistance).
4. Push all blades against ground ring face.
5. Insert gauge and advance ring forward until you have achieved a slip fit with the gauge. Remove gauge before tightening blade clamp screws.
6. Tighten clamp screws. 5-7-foot pounds torque is all that is necessary.
7. Recheck with gauge. If undersize, loosen blade clamps until just snug as in step #3 above. Back ring off slightly and gently tap the blades back against the ring face using a brass hammer. (do not tap directly on carbide). Tighten blade clamp screws again and recheck with gauge.
8. When desired size is achieved tighten ring set screw. (finger tight only).
9. For Gen-Dex® tools only, use a t-15 size torx wrench to tighten the screw on inserts. The insert should draw back against the rails of the pocket and seat as it is tightened. Only 2.9-foot pounds (35 inch pounds) of torque is necessary to tighten.

# GENESEE REGRINDING SERVICE

To enable our customers to secure more production at lower cost we offer regrinding any set of Genesee hollow mill blades at a cost that will not exceed 50% of the cost of a new set of blades.

When returning blades for regrinding please follow these instructions:

1. Do not send complete hollow mill. Just remove the blades and **return in sets.**
2. Provide the style and capacity of the Genesee hollow mill that the blades are being used in
3. For special blades include sample part or prints so that we may grind your blades most efficiently for your job.
4. With rake angle hollow mill blades advise the part or stock diameter for which the blades should be ground, or if you wish the throat length ground to one of our standard throat lengths.

# GRINDING GENESEE HOLLOW MILL BLADES

We recommend grinding Genesee hollow mill blades in a Genesee grinding fixture used in a three-way vise as shown in the attached pages from our catalog (see appendix).

The standard clearance angles set out on the charts in the appendix, taken from our catalog, are established as producing best results for general work. On some materials slight changes from those angles may give better results.

Blades used in rake angle style hollow mills are subject to the special instructions set out below with regard to maintenance of the “t-12” dimension.

Chattering indicates too large a clearance angle on either the front or the throat (edge tangent to the work) of the blade or, on rake angle blades, a t-12 dimension which is too short. Chattering may also be caused by a lack of rigidity in either the machine spindle or the work.

Tearing or rough finish indicates too small a clearance angle on the throat of the blade or, on rake angle blades, a t-12 dimension which is ground too long.

# MAINTAINING T=12 DIMENSION ON BLADES USED IN RAKE ANGLE TOOLS

The full efficiency of a Genesee adjustable hollow mills with 12-degree rake angle depends upon the shear action of the blades. The throat length of the blades must be ground so that when the mill is adjusted to the finish diameter the blades are the correct distance ahead of center. The smoothness of the finish depends upon the efficiency of this shearing action. The blade grinding instructions contained in our catalog and our t=12 formula is designed to show correct angles and throat lengths for ordinary work.

Maintaining the proper t-12 dimension on rake angle blades requires that adjustment of the hollow mill to size occur through the maintenance of the t-12 dimension, **not** through adjustment of the blades with the ring. Adjustment with the ring should only be necessary to make the small adjustments necessary to set the hollow mill to size after regrinding.

Genesee carries hollow mill blades in three "standard" throat lengths for the large, midrange and small diameters of the tool as illustrated in the blade number explanation sheet in the appendix. Often these standard throats are adequate, particularly when hollow milling larger diameters. As the diameter being hollow milled increases, close maintenance of the throat length becomes less critical.

Please remember the length of a chamfer or radius on the tip of the blade is **not part of**, the t-12 dimension. The t-12 dimension only includes the length of the blade "tangent" to the part.

If the t-12 dimension is too short, the blades will be too far behind centerline at the cutting diameter thereby producing chatter and/or rough finish.

If the t-12 dimension is ground too long, then the blades will be too far ahead of centerline at the cutting diameter and produce a tapered part larger at the end of the cut, and/or poor finish.

Please also refer to attached t-12 examples in the appendix which illustrate the above.

# MACHINING RECOMMENDATIONS

## MAXIMUM "PER SIDE" STOCK REMOVAL

Below are guidelines only and always must always yield to the specifics of the particular job. Numbers indicate **"per side"** removal and therefore should be doubled to state total reduction in part diameter.

| BLADE THICKNESS | BRASS OR ALUMINUM | STEELS |
|---|---|---|
| .110" | .180" | .060" |
| .170" | .250" | .100" |
| .230" | .312" | .150" |
| .280" | .375" | .250" |
| .350" | .438" | .312" |
| .470" | .625" | .500" |
| GEN-DEX (3/8 I.C.) | .300" | .250" |

# CALCULATING SURFACE SPEEDS AND R.P.M.

Always calculate surface speed on the beginning stock diameter **not** the finish stock diameter. Many customers will mistakenly attempt to calculate the surface speed based on the finished diameter thus selecting too high an RPM thereby reducing tool life.

T.c.t. tipped blades turning steel should be run between 200 and 350 surface feet. H.s.s. blades turning steel should be run between 60 and 100 surface feet.

# FEED RATES

H.s.s. blades should be fed to allow a minimum of .001 per blade. Carbide a minimum of .002 per blade. Feed should not exceed .005 per blade on roughing cuts or .003 per blade on finishing cuts.

Generally running any hollow mill between .002 and .003 per blade will produce the best combination of productivity and surface finish. If a better surface finish is required, lower the feed rate.

# HOLLOW MILLING TOLERANCES

STEEL (as a general rule):

- .001 total tolerance, for finishing cuts on a production basis.

BRASS or ALUMINUM (as a general rule):

- .0005 total tolerance for finishing cuts on a production basis.

If necessary, ask our engineering department to address your specific application. Call (585) 266-3201 for an answer.

# TROUBLESHOOTING SPECIFIC PROBLEMS

## CHATTER

Listed below are the primary causes of chatter problems when using Genesee hollow mills

1. Insufficient feed rate. Customers tend to reduce feed rates when they should be increasing feed rates (see “feed rates” above). With a four bladed hollow mill start at 6 to 10 thousandths per revolution and work up from there.
2. Part is loose. Does the fixturing hold the part?
3. Spindle is moving or loose or tool extension is too far from or in the spindle.
4. Excessive primary clearance angles. (see “grinding hollow mill blades” above).
5. (rake angle hollow mills only). Blades are set too far behind centerline because the throat length, (t-12 dimension) is ground too short.

If feed rate is the problem but cannot be increased, and a four bladed tool is being used, removing two opposite blades will effectively double the feed rate.

With part, fixture or spindle instability problems improvement can sometimes occur by changing the clearance angles on the blades. Look to the front of the blade first. On a straight cut hollow mill reduce the standard 10 degrees to 5 degrees. If it still chatters, reduce to 3 degrees. Next look at the primary angle on the throat of the blade. On the straight cut tool, the standard is either 1 degree or 0 degrees. Look at reducing that angle to a negative 3 degrees, this allows you to encapsulate the part. This will reduce the cutting efficiency the tool but will produce a better finish.

On rake angle hollow mills, the same process would begin by decreasing the front primary to 16 degrees and the throat primary to 3 degrees and then proceeding to experiment from that point on.

# MISALIGNMENT, PART MOVEMENT OR DEFLECTION

## STATIONARY TOOL

Because a hollow mill has a balanced cutting action, it will try to center the part, thus moving the part. When the tool is stationary, as you reach the end of the cut, near the spindle, the part will no longer move to let the stock get into the center of the hollow mill. The tool will cut small at the end of the cut and leave drag marks when retracted. Misalignment can be proven to a customer if the tool is stationary by removing two opposite blades. Depending upon whether the misalignment is horizontal or vertical moving two opposite blades will correct the misalignment. By turning the tool to the perfect spot, you should be able to cut a perfectly straight part with two opposite blades.

## ROTATING TOOL

If the tool is rotating but misaligned, you should get a correct sized part eccentrically off center. You can still have dragged mark problems and a slightly tapered cut, even when the tool is rotating, if there is part deflection or movement. In that case, drag marks should appear only on one side of the part.

If all blades are not ground alike, this can produce the same effect as misalignment if the tool is stationary. If the tool is rotating with blades that are not identical, the tool probably cuts slightly oversize with one blade doing all the cutting.

## DRAG MARKS

Drag marks are only caused by one of the problems discussed above, i.e. misalignment, part movement or deflection. Customers sometimes ask if they can correct drag mark problems by chamfering blades. Chamfering blades will probably increase the optimum cutting action of the blades, but it will not correct for drag marks. Only correction of the misalignment, part movement or deflection problem will solve the drag mark problem.