J.W. Done Corporation

Home of Cross-Drilled Hole Deburring Technology

"The Next Tool-After-Drill Deburring Technology"

GENERAL USAGE GUIDELINES

ORBITOOL® DEBURRING TOOL is a new and unique cross-hole deburring technology. And, as with every new technology, it takes the user some time to learn how to utilize it to the most of its potential.

We at J.W. Done Corporation encourage the user to experiment beyond the general guidelines presented below. Safety, however, must not be compromised.

SAFETY:

Never spin ORBITOOL DEBURRING TOOL before inserting the cutter into the bore of the part. Spinning the tool outside the part may result in the separation of the cutter and injury to the operator.

PRINCIPLES OF OPERATION:

Set up:

Trim the shaft of the cutter to the length desired. The shaft length determines the cutting pressure applied by the Orbitool. A shorter shaft extension results in an increased cutting pressure. If you have any questions about a recommended length for a particular application, please do not hesitate to contact us.

Tool Path:

Fig. 1: Position tool on the axis of the hole directly above the entrance to the part.

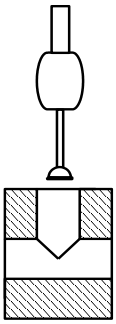


Fig. 2: Move tool inside the hole to a depth close to the intersection.

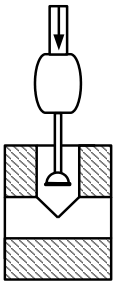


Fig. 3: Move tool to the side of the hole until the axis of the tool is located at the diameter of interpolation (see formula on pg. 4)

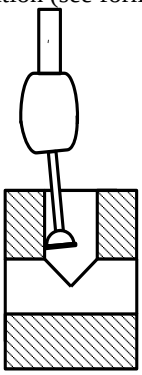
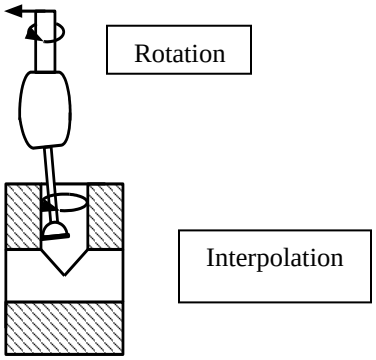


Fig. 4: Begin rotation of tool. Begin helical interpolation. **The direction of Interpolation and tool rotation must coincide (both clockwise)**



Phone (888) 535-3663 510-784-0667 510-784-0668 Fax (510) 732-6078
www.jwdone.com

Fig 5: Advance tool to the center of the second hole. Stop rotation and interpolation of the tool.

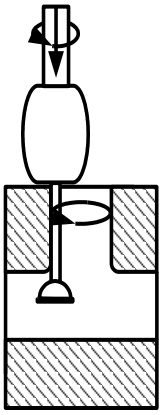


Fig 6: Move tool to the center of the bore.

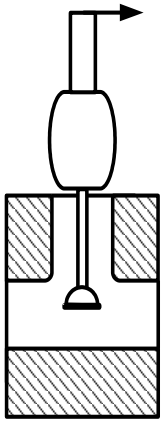
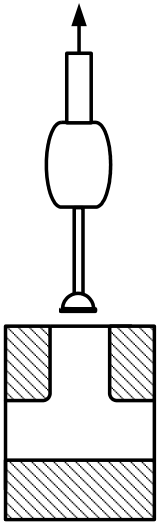


Fig 7: Withdraw tool.



Speeds and Feeds:

Excessive dwelling and a too slow feed rate can lead to disproportionate material removal and creation of chamfer as shown in *Fig.8 and 9* below. This may also lead to creation of secondary burrs.

Fig. 8:

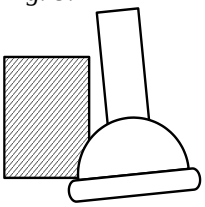
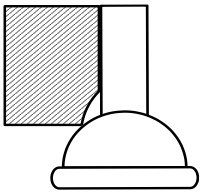
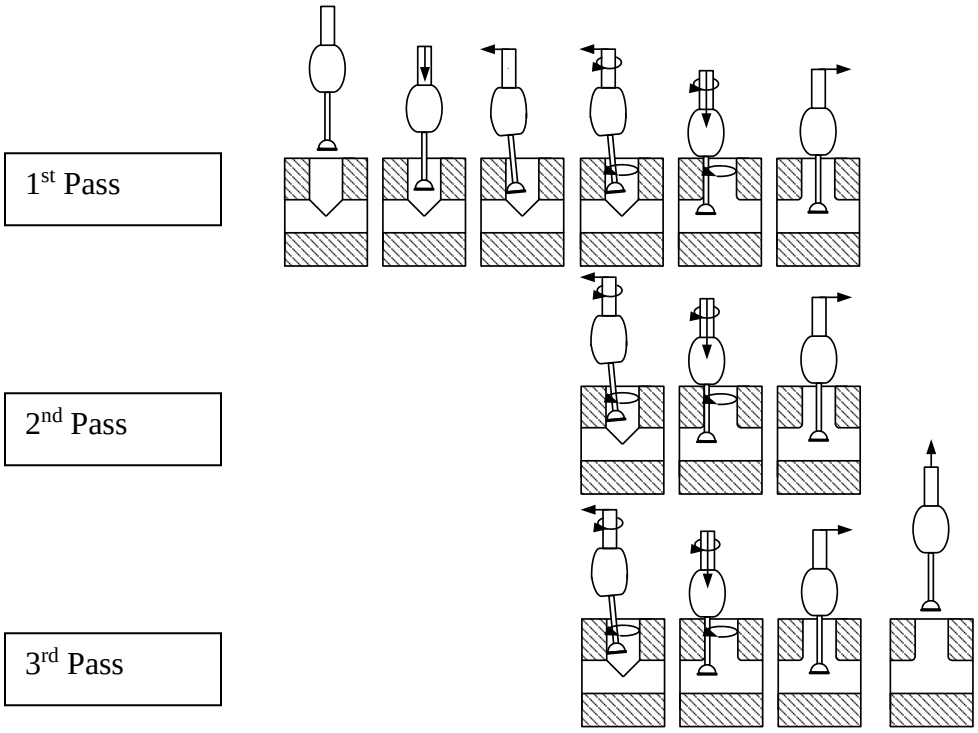


Fig. 9:

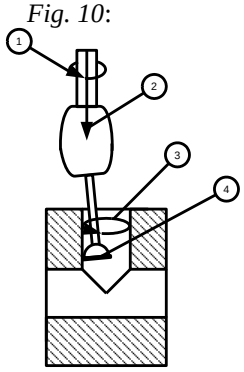


To prevent this condition from occurring the strategy of running multiple light passes can be utilized. Reduction of rpm on the last pass will improve the surface finish. Alternatively, the parameters in a single pass should be dialed back (faster feed rate, increased pitch, lower RPM, longer shaft).



Each of the variables described below and shown in *Fig. 10* can result in the aforementioned situation and must be considered:

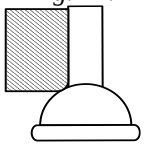
Fig. 10:



1. RPM of the ORBITOOL DEBURRING TOOL.
The tool is rotated in a clockwise direction. The faster you spin the shaft, the more material is removed. Suggested rpm for the ORBITOOL DEBURRING TOOL is 5,000 to 15,000 rpm. In general, like with most machine tools, smaller cutters should be run faster, larger cutters should be run slower.

2. PITCH (Z-Axis "step" per circular interpolation)
Helical interpolation pitch is responsible for the amount of material removal as well as the shape of the RADIUS generated by the ORBITOOL. On softer material such as aluminum, slow feed rates will result in chamfer creation instead of their RADIUS at the intersection. A pitch of 0.002 to 0.025 inches per one circular interpolation are recommended. The greater the pitch, the less material removed. We recommend starting at the "coarse" end (0.025") and decreasing as necessary.

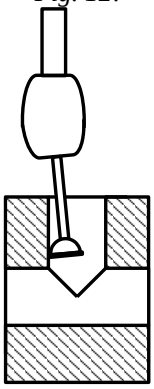
Fig. 11:



3. RPM OF INTERPOLATION (FEED RATE)
There are three possible ways to move ORBITOOL DEBURRING TOOL around the hole in the part:

SIDE VIEW

Fig. 12:



TOP VIEW

Fig. 13:
Tool is spinning.
Part is spinning.

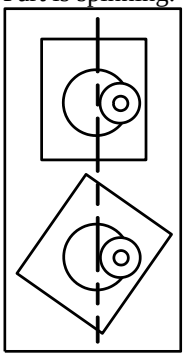


Fig. 14:
Tool is spinning and moving around hole.
Part is stationary.

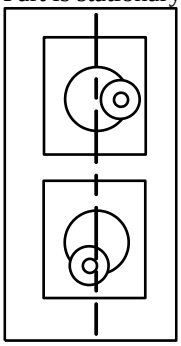
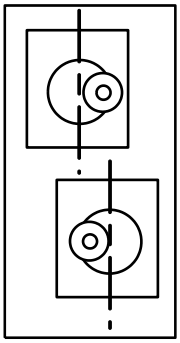


Fig. 15:
Tool is spinning.
Part is moving in X and Y axis.



Generally, using lathe terminology, 20 to 100 rpm is an acceptable range for interpolation (20-100 "orbits" per minute). This can be converted to mill terminology, to obtain a corresponding *ipm* (inches per minute) feed rate.

The diameter of the interpolation should be: **$D = D_h - D_s$**

D diameter of the interpolation
D_h diameter of the hole
D_s ORBITOOL® DEBURRING TOOL cutter shaft diameter

4. TOOL STIFFNESS.

Clearly, the stiffer the tool, the more aggressive it is. This stiffness (i.e. cutting pressure) is easily adjusted by varying the Orbitool cutter shaft extension. The shorter the shaft, the greater the amount of cutting.

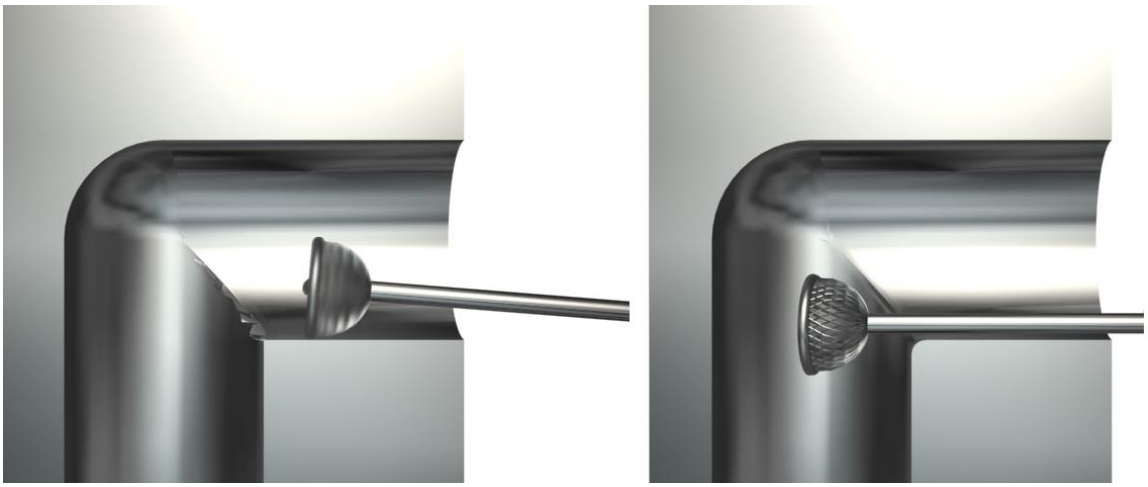
Manual Deburring

Manual deburring is possible with ORBITOOL® deburring cutters mounted in Foredom® or Dremel® type handpieces, or similar electric hand grinders (air grinders are not recommended due to their high speed nature).

The advantage of using ORBITOOL instead of regular burrs is that the patented protective disk prevents the burr from damaging the part. As ORBITOOL spins, the disk rides on the surface of the part, until it falls over the edge where the burr is located. This triggers deburring. As the tool is pulled back, deburring stops automatically and the disk rides on the surface again.

Our preferable unit of choice is pictured below. It is compact and convenient to use. It is shipped with sleeves for all ORBITOOL cutter sizes. The part number for the unit is 30004.

Because the ORBITOOL cutter is flexible, and has a protective ring, it is important to modify the deburring technique normally used with hand grinders. Instead of focusing on riding along the edge (that needs to be deburred), as one would normally do, the operator has to use an axial motion with the ORBITOOL, using repeated in-and-out strokes. This can be likened to using a file on the edge.



When deburring manually, the operator imparts the necessary helical motion by hand.

The results obtainable manually will be superior to other hand deburring methods because of the unique design of the ORBITOOL cutter. As with other manual methods though, the results depend a great deal on the skill and attention of the operator.

As with all rotary tools, operator safety is the paramount concern when deburring manually. Care must be taken to ensure the cutting end of the tool is fully inserted into the hole of the workpiece before the tool is spun. In addition, the tool rpm must be controlled and must never exceed 12,000 rpm, even when the tool is properly inserted into the workpiece.

FAILURE TO OBSERVE THESE PRECAUTIONS MAY RESULT IN CATASTROPHIC FAILURE OF THE ORBITOOL DEBURRING TOOL, RESULTING IN POSSIBLE INJURY TO THE OPERATOR AND BYSTANDERS, AS WELL AS DAMAGE TO THE WORKPIECE AND EQUIPMENT IN THE AREA.

All other customary precautions associated with using cutting tools in powered hand tools must be observed as well. These precautions include, but are not limited to: use of guards, safety apparel (eye wear, face shields, hair nets, etc.), electrical safety, etc.

SAFE AND PROPER USE OF ORBITOOL DEBURRING TOOLS IS THE RESPONSIBILITY OF THE USER.

J.W. Done Corporation

Home of Cross-Drilled Hole Deburring Technology

CNC Programming for ORBITOOL® deburring tool:

PRECAUTIONS: NEVER SPIN TOOL WITH HEAD OUTSIDE OF HOLE
DO NOT SPIN TOOL IN EXCESS OF 10,000 RPM
OBSERVE ALL RELEVANT SAFETY PRECAUTIONS

Note: All dimensions given herein are in inches

1. Rapid moves are OK provided adequate clearances are verified in advance. Single blocking is always good practice.

2. Move the tool in rapid approach mode to hole centerline to within .100 of start of hole.

3. Move the tool into the hole in rapid mode along hole centerline to within .05 of start of intersection.

4. Move the tool in rapid mode to the preload position against wall of bore. Preload position is the amount the tool centerline is offset from the hole centerline and can be calculated as follows:
   (D-d)/2 -.002 = amount to offset where D is the hole diameter, d is ORBITOOL shaft dia[1] and .002 is optional clearance.

5. Spin tool and part CW (viewed from spindle towards part). On mills use helical interpolation, similar to hole/thread milling. Refer to *User Guide* for discussion of feeds & speeds and troubleshooting.

6. Advance ORBITOOL into hole until the entire head of the tool* is completely past the line of intersection of the bores.

7. **CAUTION**: In blind holes make certain the tool does not crash into far wall of intersecting bore.

8. Stop tool and part rotation.

9. Move the tool in rapid mode to hole centerline.

10. Retract the tool in rapid mode. For multiple passes, i.e. when generating a radius, retract to position described in step 3. For single pass, i.e. burr removal, retract completely out of hole to home or tool index position.

11. Coolant use is optional. High-pressure coolant should be avoided as coolant stream may "lift" tool and impair cutting. In normal operation there is negligible heat generation at the tool. A "hot" tool is indicative of overworking and may lead to premature failure of the tool.

Programming Suggestions:

1. Keep original part program intact. Create subroutine to deburr cross-drilled features.
2. If possible, program a subroutine with variables for: feed rate(s), rapid moves, spindle rpm (lathes), tool rpm, tool preload offset, etc.
3. Deburr ***after*** part is completely machined.
4. Ensure that chips are not packed into bores before deburring. Include optional stops for chip removal as necessary.

[1] Refer to specification drawings for nominal tool dimensions.

J.W. Done Corporation

Home of Cross-Drilled Hole Deburring Technology

INSTRUCTIONS:ORBITOOL® DEBURRING MACRO

The sample macro that follows these instructions is provided as an aid to users programming CNC mills for ORBITOOL deburring tools. The macro was written for FANUC controls. Modification of the macro may be necessary in order to run it in your particular CNC machine.

I. ENTER THE MACHINES MAXIMUM FEED RATE IPM INTO VARIABLE #106
II. IF YOUR MACHINE HAS A MAXIMUM FEED RATE OF 200 IPM THE LINE SHOULD READ #106=200

G65 P9160 K____S____Z____Q____H____V____

G65 P9160 is the macro call number to start the deburring operation. Input the following information on the G65 P9160 line.

a) **K** *REQUIRED*-SHANK DIAMETER OF THE DEBURR TOOL

b) **S** *REQUIRED* -SPINDLE SPEED (GENERALLY 2000-12,000 RPM)

c) **Z** *REQUIRED* -INCREMENTAL DISTANCE THE TOOL WILL TRAVEL IN THE Z-AXIS FROM THE START POSITION OF THE TOOL. THIS VALUE MUST BE POSITIVE. THE Z VARIABLE IS THE DISTANCE FROM START DEPTH TO FINAL DEPTH PLUS THE LENGTH OF THE TOOL HEAD. NOMINAL VALUES ARE: .100 FOR 1/8, .161 FOR 1/4, AND .216 FOR 3/8 DIAMETER TOOL. CHECK ACTUAL TOOL.

d) **Q** *REQUIRED* -STEP DEPTH. THIS IS THE INCREMENTAL DISTANCE THE TOOL WILL TRAVEL IN THE Z-AXIS EACH CIRCULAR MOTION AROUND THE HOLE

e) **H** *REQUIRED* -HOLE DIAMETER

f) **V** *REQUIRED* -RPM OF INTERPOLATION. THIS VARIBLE REPRESENTS THE NUMBER OF TIMES PER MINUTE THE TOOL WILL TRAVEL AROUND THE HOLE (GENERALLY 20-100 RPM). FEED RATE WILL INCREASE WITH THIS NUMBER

EXAMPLE PROGRAM MH-40.
NOTE: PRIOR TO CALLING THE MACRO THE CUTTER IS SET AT THE CENTER OF THE HOLE, AT THE PROPER START DEPTH (APPROX. .05 AWAY FROM THE FRONT OF THE HOLE). AT THE END OF THE MACRO THE TOOL WILL BE AT THE FINAL DEPTH AT THE CENTER OF THE HOLE.

```
(.276 JW-DONE DEBURR TOOL)
T()
G90G0G54.1P1B0
G0X0Y0
G43Z.2H38
G0Z-.180
G65P9160K.094S6000Z.250Q.01H.3125V50
G0Z.2
    (ABSOLUTE VALUE OF THE FINAL DEPTH WILL BE Z-.430)
```

J.W. Done Corporation

Home of Cross-Drilled Hole Deburring Technology

!!!!!!!!!!CAUTION!!!!!!!!!!
ALWAYS HAVE THE TOOL POSITIONED INSIDE THE HOLE BEFORE RUNNING THIS MACRO

```
%
O9160
(APM 5/10/2002)
(JW DONE DEBURRING MACRO)
(K=SHANK DIA #6)
(S=SPINDLE SPEED #19)
(Z=INCREMENTAL DEPTH #26)
(Q=STEP DEPTH #17)
(H=HOLE DIA #11)
(V=INTERPOLATION RPM #22)
IF[#6EQ#0]GO1000
IF[#19EQ#0]GO1001
IF[#26EQ#0]GO1002
IF[#17EQ#0]GO1003
IF[#11EQ#0]GO1004
IF[#22EQ#0]GO1005
IF[#26LE0]GO1006
#100=#4003
#101=[#11-#6]/2-.01
#102=ROUND[#26/#17]
#103=[[#101*2]*3.14159]*#22
#104=1
#105=#26/#102
#106=
IF[#103GT#106]THEN#103=#106
G91
G0X#101
S#19M3
G3Z-#105I-#101F#103
N1WHILE[#104 LE #102]DO1
#104=[#104+1.]
Z-#105I-#101
END1
M5
G0X-#101
G#100
#100=0
#101=0
#102=0
#103=0
#104=0
#105=0
#106=0
M99
N1000#3000=1(NO SHANK DIAMETER DEFINED)
N1001#3000=2(NO SPEED DEFINED)
N1002#3000=3(NO DEPTH DEFINED)
N1003#3000=4(NO STEP INCREMENT DEFINED)
N1004#3000=5(NO HOLE DIAMETER DEFINED)
N1005#3000=6(NO INTERPOLATION RPM DEFINED)
N1006#3000=7(NEGATIVE VALUE IN Z VARIABLE)
%
```