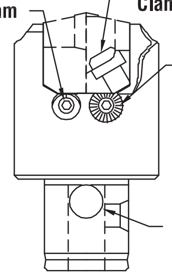
Technical Information

Operating Instructions: Twin Cutter

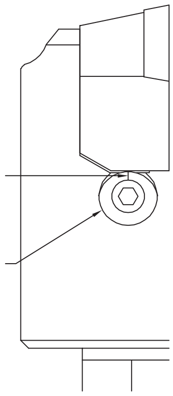
INSTALLING INSERT HOLDERS:



1. Remove the two insert holder clamps and set aside.
2. Place the insert holders in the head, making sure the slot in the bottom of the insert holder engages the head of the diameter adjusting screw.
3. Install the insert holder clamps (cap screw on size two). Make sure that the nut is flush with the end of the screw.
4. Snug up the insert holder clamps.

It is recommended that the Parlec twin boring heads be adjusted on a tool presetter. Refer to Parlec Parsetter catalogs for more information.

ADJUSTING FOR BALANCED CUTTING: (see next page).



Balanced cutting allows both cutting edges to work simultaneously. A properly balanced twin cutter may be fed at almost four times the rate of a single cutter. Make sure the height cam is located with the reference mark (lowest point) in the vertical position as shown.

1. Loosen the insert holder clamps. Re-tighten enough to put drag on the insert holder.
2. Adjust the diameter by turning the adjusting screw. Always adjust in the clockwise direction.
3. Tighten the insert holder clamps.
4. Repeat for the second insert holder, adjusting both to ± .001 on the diameter.

ADJUSTING FOR STEPPED CUTTING: (see page 150).

Stepped cutting allows removal of more metal since each insert is set at a different diameter.

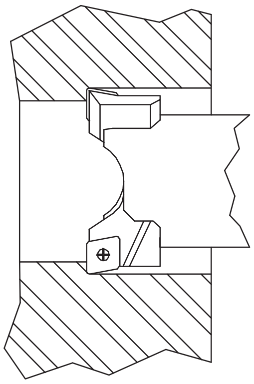
1. Make sure the height cam is located with the reference mark in the vertical position as shown.
2. Loosen the insert holder clamps. Re-tighten enough to put drag on the insert holder.
3. Adjust the diameter by turning the adjusting screw. Always adjust in the clockwise direction. Set the diameter of each insert to remove approximately one half the material.
4. Using the cam screw, adjust the inner cutting edge so that it has a lead over the outer cutting edge. This lead should be a minimum of 1 1/2 times the feed per revolution.
5. Tighten the insert holder clamps.

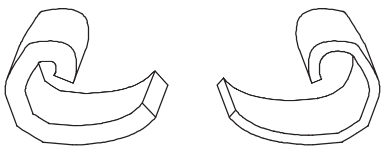
NOTE: When using stepped cutting, the feed rate must be that of a single cutter.

RECOMMENDED TIGHTENING TORQUE IN LBS.

Rough Head	Insert Holder	PC Screw
PC2-4205	12	36
PC3-4305	36	48
PC4-4405	48	72
PC5-4505	72	96
PC6-4605	72	120

Technical Information
Balanced Cutting





Balanced cutting occurs when both inserts are set to exactly the same height. This height balancing is much more important than diametral balancing. A slight difference in height, even that caused by the insert tolerance, can have a dramatic effect upon the tool's performance. This is particularly true in the case of long chipping materials.

Example of unbalanced cut:

- Feed rate .016 IPR.
- Insert "A" is .003 higher than insert "B." (The tolerance on an M style insert is .002-.004.)
- The material removed by insert "A" is .008" + .003" = .011"
- The material removed by insert "B" is .008" - .003" = .005"
- The chip taken by insert "A" is over twice as thick as that taken by insert "B".

The difference in cutting forces caused by the differences in insert height illustrated above can have the following effects on the bar's performance:

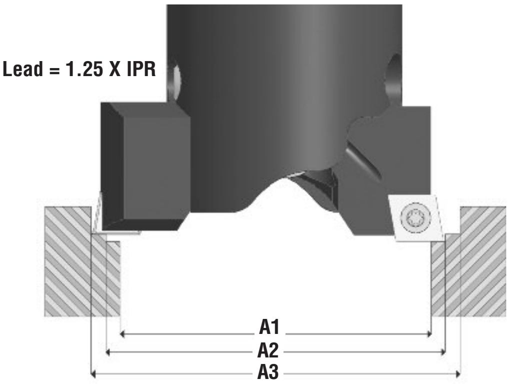
1. Possible wobble or chatter; extra load on the machine tool;
2. Generally, the bore diameter becomes larger than the set diameter;
3. Uniform chip formation is not possible, making it difficult to break and clear chips.

Maximum Allowable Setting Difference Between Inserts For Best Performance:

MAXIMUM SETTING DIFFERENCE

Bore Diameter Range	Insert Height	Cutting Diameter
.95 - 1.31	.001	.008
1.26 - 1.74	.001	012
1.58 - 2.17	.001	.012
2.06 - 2.83	.002	.016
2.61 - 6.00	.002	.016
6.00+	.002	.020

Technical Information
Stepped Cutting



Stepped cutting is utilized when heavy depth of cut is required. The inserts are set at different diameters. The insert cutting the smaller diameter is given axial lead 1.25 times greater than the feed per revolution over the other insert. Use only insert holders with 0° lead. Stepped cutting allows 1.75 x the depth of cut per tables on page 144. Feed rates must be reduced to .5 x appropriate value.

Rules of Stepped Cutting:

1. Use insert holders with 0° lead.
2. Set height in inner cutting edge to provide lead 1.25 times greater than the feed per revolution.
3. Feed rate as roughing with a single cutter.
4. Remove half of the material to be removed with each insert. This should be sufficient for most applications.

To Balance Cutting Forces, Use the Formula Below:

$$A_2 = .7071 \sqrt{A_3^2 + A_1^2}$$

A_1 – Hole starting diameter
A_2 – Inside cutter set diameter
A_3 – Outside cutter set diameter

Technical Information
Roughing Speeds

PARLEC Measurably Better

RECOMMENDED ROUGHING SPEEDS

STEELS

Material	BHN	TT/SN	VN	CT
Carbon Steel C = 0.15%	125	300-450	600-800	650-1000
Carbon Steel C = 0.35%	150	300-500	600-800	625-950
Carbon Steel C = 0.70%	180-250	250-450	550-750	500-750
Alloy Steel 4000	125-200	300-500	550-750	500-750
Alloy Steel 5000	225	200-400	350-525	300-600
Alloy Steel 8000	300	200-400	300-525	350-475
Stainless Steel, Annealed 400 SERIES	150-270	250-400	400-600	475-750
Stainless Steel, Annealed 300 SERIES	150-220	300-425	350-500	550-650
Cast Steel, Low Carbon	150	200-325	450-650	400-550
Cast Steel, Low Alloy	150-250	200-300	250-400	300-425

All values are in SFM.

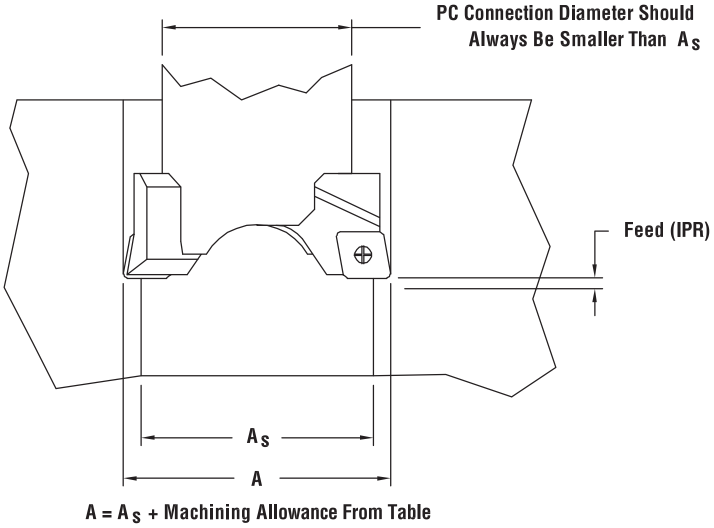
OTHER MATERIAL

Material	BHN	TT/SN	AL/VN	AS	C2
Malleable Cast Iron, Ferritic	110-150	525-700	600-1000	400-900	150-350
Malleable Cast Iron, Pearlitic	150-270	250-400	400-800	300-800	100-250
Grey Cast Iron, Low Tensile	150-220	525-800	400-900	300-1000	200-400
Grey Cast Iron, High Tensile	200-330	350-600	400-700	300-600	150-300
Nodular Iron, Ferritic	125-230	300-500	400-900	450-900	150-375
Nodular Iron, Pearlitic	200-300	250-400	400-900	350-650	100-250
Aluminum Alloys	30-120	–	–	–	600-1200
Aluminum Alloys, Cast	100-130	–	–	–	600-1200

All values are in SFM.

330 BHN = RC 35
250 BHN = RC 24-25
220 BHN = RC 20

Technical Information
Rough Boring Feed Rates & Machining Allowance



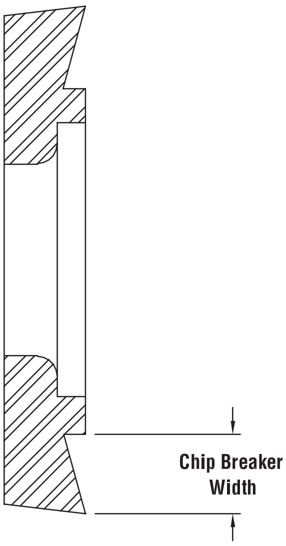
ALLOWANCE & FEED RATES

Twin Boring Head	Material	Machining Allowance on Diameter (Inches) DOC Optimum	Min.	Max.	For Best Finish	Feed Rates (IPR)* Min.	Max.
PC2-4205 (.94 - 1.34)	Steels	.10	.02	.14	.010	.008	.014
	Cast Iron	.16	.02	.24	.010	.006	.012
	Aluminum	.12	.02	.24	.010	.006	.012
PC3-4305 (1.26 - 1.70)	Steels	.12 - .14	.04	.16 - .18	.014	.012	.018
	Cast Iron	.20	.04	.28	.012	.008	.016
	Aluminum	.20	.04	.28	.014	.012	.018
PC4-4405 (1.58 - 2.17)	Steels	.14 - .16	.04	.18 - .20	.014	.012	.020
	Cast Iron	.24	.04	.31	.012	.008	.016
	Aluminum	.24	.06	.31	.014	.012	.018
PC5-4505 (2.06 - 3.30)	Steels	.24	.06	.35 - .47	.018	.012	.024
	Cast Iron	.39	.04	.55	.016	.008	.018
	Aluminum	.39	.06	.55	.018	.016	.024
PC6-4605 (2.61 - 6.00) PC6-4606 & PC7-4705	Steels	.28 - .39	.06	.35 - .47	.018	.012	.024
	Cast Iron	.47	.04	.55	.016	.008	.018
	Aluminum	.47	.04	.55	.018	.016	.024
All Big Bore Above 6.00	Steels	.28 - .39	.06	.35 - .47	.018	.012	.024
	Cast Iron	.47	.04	.55	.016	.008	.018
	Aluminum	.47	.04	.55	.018	.016	.024

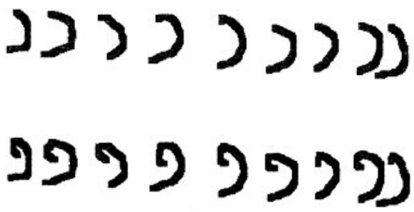
Part numbers in **bold face** are in-stock items.

***Feed rate is based on two cutting edges. When step cutting, multiply by .5.**

Technical Information
Rough Boring Chip Formation



Chip formation when rough boring is very important. An ideal chip should be "C" shaped or look like a six or nine. The chips should also be short as illustrated below.



Proper chip formation is a function of the feed rate and the chip breaker on the insert. Generally the feed rate should be 25% to 35% of the width of the chip breaker.

If the chip does not break, increase the feed rate provided that the following conditions exist:

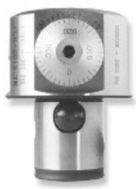
- The part and the fixture are sufficiently rigid;
- The machine has sufficient horsepower;
- The machining allowance does not exceed the stated maximum;
- The tool is not overly long.

If the chip does not break with the increased feed rate, it may be necessary to change to an insert with a smaller chip breaker.

It is important to remember that the thin, wide chip produced from a low feed rate and a large machining allowance is difficult at best to break. As a general rule, the feed rate should be 10% to 30% of the machining allowance.

Technical Information
Operating Instructions: Finish Boring

BORING BAR INSTALLATION



Vernier
Scale side.

Slide Lock
Screw side.

Clamp
Screws side.

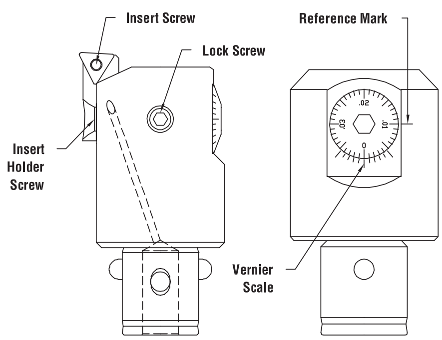
Oil side.

1. Loosen the clamp screws.
2. Insert boring bar and reduction bushing (if applicable) into the carrier. Align reduction bushing with slots 90° to clamp screws.
   Note: We do not recommend using boring bars with flats.
3. Adjust the boring bar to the minimum desired length.
4. Rotate the boring bar to align the insert tip to the alignment mark atop the boring head body. The bar should be above the line for best timing.
   Note: Adjusting the position of the insert tip adjusts the "timing" of the bar which optimizes the surface finish and performance.
5. Tighten the clamp screws.

Torque Specifications:

Slide Lock Screw: 10 ft. lbs.
Boring Bar Clamp Screw: 20 ft. lbs.

BORING HEAD ADJUSTMENT



1. Install insert using the torque wrench provided.
2. Loosen lock screw.
3. Adjust tool position by turning and reading the dial. The dial is graduated in increments of .001" per graduation on the diameter. Fine adjustments of .0001" can be made utilizing the vernier scale. Always adjust by turning the spindle in the clockwise direction.
4. Tighten the lock screw.

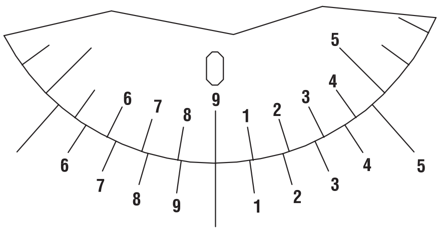
Torque Specifications:

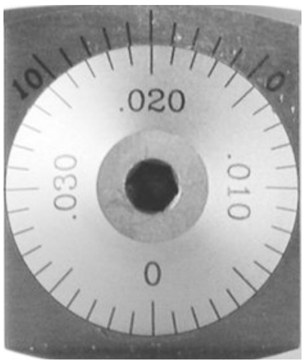
Slide Lock Screw: 10 ft. lbs.

Technical Information

Operating Instructions: Finish Boring

USING THE VERNIER





Rotate the dial to align these marks for .0001" adjustment.

Find the line on the dial that exactly lines up with a line on the Vernier scale. This is at the zero mark on the dial. This is the reference point. Diametral increases in tenths are achieved by turning the dial clockwise until the appropriate line on the dial lines up with the appropriate line on the Vernier scale. For a .0006" adjustment, move the 6 on the dial in line with the 6 on the scale.

Torque Speciflications:

Slide Lock Screw: 10 ft. lbs.

Boring Bar Clamp Screw: 20 ft. lbs.

The vernier graduation is established by counting over the number of tenths adjustment desired beginning with the first graduation immediately to the right of the reference point on the vernier scale. If required, continue counting on the left side of the scale again moving counterclockwise as shown.

FINISH BORING HEAD MAINTENANCE

The Parlec precision boring head is constructed of alloy steel, stainless steel, and a graphite-based epoxy resin. The graphite resin is self-lubricating and requires no added maintenance. To insure long service life, two to four times a year depending on the severity of conditions, the head should be adjusted to the outward end of travel (do not force beyond mechanical stop) and a light spindle or machine oil applied to the exposed section of the carrier. This will lubricate the alloy components of the head. No further maintenance should be required.

Technical Information
Finish Boring Speeds

EFFECTS OF CUTTING SPEED

The effects of cutting speed are illustrated in this chart:

CUTTING SPEED EFFECTS

Variable	Low Cutting Speed	High Cutting Speed
Machining Time	Longer	Shorter
Surface Finish	Coarser	Finer
Probability of Vibration	Lower	Higher

RECOMMENDED FINISHING SPEEDS

STEELS

Material	BHN	TR/TT/SN	C1/C2	AL	AS	TE
Carbon Steel C = 0.15%	125	550-750	600-800	150-350	650-1000	950-1300
Carbon Steel C = 0.35%	150	525-800	600-800	150-350	625-950	850-1200
Carbon Steel C = 0.70%	180-250	425-625	550-750	150-250	500-750	750-950
Alloy Steel 4000	125-200	425-625	550-750	150-250	500-750	750-950
Alloy Steel 5000	225	250-500	350-525	150-250	300-600	400-650
Alloy Steel 8000	300	200-400	300-525	100-200	350-475	400-500
Stainless Steel, Annealed 400 SERIES	150-270	400-625	400-600	150-250	475-750	425-650
Stainless Steel, Annealed 300 SERIES	150-220	450-550	350-500	150-300	550-650	425-650
Cast Steel, Low Carbon	150	325-450	450-650	100-250	400-550	475-600
Cast Steel, Low Alloy	150-250	250-350	250-400	100-250	300-425	400-575
Cast Steel, High Alloy	160-250	–	250-400	75-250	–	400-500

All values are in SFM.

OTHER MATERIAL

Material	BHN	TR/TT/SN	C1/C2	AL	AS	TE
Malleable Cast Iron, Ferritic	110-150	525-700	300-450	600-1100	700-1000	–
Malleable Cast Iron, Pearlitic	150-270	250-400	200-250	600-1000	300-750	–
Grey Cast Iron, Low Tensile	150-220	525-800	325-525	400-1200	600-1600	–
Grey Cast Iron, High Tensile	200-330	350-600	225-400	400-900	350-900	–
Nodular Iron, Ferritic	125-230	300-500	300-400	400-950	450-900	–
Nodular Iron, Pearlitic	200-300	250-400	200-350	400-700	350-700	–
Aluminum Alloys	30-120	–	600-3000	–	–	–
Aluminum Alloys,Cast	100-130	–	600-3000	–	–	–
Extra Hard Steel	50-65RC	–	60-120	–	–	–
Waspalloy, Discalloy, Incoloy	180-250	–	50-200	–	–	150-350
Monel, Inconel	125-250	–	45-90	–	–	90-300
Titanium	100-200	–	120-250	–	–	325-500
Copper, Brass, Zinc	50-150	–	500-1000	–	–	50-1500

All values are in SFM.

Feed rates .002 - .008 IPR. For best finish, feed rate should be approximately 25% of insert nose radius. Refer to next page.

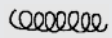
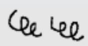
330 BHN = RC 35
250 BHN = RC 24-25
220 BHN = RC 20

Technical Information
Finish Boring Speeds

EFFECTS OF FINISH FEED RATES

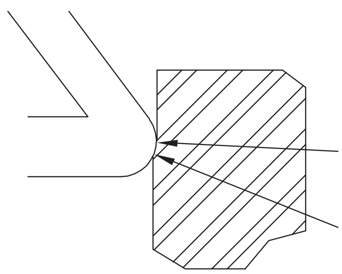
The best surface finish is produced when the tool is fed at approximately 25% of the tool nose radius. The effect on chip formation of the feed rate and depth of cut is illustrated below:

FINISH FEED RATE EFFECTS

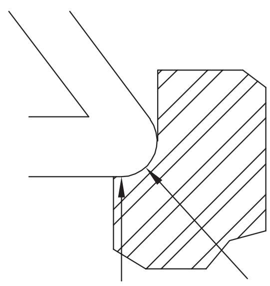
Feed Rates/Depth of Cut	Result	Chip Form	Solutions
Low	Chips Bird Nest: Difficult to Remove		1. Increase Depth of Cut 2. Increase Feed Rate
Moderately Low	Long Stringy Chips: Difficult to Remove		1. Increase Depth of Cut 2. Increase Feed Rate
Ideal	Chips Coil Like a Spring: Easy to Remove		Keep Running Make Money!
Slightly Heavy	Slight Deformation of Chip		If Finish is Bad, Decrease Speed
Heavy	Deformation of Chip Increased Cutting Forces		If Finish is Bad, Decrease Speed
Very Heavy	Severe Deformation of Chip Increased Cutting Forces Heat Build Up		If Finish is Bad, Decrease Speed Good Chip for Roughing

DEPTH OF CUT

The depth of cut must be sufficient to allow the tool to cut and not rub. Too small a cut will cause the tool to be pushed away from the workpiece by the cutting pressure. The depth of cut must be large enough to let the insert bite as it cuts. Depth of cut is a function of material and insert nose radius. **Generally, the ideal minimum diametral depth of cut is equal to the insert nose radius.**



Small Depth of Cut Allows Cutting Forces to Push Insert Out of Cut



Depth of Cut is Large Enough to Prevent Cutting Forces From Pushing Insert Out of Cut

EFFECT OF NOSE RADIUS ON SURFACE FINISH

A larger nose radius produces a better surface finish. Care must be taken not to over feed the nose radius.

- A .016 nose radius insert fed at .004 IPR produces a finish as shown at the right:

*For best finish, generally feed at 25% of nose radius.

- A .016 nose radius insert fed at .008 IPR produces a finish as shown at the right:

- A .016 nose radius insert fed at .016 IPR produces a finish as shown at the right: