



Onsrud Machining Facts: COLLETING / TOOL SELECTION / FIXTURING METHODS

COLLETING

COLLET LIFE SPAN

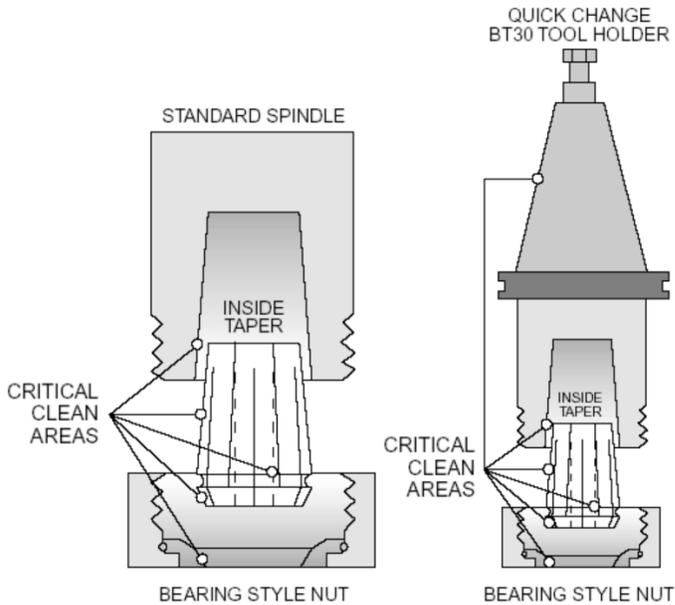
Collets have a life span of 3 months if used 8 hours a day. Replacing the collets will ensure your operation runs consistently and prevents tool breakage. When inserting a tool into the collet make sure the flute fadeout does not enter the collet. This will cause run out and potentially lead to tool breakage. To ensure proper clamping the tool shank should fill, at the minimum, 80% of the depth of the collet. If this can not be achieved, use a collet life plug to ensure a proper clamping effect.

COLLET MAINTENANCE

Cleaning is an essential part of collet maintenance. As material is cut it causes the collet, tool holder, collet nut and spindle to become dirty. This causes your tool to cut in an elliptical fashion which will decrease tool life and cause inconsistency in your operation. Collets, tool holder, and collet nut should be cleaned daily using the Rust Free solvent and a brass brush (OC series 33-21 and 33-10). Refer to the critical areas diagram to see which surfaces must be clean.

CLEANING INSTRUCTIONS

1. Spray the cleaner on the surface and allow it to soak for a minute
2. Use a brass brush to clean the surface thoroughly.
3. Rinse off using distilled alcohol. Feel the surface using your fingers to make sure the surface is clean
4. Apply a small amount of Lubricant T-9 to prevent rusting.



TOOL BREAKAGE

If a condition arises where multiple tools should break follow these steps to solve your problem:

1. Are you using the proper tool for the job?
2. Make sure your collets and tool holders are clean and the tool is colleted properly.
3. Check your speed and feed (is your tool hot?)
4. Is your depth of cut too excessive for the material you're cutting?
5. Do you have any part movement?
6. Do you have ample part hold down?
7. Stop running parts and check with your distributor or Onsrud's Technical Support.

If you have to contact your distributor or Technical Support, have the following information:

1. Machine being used.
2. Material being cut.
3. Part number of tool along with the batch number which is below the part number.
4. Speed / Feed / Depth of cut.
5. Where did the tool break (flute, shank, or in the collet)?
6. How long did the tool work before it broke?

7. Have you done this operation in the past using this tool?

TOOL SELECTION

TOOL MATERIAL

1. Solid Carbide: Primarily used in CNC operations. Material provides best rigidity and long tool life.
2. Carbide Tipped: Incorporates the wear resistance of carbide and the toughness of a HSS body-mainly hand held.
3. HSS: Primarily used in hand routing. Material provides a tough body and sharper cutting edge. Good in CNC.

FLUTE GEOMETRY

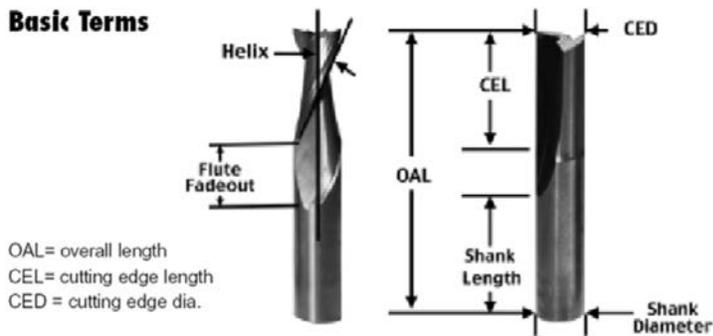
1. Straight flute: Offers a neutral cutting action - highest force.
2. Upcut flute: Provides the best surface finish and allows for good chip extraction. May cause part lifting if vacuum or fixturing is not sufficient.
3. Downcut flute: Provides a downward force which helps eliminate part lifting. Chip rewelding MAY occur if there is no space below the part for chip expansion.
4. Compression: Used for laminated materials, produces a good top and bottom finish on the part.

NUMBER OF FLUTES

1. Single Flute: Allows for larger chiploads in softer materials.
2. Double Flute: Allows for better part finish in harder materials.
3. Multiple Flutes: Allows for an even better part finish in harder materials.

Note: As the number of cutting edges increase, your feed rate should increase to prevent burning and premature tool dulling.

Basic Terms



OPTIMIZING SPEED AND FEEDS

1. Start off using the recommended chipload and RPM for the material you are cutting.
2. Increase the feedrate until the part finish starts to decrease or you risk moving the part off the vacuum. Decrease the feed by 10%.
3. Next decrease your RPM by a set increment until your surface finish deteriorates again. Once this happens increase your RPM until the finish is acceptable.
4. You have now optimized your speed and feed by taking the largest chip possible.

Note: This should be done in the first sheet of material to prevent tool dulling due to excessive heat.

TOOL HEAT

If a feed rate is too low heat will be generated causing the cutting edge to break down and dull quickly. To check this, run a nest of parts and stop the spindle. When the spindle has stopped rotating, carefully feel the tool's temperature. It should be at or near room temperature. If the tool is hot review "Optimizing Speed and Feeds".

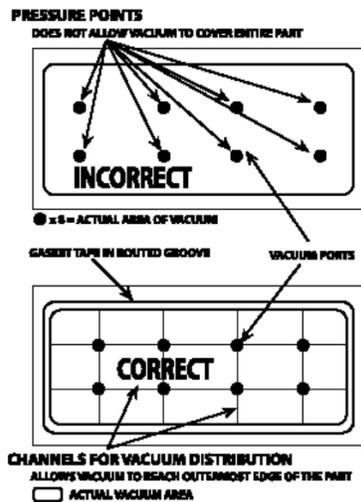
FIXTURING METHODS

FLOW THROUGH VACUUM

This style uses LDF (Low Density Fiberboard) or MDF (Medium Density Fiberboard) as a sacrificial surface for sheet material to be cut on. The porous nature of LDF or MDF allows vacuum to pass through allowing the material to be held in place for machining. As parts are cut out of the sheet material, vacuum loss starts to occur from the slot produced by the cutting tool. This can lead to part lifting or movement especially in small parts. Cutter diameter will also influence part movement. A 1/2 diameter tool will exert 25% more lateral pressure than a 3/8 diameter tool.

When cutting small parts in sheet material, one may want to consider tab or skin cutting to prevent part movement.

DEDICATED SPOILBOARD



Proper Spoilboard Techniques

Dedicated spoilboards are used for reoccurring production runs where optimal cycle times are needed. This work holding method creates vacuum chambers in the sacrificial board specifically to the shape of the parts being cut. This elimination of vacuum loss relates to improved cycle times and part finish.

STEPS TO CREATE A DEDICATED SPOILBOARD:

1. Surface both sides of your MDF board
2. Lay out the part pattern on the MDF and determine quantity that will fit.
3. Cut the part profile into the MDF board using a larger diameter tool than would normally cut the part. Make your slot depth 1 to 1.5 times the cutter diameter.
4. A gasket groove must be cut next inside the part profile to create a vacuum seal. The groove should be 1/2 the gasket material thickness to allow for proper compression.
5. A grid pattern must then be cut inside the gasket groove to distribute the vacuum evenly through out the vacuum area.
6. Drill holes through out the pattern in the intersections of the vacuum grid until there is no resistance on your vacuum gage on the machine table.
7. Seal the board using rubberized coatings, polyurethane sealers or a sanding sealer to prevent vacuum from passing through the board in unwanted areas.
8. Apply the gasket tape.

This operations sounds time consuming. It will be for your first board. Once you become familiar making these fixtures, you will make up for it in your cycle time reductions and part finish. A lot of headaches and problems can be resolved by using the proper work holding.

RAISED SPOILBOARD

This is generally used where secondary operations are needed and the spoilboard will interfere with the secondary tool. Raised spoilboards are another type of fixturing that works well for routing parts such as circles from squares where the scrap or off-fall is of such a size to be potentially harmful to the tool and or operator when it is cut free. A raised spoilboard should make sure the off-fall would not interfere with the first and second tool and that the off-fall would be free and clear of the tool path.

SURFACING SPOILBOARDS

When creating new fixtures or using a new MDF sheet, the spoilboard must be surfaced to level the board to the machine table. This consists using a large diameter cutter (OC 90-000 series) to quickly level the entire surface. The following benefits will be achieved by surfacing your spoilboard:

1. Leveling material to get consistent cuts.
2. Remove grooves caused by routing.
3. Reduce vacuum loss due to clogged pores at the material surface due to dust and chips.
4. Preventing material warpage caused by humidity in summer time.