GENERAL TURNING
Contents

Turning

A. General turning

Single-point machining ........................................... A5
Turning ................................................................. A5
Chip formation ...................................................... A6
Chipbreaking ........................................................ A6
Cutting data ........................................................ A8
Tool geometry ....................................................... A9
Insert shape and nose radius ................................. A10
Wiper technology ............................................... A11
Toolholders ........................................................ A12
CoroTurn RC ........................................................ A14
T-Max P clamping systems .................................. A16
CoroTurn 107 and 111 screw clamping system .... A17
Clamping systems for ceramic and CBN inserts .... A18

Internal turning .................................................... A19
Boring ................................................................. A19
Cutting forces in boring operations ..................... A19
Holding the boring bar ....................................... A21
Boring very deep holes ...................................... A23
Eliminating vibration tendency with tuned boring bars ........................................... A24
Recommended combinations of bars, sleeves and adaptors .......................... A25
Clamping with EasyFix sleeves for cylindrical bars .. A25
Coolant connection for boring bars ........................ A25

Selecting turning tools .......................................... A26
Selection factors for turning operations .............. A26
Application checklist and hints for internal turning ................................. A27
Selecting tools for turning .................................... A28
Before starting – look at the tooling alternatives .......... A29
Toolholder selection ............................................ A30
Toolholders for external turning .......................... A30
Tool selection for internal turning ......................... A31
Toolholders for external turning .......................... A32
Toolholders for internal turning ........................... A33
Tools for internal turning .................................... A34

Insert shape .......................................................... A36
Insert shape depending upon operation ................ A37
Insert shapes and sizes ...................................... A38
Insert size .......................................................... A40
Selecting the insert size ..................................... A41
Insert nose radius .............................................. A42
Wiper inserts ...................................................... A43
T-Max P Wiper inserts ...................................... A43
CoroTurn 107 Wiper inserts .............................. A43
Wiper insert geometry effect on workpiece dimensions ........................................... A44
CoroTurn 107 knife edge Wiper inserts .............. A45
Comparison of standard inserts versus Wiper inserts ................................. A46
General rules on achieving surface finish with Wiper inserts ................................. A46
High-feed turning .............................................. A47
Insert geometry .................................................... A48
Turning geometry description ............................. A49
Negative basic-shape inserts T-Max P ................. A50
Positive basic-shape inserts CoroTurn 107 ............ A58
Indexable insert selection factors ........................ A65
Insert grade ......................................................... A66
General turning ................................................ A67
Coated carbide .................................................. A73
Cermet ............................................................... A75
Uncoated carbide ............................................. A75
Ceramics ............................................................ A76
Cubic boron nitride ............................................ A77
Diamond .......................................................... A77
Cutting data ......................................................... A78
Cutting speed recommendations ........................ A80
General cutting data recommendations for cast-iron, hardened steels and HRSA (advanced tool materials) ................................. A84
Turning without coolant ..................................... A87
If problems should occur .................................. A88
B. Parting and grooving
Cutting off and making grooves ......................... B2
Machining factors ........................................ B3
Parting off – tool selection ................................ B4
Application factors ....................................... B5
Grooving ...................................................... B7
Circlip grooving .......................................... B9
Undercutting ............................................... B9
Face grooving .............................................. B9
Wider face grooves ....................................... B10
Turning and profiling .................................... B11
Selecting tools for parting and grooving ............. B13
Tooling alternatives for parting and grooving ........ B13
Tool selection for parting and grooving .............. B14
Tool system ............................................... B14
Selecting toolholder types ................................ B15
CoroCut family toolholders ................................ B17
Choosing the right holder for face grooving ........ B17
CoroCut SL – external and internal machining ...... B18
Recommendations when choosing CoroCut SL cutting blades ......................... B19
Cutting depth limitation for re-inforced CoroCut blades ........................................ B20
Parting and grooving geometries and grades ...... B21
The Wiper effect with CoroCut ......................... B22
Parting and grooving grades ........................... B23
Feed recommendations for parting and grooving geometries ........................................ B24
Cutting speed for parting and grooving tools ....... B32
Application hints for parting and grooving .......... B36
Tool wear indicators .................................... B38
Tailor Made ................................................. B39

C. Thread turning
Turning screw threads .................................... C1
The geometrical shape of the screw thread .......... C2
Selecting infeed type ..................................... C3
Radial infeed ............................................ C3
Modified flank infeed .................................. C3
Incremental infeed ....................................... C3
Optimizing tool-life and threading economy ....... C4
Infeed passes – number and sizes .................... C5
Infeed recommendations ............................... C5
Method of threading – right- and left hand threads C13
Inclining the insert for clearance ..................... C14
Selecting shims for inclination ....................... C15
When to use the U-Lock system ..................... C16
Toolholders for threading ............................. C17
Cutting speed for threading ........................... C18
Grades for T-Max U-Lock threading ................ C19
Selecting the insert geometry and grade .......... C20
Tailor Made ................................................. C21
Toolholders for threading and circlip grooving .... C22
Threading close to the centre ......................... C23
Thread turning application hints ..................... C23
If problems should occur ............................. C24
Infeed values recommendations ...................... C26
Turning

General turning

Single-point machining
Turning, basically, generates cylindrical forms with a single point tool and in most cases the tool is stationary with the workpiece rotating. In many respects it is the most straight-forward metal cutting method with relatively uncomplicated definitions. On the other hand, being the most widely used process and easily lending itself to development, turning today is a highly optimized process, requiring thorough appraisal of the various factors in applications.

In spite of generally being a single cutting-edge operation, the turning process is varied in that the workpiece shape and material, type of operation, conditions, requirements, costs, etc. determine a number of cutting tool factors. Today’s turning tool is carefully designed, based on decades of experience, research and development.

From the micro geometry and tool material at its point of engagement, to the basic shape and clamping of the indexable insert through to the toolholder - shank type or modular - the tool handles the dynamics of metal cutting today in a way which would have been unthinkable a couple of decades ago. Many of the principles that apply to single-point machining apply also to other metal cutting methods, such as boring and even multi-point, rotating tool machining such as milling. There are several basic types of turning operations, requiring specific types of tools for the operation to be performed in the most efficient way.

Turning...
... is the combination of two movements: rotation of the workpiece and feed movement of the tool. In some applications, the workpiece can be stationary with the tool revolving around it to make the cut, but...
basically the principle is the same. The feed movement of the tool can be along the axis of the workpiece, which means the diameter of the part will be turned down to a smaller size. Alternatively, the tool can be fed towards the centre (fac ing off), at the end of the part. Often feeds are combinations of these two directions, resulting in tapered or curved surfaces which today’s CNC-lathe control-units, with their many program possibilities, are able to more than cope with.

This part will deal mainly with external turning, leaving other, more specialized operations, such as threading, grooving, cutting off and boring to be discussed in separate application chapters.

Turning can be broken down into a number of basic cuts for selecting tool types, cutting data and also programming for a certain operation.

**Chip formation**

Turning has been developed to a stage where not only is metal removed at very high speeds but the parameters that make up the process are closely controlled as are the end-results in the form of component quality and reliability. Chips are sheared off a workpiece according to carefully defined data that ensures correct shape and size.

Although the machining process is intended for cutting a workpiece into a specific shape, size and finish, the chips produced most comply with the those acceptable for the application as well as being necessary because of the high cutting data involved in today’s CNC machines. Masses of long, stringy chips would otherwise accumulate in a very short time in the machining area in uncontrollable nests of tangled swarf. This is a threat to the process and one that can endanger not only operators but also the finished component. The chip form is to a large extent influenced by the materials being cut, varying between continuous forms of ductile material to crumbling, brittle material.

An orthogonal view of the chip formation is where the cutting speed direction or axis of rotation of the workpiece material being cut is at right angles to the main cutting edge. This is a simplified view of the cutting process, employed only in a few operations such as some facing and plunging. Most metal cutting is oblique, where the cutting direction is at a certain angle relative to the main edge. This changes the geometrical conditions considerably and the chip flow direction is altered. Basically, instead of a watch spring type chip, as in a typical parting operation, the chip will take various forms of comma or helical shapes.

The entering angle and nose radius of the tool affects the chip formation in that the chip cross-section changes. The...
Turning

A chip thickness is reduced and the width increased with a smaller angle. The direction of chip flow is also changed, usually advantageously, with the spiral pitch being increased. Depending upon the depth of cut (DOC = ap), the shape and direction of chips also vary with the nose radius on the cutting edge. When the DOC is small in relation to the nose radius, the radius part is the main part of the cutting edge and spiral chips will be generated. A larger depth leads to less influence from the radius and more from the actual entering angle of the edge with an outward directed spiral chip as the result. The feed rate, however, also affects the width of the chip cross-section and the chip flow.

A square chip cross-section usually means excessively hard chip compression while a wide, thin band-like chip is formed in unsuitably long strands. When the chip curve becomes smaller for a thicker chip, the chip/tool contact-length becomes longer with more deformation and pressure. Excessive thickness has a negative influence on the machining process. Furthermore, if the feed rate is increased to above what the insert geometry has been designed for, the chip will pass over the chip forming geometry, with the effect that machining is performed with a negative instead of positive geometry without balanced chipbreaking.

A finishing insert, working mostly with its nose radius will have the geometry concentrated to the corner of the insert while a heavy roughing insert will have geometry over the rake face.
Some inserts are capable of providing satisfactory chip formation across a broad intermediate range, having incorporated combinations of chipbreakers, ranging from the corner radius and across the insert.

**Chipbreaking**

As shown on the previous page, there are different ways for a chip to break, mainly:
- A. Self-breaking as with turning cast-iron
- B. Breaking against the tool
- C. Breaking against the workpiece

The type of chipbreaking encountered partly depends upon the insert and tool geometry and cutting data. Any of the chipbreaking ways may cause a disadvantage but this can usually be overcome by choice of geometry or cutting data. When self-breaking, if the tool-life is unacceptable, employ a more open chipbreaker as part of the insert geometry. When breaking against the clearance face of the tool, chip-hammering may be a disadvantage and a different (tighter or open chipbreaker) geometry may be better. Alternatively, adjust cutting data. When breaking a against the workpiece shoulder when large D O C are employed, unsatisfactory chip spraying may be encountered and a smaller entering angle should be considered.

Short chipping materials need little or no chipbreaking while some long chipping materials need chipbreakers designed into the insert geometry to deform and break the chip. The initial cutting of the chip is in most cases not sufficient to break the chip into required lengths. A chipbreaker in its simplest form is a built-in obstruction to the chip flow. This crude form has many disadvantages and has had, in many cases, a negative effect on machining performance.

Various forms of ground and later pressed indexable insert chipbreakers were developed before today’s modern inserts. The modern indexable insert is a complex combination of angles, flats and radii to optimize chip formation through cutting action, contact length, chip breaking, etc.

Most inserts have positive rake angles, combined with being inclined negatively in the toolholder, to promote good chip formation and positive cutting action. Negative primary lands of varying lengths depending upon the working area of the geometry, are applied to strengthen the cutting edge.

Chip control is, thus, one of the key factors especially in turning and drilling. Milling creates a natural chip length thanks to the limited length of cutting edge engagement. In drilling and boring, chip control is vital because of the limited space inside holes being machined. Also in modern high-performance drilling, chips have to be of exact form so as to be evacuated efficiently from the cutting zone - any congestion, quickly leads to tool breakdown.

The chipbreaking diagram for an insert geometry (based on the recommended ranges of feed and depth of cut), in combination with the tool material, is the key to insert application. The modern latest programme of inserts will include cutting geometries to cover most workpiece materials. These geometries will cover applications including finishing, semi-finishing to roughing as well as heavy duty rough machining.

Chip control is thus mainly exercised through the geometry of the indexable insert in combination with cutting data.
**Cutting data**

The workpiece rotates in the lathe, with a certain **spindle speed (n)**, at a certain number of revolutions per minute. In relation to the diameter of the workpiece, at the point it is being machined, this will give rise to a **cutting speed**, or **surface speed** (Vc) in m/min. This is the speed at which the cutting edge machines the surface of the workpiece and it is the speed at which the periphery of the cut diameter passes the cutting edge.

The cutting speed is only constant for as long as the spindle speed and/or part diameter remains the same. In a facing operation, where the tool is fed in towards the centre, the cutting speed will change progressively if the workpiece rotates at a fixed spindle speed. On most modern CNC-lathes, the spindle speed is increased as the tool moves in towards the centre. For some of the cut, this makes up for the decreasing diameter but for very small diameters, and very close to the centre, this compensation will be impractical as the speed range on machines is limited. Also if a workpiece, as is often the case, has different diameters or is tapered or curved, the cutting speed should be taken into account along the variations.

The **feed (fn)** in mm/rev is the movement of the tool in relation to the revolving workpiece. This is a key value in determining the quality of the surface being machined and for ensuring that the chip formation is within the scope of the tool geometry. This value influences, not only how thick the chip is, but also how the chip forms against the insert geometry.

The **cutting depth (ap)** in mm is the difference between un-cut and cut surface. It is half of the difference between the un-cut and cut diameter of the workpiece. The cutting depth is always measured at right angles to the feed direction of the tool.

The cutting edge approach to the workpiece is expressed through the **entering angle (κr)**. This is the angle between the cutting edge and the direction of feed and is an important angle in the basic selection of a turning tool for an operation. In addition to influencing the chip formation, it affects factors such as the direction of forces involved, the length of cutting edge engaged in cut, the way in which the cutting edge makes contact with the workpiece and the variation of cuts that can be taken with the tool in question. The entering angle usually varies between 45 to 95 degrees but for profiling operations, even larger entering angles are useful.

The entering angle can be selected for accessibility and to enable the tool to machine in several feed directions, giving versatility and reducing the number of tools needed. Alternatively it can be made to provide the cutting edge with a larger corner and can add cutting edge strength by distributing machining pressure along a greater length of the cutting edge. It can also give strength to the tool at entry and exit of cut and it can direct forces to provide stability during the cut.
Turning

Tool geometry

The cutting action is to a great extent determined by the tool geometry. The tool geometry is designed to cut various workpiece metals by forming chips in a smooth way, while also providing a strong cutting edge, and to break chips into manageable swarf. Many indexable inserts have combinations of chipbreaking functions to cope with light cuts at the corner and larger depths of cut along the cutting edge. Each insert geometry is developed to cover an application area made up of the recommended feed and cutting depth ranges.

At one end, an insert geometry for finishing will have an area comprising smaller feeds and depths while at the other end, a rough-turning geometry will have a range of large feed and depth values. An all-round insert geometry covers a large intermediate area for a large variation of operations. The finishing insert uses the geometry at the corner of the insert while the roughing one uses a relatively long part of the main cutting edge.

The various chipbreakers are made up of different sizes and combinations of angles, flats and radii. Straight turning and facing operation will remain around one set of values in the application diagram, while profiling operations, with varying cutting depths and feed rates will give rise to movement or several points throughout the insert application area, providing acceptable chipbreaking. Other factors that determine the choice of insert geometry are the occurrence of intermittent machining, vibration tendencies. The amount of machine power can also determine the choice of insert geometry.

There is a distinction in cutting edge geometry between negative and positive insert geometry. A negative insert has a wedge angle of 90 degrees seen in a cross-section of the basic shape of the cutting edge. A positive insert has an angle of less than 90 degrees. The negative insert has to be inclined negatively in the toolholder so as to provide a clearance angle tangential to the workpiece while the positive insert has this clearance built-in. The inclination angle ($\lambda$) is a measure of at what angle the insert is mounted in the toolholder.

When the insert is mounted in the toolholder, the insert geometry and inclination in the toolholder will determine the resulting cutting angle with which the cutting edge cuts. The rake angle ($\gamma$) is a measure of the edge in relation to the cut although it is often expressed through a flat insert. The rake angle of the insert itself is usually positive and varies along the cutting edge, from the nose radius along the straight cutting edge. A flat insert has a rake angle of zero degrees. The actual cutting function of the rake angle also varies along the face of the insert, back from the cutting edge, until the chipbreaking function takes over the chip formation.

Also the actual cutting edge of the insert is subject to various development. The micro-geometry of the cutting edge is critical as regards strength and tool wear development. Edge preparation along the transition between the edge face and the clearance face is in the form of a radius, chamfer or land and affects tool strength, power consumption, finishing ability of the tool, vibration tendency and chip formation.

The edge preparation are applied to inserts depending upon application and may vary from a ground sharp edge for finishing in light metal to a wide negative land for heavier operations in demanding materials. Edge rounding (ER) is the most frequently used preparation and may also be used in combination with the application of a land. The radii are measured in microns and applied with precision through a special process. The extent of edge rounding is also combined with the tool material and the coating process of indexable inserts.
Insert shape and nose radius
Looking at the tool from above, the insert has a basic shape and a radius on the corners. The insert shape varies considerably and the point angle can be as little as 35 degrees extending to 100 degrees with the addition of a round insert. In between these extremes are square, triangular and rhombic shapes with point angles of 55, 60 and 80 degrees. The range gives properties ranging from the largest for highest roughing strength to the most pointed giving best profiling accessibility.

With high edge strength through long cutting edge engagement of the larger point angles comes the tendency for vibrations in the machining process and high power requirement. With high edge accessibility during machining comes a weaker cutting edge. A balanced choice for the operation in question is needed.

The nose radius ($r_\varepsilon$) is a key factor in many turning operations and one that needs consideration as the right choice affects cutting edge strength to surface finish of the component. An insert is available in several nose radii where the smallest nose radius is theoretically zero but where 0.2 mm is more commonly the smallest. The largest is normally 2.4 mm, although the full range is not available for one and the same insert shape or size.

In rough turning, the nose radius can be as large as possible for strength, without giving rise to vibration tendencies. The feed rate of the tool is also affected by the nose radius or vice versa. A large nose radius provides a strong edge, capable and dependent upon high feeds for proper cutting edge engagement. The small nose radius means a weaker point but one capable of fine cuts.

![Nose Radius Diagram]

The nose radius of an insert is an important performance factor.
In turning operations, the surface finish generated will be directly influenced by the combination of nose radius and feed rate.

The surface generated by a single point tool is made up of how the nose radius moves along the workpiece surface. The theoretical maximum profile height is calculated through a simple formula, giving an indication of the values to be expected and which can be compared to the limits set out for the component to be made. Alternatively, starting out with a certain nose radius and a required profile height, a starting value for the feed rate can be calculated.

**Wiper technology**

- a new perspective on feed and surface finish in turning

Generating a good surface finish on turned components and becoming a demand for semi-finishing and even roughing operations. The Wiper indexable insert technology has provided turning with a new means to achieve improved production performance where the key is to being able to raise the feed rate. The generated surface finish and tolerance are affected by a combination of nose radius size, feed rate, machining stability, workpiece, tool clamping and machine condition.

The conventional relationship in turning is for the surface finish to be directly related to the tool feed and the size of the nose radius. A large feed will give shorter cutting times but poorer surface finish. A large nose radius will generate a better surface finish and provide more strength. But an excessively large nose radius can lead to vibration tendencies,
unsatisfactory chipbreaking and shorter tool-life because of insufficient cutting edge engagement. In practice, therefore, the size of the insert nose radius and the feed may be limited in an operation.

To upset this relationship – to achieve a better surface finish at a higher feed, the Wiper technology for the indexable insert nose-radius has been developed. Principally, this has been in the form of a care-

finish through the wiper effect. Similarly, the Wiper insert for turning has been developed to provide a high capability of generating a better surface finish. Or, put another way, is capable of generating the same finish at a much higher feed.

Conventionally, a turning operation can often be characterized as having a certain feed rate established for an operation. For instance, using the largest permissible 1.2 mm nose radius at a feed rate of 0.15 mm/min might generate a surface finish of Ra 1 micron on a low-alloy steel component. If the feed rate is doubled, the surface finish obtained will be in the region of Ra 2.5. When the feed is raised within the application area of the insert geometry, the surface finish often becomes unsatisfactory.

Changing the feed rate can also lead to chip control being unsatisfactory, the cutting edge load excessive and tool wear too rapid. Yet feed rate remains the most direct way to lower actual machining times but does not have the marked effect on tool-life that cutting speed does. The Wiper technology is a direct means with which to improve productivity.

The Wiper technology has provided finish turning with new possibilities.

### Guidelines

- Two times the feed rate = Same surface finish
- Same feed rate = Twice as good a surface finish

The Wiper insert has a completely new nose configuration.

The Wiper insert has a completely new type of nose configuration. In milling, there are specific wiper inserts which are mounted somewhat below the others in a facemill which, due to having a wider parallel land, smooths the surface to a better finish. The Wiper insert has an adapted chipbreaker while conventional nose-radius insert has a geometry that limits feed rates before that of a Wiper insert. The Wiper chipbreaking ability has been adapted to be in line with the modified nose radius and higher feed-capability. Chip control therefore extends over new areas beyond conventional cutting data limitations. This is one of the reasons why Wiper inserts are now also often seen as problem solvers in turning operations where unacceptable chip forms create problems.
This catalogue has been split into smaller parts to enhance downloading speeds.

If you want to view the next page please click HERE!