TOTEM High Speed Steel Taps

"TOTEM" high precision HSS Taps & Dies are manufactured from high quality imported High Speed Steels.
Strict quality control at all stages, rigid inspection with sophisticated equipment and proper heat treatment with modern furnaces under controlled conditions, ensure that **"TOTEM"** Taps & Dies have the same consistency in quality, for better performance and longer life.
" TOTEM" HSS Ground Thread Taps are available in the following types :

Short Machine (Hand) Taps

i) Serial form (Rougher, Intermediate or Finisher)
ii) Full form (Taper, Second or Bottoming)
iii) Spiral Pointed
iv) Spiral Fluted
v) Fluteless (Roll form)

Hand Taps

For hand taping, the conventional set of 3 taps with straight flutes is recommended. It is essential that the tap is presented squarely to the work and that the taps are correctly aligned. When these taps are used in a machine it is usually not necessary to use the taper tap.

Serial Taps

On the more difficult to machine material or on long blind holes it may be necessary to use either two or three serial taps. The tap thread is generally serialized by either truncating the full form thread or by grinding the full thread from progressively undersize.

Spiral Pointed Taps

"TOTEM" HSS Ground Thread Spiral Pointed Taps are designed for high speed production on through hole work. The threading is completed in one operation. Spiral Pointed Taps are made with a special leading flute ground at an angle to the tap axis. This left hand flute at the lead pushes the swarf ahead of the tap thread so permitting the use of small flutes. This particular feature eliminates clogging of flutes with swarf and facilitates smooth flow of coolant to the cutting point. Further, the flute area is substantially reduced, thereby increasing the strength of the tap. The result is therefore stronger taps which are suitable for through hole tapping on most materials. The speed of tapping operation and the number of holes threaded per tap increases.
These are made with a special leading flute ground at an angle to the tap axis. This left hand flute at the lead pushes the swarf ahead on the tap thread so permitting the use of small flutes. The result is therefore stronger taps which are suitable for through hole tapping on most materials. Blind hole tapping should only of the core hole for the accumulation of swarf.

Spiral Fluted Taps

Taps specially developed by TOTEM for use in blind holes, where the swarf is drawn out, preventing hole clogging - for perfect threads.These taps have continuous spiral flutes the same hand as the thread so forcing the swarf up the hole. Most suitable applications are in blind holes in materials producing long continuous chips.

Long Shank Machine Taps

'A' (Taper), 'B' (Spiral Pointed), 'C' (Bottoming) or 'D' (Second).
These taps have reduced shanks and may be used instead of a standard tap where greater thread length or extra reach is required.

Machine NIB Taps

Special designed taps, to give perfect thread in Nuts; used extensively in Nut manufacturing industry.

Pipe Taps

The nominal size is the size of the pipe fitting not the size of the tap and may be either paralled thread e.g. BSP-PI. NPS or taper thread - sometimes known as full thread taper taps.
e.g. BSPT, NPT. Only two taps and bottoming are normally used for full thread taper forms.
For use on a material which tend to load the tap e.g. galvanized pipe. it may be necessary to use a pipe tap with alternate threads removed (interrupted thread). This is only recommended where difficulty is experienced with standards taps.

(B) Unified, Whitworth and BA Taps

Class Tolerance Designation	Old Zone Tolerance Designation	Suitable for Producing
Class 1	Zone 1 & Zone 2 combine	Unified Class 3B nuts Whitworth Close Class nuts
Class 2	Zone 3	Unified Class 2B nuts Whitworth Medium Class nuts BA Normal Class nuts
Class 3	Zone 4	Unified Class 1B nuts Whitworth Normal Class nuts

2. ISO Pipe Thread Taps
There are three types of ISO component Pipe Threads as follows :
G Series Thread (BSPF Series threads) contained in BS 2779. This is a parallel fastening thread where pressure tight joints are not required.
Rc Series Threads (BSPT Series threads) contained in BS 21. This is a Taper pipe thread, both external and internal, where pressure tight joints are required.
Rp Series Threads (BSPPI Series threads) contained in BS 21. This is an assembly of a parallel internal thread with a tapered external thread when pressure tight joints are required. In all three pipe thread series, there is only one class of internal thread fit , and therefore there is only a required for one class of tap tolerance which is Class 2 in the case of the G and Rp parallel internal threads. (American pipe thread taps i.e. NPT, NPS etc. will be supplied with dimensions shows in BS 949 : 1951, BS 949 : 1969 and 94.7 : 1971.)

Tap Lead (Chamfer)
Straight flute taps, except taper thread pipe taps, can have the following standard chamfers :
Taper tap 4^0 per side (8-10 threads)
Second tap 8^0 per side (4-5 threads)
Bottoming tap 18^0 - 23^0 per side (1.1/2-2.1/2 threads)
Spiral flute and spiral point taps have only one chamfer angle approximating to bottom and second lead respectively.

Straight Flute Taps

TOTEM High Speed Steel Ground Thread Straight Flute Taps are produced with Taper, Second and Bottoming leads.

Taper (or first) Tap

The taper tap is tapered for 8/10 threads (4^{0} per side) and is used first, gradually cutting to the full thread.

Second (or intermediate Tap)

The second tap usually has 4 or 5 threads lead (8^{0} per side). This tap can finish a through hole.

Bottoming (or Plug) Tap

The bottoming tap has a 1.1/2-2.1/2 thread lead (18^{0} - 23^{0} per side) and is the finishing tap. When hand tapping, it is usually necessary to use all three taps; when machine tapping if is not often necessary to use the Taper (first) tap.

Tap Sets of 3

TOTEM High Speed Steel Ground Thread Straight Flute Taps to BS 949/ISO 529 are also available in sets of 3 per box. Each set contains 1 each Taper, Second and Bottoming leads of identical size.

Spiral Point Taps

Approximates to second lead only.

Spiral Flute Taps (35^{0} helix)

Approximately to 18^{0} Lead.

Spiral Flute Taps (15^{0} helix)

Approximately to intermediate lead 15^{0} angle.

Serial Taps to DIN 352, DIN 2181

On the more difficult to machine materials or on long blind holes it may be necessary to use either two or three serial taps. The taps thread is generally serialized by either truncating the full form threads or by grinding the full thread forms progressively undersized.

Identification

To help identification Serial Taps are marked as follows :

Rougher (First Tap)	- 1 ring around the shank
Intermediate (Second Tap)	- 2 rings around the shank
Finishing (Third Tap) -	- no rings

TAP CLASS AND NUT CLASSES

In BS 949 there are three tap classes for both imperial and metric sizes. There is an additional cut thread class.

In the respective nut standard BS 1580 (unified) each commonly used nut class has zero minimum clearance for pitch diameter, i.e. minimum size is basic. In the metric series there is also a G tolerance having positive minimum clearance above basic which gives a specified over size for plating purpose.

The relationship of tap size and tapped hole size is of a general nature since the accuracy of tapping depends upon external factors other than tap size.

Class 1 Tap

This is closest to basic, having little wear allowance, and is normally specified for 'close' fit threads, e.g. Unified 3B; Metric 4H, 5H.

Class 2 Tap

This is normally specified for 'medium' fit thread, e.g. Unified 2B Metric 6H; 4G, 5G.

Class 3 Tap

This is the furthest above basic and used for 'free' fit thread, e.g. Unified 1B; Metric 7H, 8H, 6G. All the above tap classes have a similar close manufacturing tolerance and vary only in their minimum size above basic.

Cut Thread Tap Class (Previously Zone 5)

This class has a much wider manufacturing tolerance and is suitable only for free fit threads.

Tap Drill Size
Use the correct tap drill size, ensure that hole is round and true. The tap drill sizes are arrived generally to give a thread depth of 75% or more of the tap diameter to short hole tapping.
For long thread lengths, fine pitches and small diameter a reduced amount of thread depth is sometimes desirable.
Tapping costs rise rapidly as the depth of thread increases. For deep hole tapping or average commercial work a thread depth of 60% - 70% will usually provide greater strength that of the mating screw.

Lubricant
First class tapping can only be done with a copious supply of proper lubricant. Use of lubricant is an important as its choice. Keep it clean and direct the flow into the hole being tapped. An ample supply is needed on the cutting edges, not only to disperse the heat, but in many cases to assist in the formation and disposal of the chips.

Re-sharpenning
A dull tap may produce undersize or oversize holes with poor finish. Normally smaller taps need regrinding on the chamfer only; larger taps may also need regrinding in the flutes. Regrind only on a Tool and Cutter grinder and duplicate the grinding (as near as possible) to that found on a new TOTEM tap of the same size and type.

Speeds
Efficient tapping has its optimum speed. See page 116 for recommended speeds (use these as a guide.) Exact figures cannot be given because of variable such as machine ability of material being tapped, condition of machine, depth, pitch and thread, holding fixture, etc.
Adjust our recommended speed up or down until optimum result are obtained.

Feeds
When starting a tap do not force or retard the tap, or a bell mouthed hole will be produced with 'Thin' thread. Allow the tap to find its own lead.
Do not rough out a tapped hole too near to its finished size, otherwise the finishing tap will not cut, but rub; inducing work hardening dulled taps and breakages. When tapping deep holes avoid the flutes becoming clogged with chips. Breakage is inevitable in such circumstances. Use one or more roughing taps and a finisher.
This problem of chip disposal is more prevalent in the tap size range 1/2" diameter and less, where flute space is restricted and chip disposal more difficult for this reason, spiral pointed taps are available in this range of sizes and are used for through hole tapping only. Size larger than 1/2" have sufficient flute space to cope with chip disposal.
NOTE : Spiral pointed taps are eminently suited for tapping short through holes.

Some Common Reasons For Tapping Faults & Failures

Tap Cuts Oversize

Tap out of alignment with the hole or tap not running true. Feed pressure incorrect producing thin or deformed thread.

Core hole too small.

Incorrect tap for the material e.g.

(a) Cutting rake too great

(b) Incorrect thread relief

(c) Chamfer lead too short

(d) Incorrect sharpening e.g. chamfer relief uneven or excessive.

Tap Cutting Edges Chip

Tap hitting bottom of a blind hole.

Tap reversed carelessly.

Lubrication lacking or of wrong quality.

Material too hard or abrasive for the type of tap.

Rapid Tap Wear

Core hole too small.

Lubrication lacking or wrong quality.

Material too hard or abrasive for the type of tap.

Speed too fast.

Poor Thread Finish

Core hole too small

Incorrect tap for the material e.g.

(a) Cutting rake too great

(b) Incorrect thread relief

(c) Chamfer lead too short

Lubrication lacking or of wrong quality.

Tap required resharpening.

Tap Breaks

Tap out of alignment with the hole or tap not running true.

Core hole too small.

Chamfer lead too short.

Incorrect sharpening e.g. chamfer relief un even or excessive.

Tap hitting bottom of a blind hole.

Tap reversed carelessly.

Lubrication lacking or of wrong quality.

Material too hard or abrasive for the type of tap.

Speed too fast.

Tap requires resharpening.

Tap flutes blocked with swarf.

Note on Tap Major Diameter

Except when a screwed connection has to be tight against gaseous or liquid pressure, it is undesirable for the mating thread to bear on the crests and roots.

By avoiding contact in this regions of the thread, the opposite flanks of the two thread are allowed to make proper load-bearing contact when the connection is tightened. In general practice, the desired clearance between crests and roots of mating thread is best obtained by increasing the major and minor diameter of the internal thread. Such an increase of the minor diameter is already provided on threads, such as the ISO metric thread, in which there is a basic clearance between the crest of minimum size nuts and the roots of maximum size bolts.

For this reason, coupled with the fact that taps are particularly susceptible to wear on the crests of their threads, a minimum size is specified for the major diameter of new taps which provides a reasonable margin for the wear of their crests and which at the same time should result in the desired clearance at major diameter of the hole.

In this connection it is necessary to emphasize that a tap should not be judge for size merely from measurement of its major diameter. Provided this measured size is not less tan the lower limit specified for the major diameter.

Material to be Tapped	Speed in Feet per Minute
Allegheny Metal	15-25
Aluminum	90-100
Brass	90-100
Bronze	40-60
Bronze-Manganese	35-60
Cppper	90-100
Die Castings :	
Aluminum	60-70
Zinc	60-70
Duralumin	90-100
Iron :	
Cast	70-80
Malleable	35-60
Monel Metal	20-25
Nickel Silver	75-85
Plastic Materials	60-70
Steel :	
Cast	20-30
Chrome	20-30
Mild	40-60
Manganese	10-15
Molybdenum	20-30
Nickel	25-35
Stainless	15-25
Tungsten	20-30
Vanadium	25-35

The above recommendation are made as a general guide. Speeds depend on the size of the tap, number of flutes, accuracy of fit and lubricants.

Terminology / Related Terms :

1. **Angle of Thread :** The angle included between the sides of the thread measured in an axial plane.

2. **Basic Size :** The theoretical or nominal standard size from which all variation are made.

3. **Lead / Chamfer :** The taper at the thread end of a tap or the throat of the die, made by cutting away the crests of the first few threads is to distribute the work of cutting, over several threads and act as guide in starting the tap or die. The chamfer is relieved to facilitate cutting, and the tap is classed Taper, Second or Bottoming, according to the length of chamfer, which approximate to the following.
   a. Taper Tap..................4° per side (10 to 12 Threads)
   b. Second tap.................8° per side (6 to 8 Threads)
   c. Bottoming taps..........22° per side (1-1/2 to 2-1/2 Threads)

   For dies
   a. Stamped side.............45°
   b. Back side...................60°

4. **Clearance Hole :** The holes in the die that provide the cutting rake and chip clearance.

5. **Cutting Edge :** The leading edge of the land in the direction of rotation for cutting and which does the actual cutting.

6. **Crest :** The top surface joining the two flanks of a thread.

7. **Depth of Thread :** The depth of thread, in profile, is the distance between the crest and the root of the thread measured normal to the axis.

8. **Effective Diameter :** On a parallel screw threads the diameter of an imaginary cylinder which would pass through the threads at such points as to make width of the thread at these points equal.

9. **Flute :** The grooves in the tap that provide the cutting rake and chip clearance.

10. **Helix Angle :** The angle made by helix of a thread at the pitch diameter with a plane perpendicular to the axis. The helix angle increases continuously from the crest to the root of the thread.

11. **Land :** That portion of the thread not cut away by the flutes or clearance holes.

12. **Land Width :** The chordal width of the land between the cutting edge and the heel measured normal to the cutting edge.

13. **Lead / Pitch :** The distance a screw thread advanced axially in one complete turn. On a single thread the lead and pitch are identical.

14. **Major Diameter :** The largest diameter on a parallel screw thread or tap. The term "Major Diameter"is replace the term "Outside Diameter" as applied to the thread of a screw and also the term "Full Diameter" as applied to the thread of a nut.

15. **Minor Diameter :** The smallest diameter on a parallel screw thread or a tap. The term "Minor Diameter" replace the term "Core Diameter" and "Root Diameter" as applied to the thread of a screw and also the term "Inside Diameter" as applied to the thread of a nut.

16. **Number of Threads / TPI :** The number of threads in a length of one inch.

Hints On Tapping :

The success of any tapping operation depends entirely on the use of the correct tap aided by the ideal tapping setup for the intended job.

There are several factors constituting an ideal set up for tapping operation. Basically CS Taps are hand taps and should not be used on machine (however customers do use these taps on machine). A few general recommendations for satisfactory tapping, most common causes of inaccuracy and their cures are enumerated below -

1. In cored holes, presence of hard spots, scale, sand can ruin a tap when the first thread itself is cut.

2. Use of dull drills or reamers can work - harden the surface of the job.

3. Multisided or oval holes give rise to uneven threads resulting in improper fit and is a common cause of breakage.

4. Use of worn out drill gives a smaller hole which increases the torque required for tapping resulting in breakage.

Recommendations :

1. Select the correct type of tap to suit the job requirements.

2. Select the correct size of tap drill. Traditionally a thread depth of 75% has been the rule. It should be born in mind that an increase in the thread depth affects the tapping torque while a decrease result in weak threads.

3. Hold the tap, if possible in a special purpose tapping attachment or chunk, ensuring that the tap will be presented to the hole squarely and that there is no relative eccentricity between the tap and the hole. It is usual to allow some float to the tap to take care of misalignment.

4. When the tap is to be used on a machine without lead screw feed, ensure that the tap will be able to advance into the hole freely. This ensures that it will cuts its own correct pitch.

5. Efficient tapping has its optimum speed. Always start with lower speeds and increase slowly if work permits.

6. First class tapping can only be done with a copious supply of proper lubricant. Use of lubricant is an important as it's choice. Choose suitable lubricant. Keep it clean and direct the flow into the hole being tapped. An ample supply is needed on the cutting edges, not only to disperse the heat, but in many cases to assist in the formation and disposal of chips.

7. The tapping operation is then ready to start. The final points to note are that the tap must be started in the hole smoothly. Do not try to hold it back, push it forward or make several attempts to start it, as this will lead to bell mouthing of the resultant tapped hole. After completing the tapping of the hole ensure that the flutes are cleared of any remaining swarf before commencing to tap the next hole.

8. When the tap is used by hand, the above points are of course still valid. An extra point is to note is that the tap should be turned smoothly.

9. Proper care, correct size and use of drills, with equipment in good condition, is very important in producing holes for tapping. Formula for obtaining correct tap drill size is given below. Select nearest commercial drill.

$$\text{Major Diameter of Tap in mm} - \frac{0.3299 \times \text{amount of percentage of full thread}}{\text{No. of thread per inch (TPI)}} = \text{Drilled Hole Size in mm.}$$

Traditionally 75% amount of percentage of full thread is rule.

Common questions related to Tap / Die geometry :

a. What is cutting and sizing portion of tap or die ?

A tap or die can be divided into two parts -
1. Part where we don't have full thread (lead), this part is called as cutting portion.
2. Part where we have full thread, this is called sizing portion.

b. Why in through hole tapping taper tap will give more life than bottom tap ?

As actual cutting takes place in the cutting portion of the tape only, taper tap has longest cutting portion and hence cutting load is distributed on longer area, hence more life. It has been found that bottoming tap gives 50 to 60% life of taper tap.

c. How do you know that life of tap or die is over ?

We come to know that life of the tap or die is over from following -
1. Go gauge becomes tight or does not enters at all as tap or die has worn out but the tap or die still cuts threads.
2. Tap or die becomes blunt - Cutting edges of the tap or die wear out and does not cut threads at all.
3. Tap or die breakage.

d. What is lead angle and how to select proper lead angle ?

Angle formed by cutting portion of the tap with axis of the tap is called as lead angle. Lesser the lead angle more the lead length and more life, better thread finish.

Hints On Use Of Circular Dies :

1. Always ensure the die is presented to the bolt squarely, and is concentric. if this is not correct the die will not cut evenly on all lands.

2. When adjusting a split die, try to avoid opening it out excessively, as it tend to rubbing or breakage.

3. When closing in, do not apply all the pressure on one position but keep it even. Dies will cut satisfactorily provided adjustment is kept in the range of approximately 0.4 mm.

4. Thread tearing and teeth chipping are due to incorrect sharpening of lands or due to causing the die to hit the shoulder.

5. Before starting a die always chamfer the end of the bolt at around 45° to allow the die to start cutting gently, failure to do this throws a sudden jerk on to the cutting edge, causing chipping.

6. The size of the bolt to be threaded should be basic size, as an oversize material will put an extra load on the die whilst cutting.

7. Whenever dies are used on machine it is recommended to use ground thread dies.

8 . When the dies is to be used on machine without lead screw feed, ensure that the die will be able to advance into the hole freely. This ensures that it will cuts its own correct pitch.

9. Efficient threading using die has it's optimum speed. Always start with lower speeds and increase slowly if work permits.

10. First class threading can only be done with copious supply of proper lubricant. Use of lubricant is as important as it's choice. Choose suitable lubricant. Keep it clean and direct the flow into the bolt being threaded. An ample supply is needed on the cutting edges, not only to disperse the heat, but in many causes to assist in the formation and disposal of chips.

11. When the die is used by hand, the above points are of course still valid. An extra points is to note is that the tap should be turned smoothly.

12. In order to achieve good results, the rake angle must be suitable for the material to be threaded. Long chipping materials, materials having more tensile strength requires more rake angle. Short chipping materials, ductile materials, materials having lesser tensile strength need lesser rake angle. Our standard dies are designed for general purpose, steel application with 8 to 12^0 rake angle.

Material specific dies :

1. We have developed dies for ductile material application like brass etc. with lower rake angle of 0 to 4^{o}. Land width of these dies kept on lower side as machining of these material is very easy and higher land width unnecessarily increases friction. Generally these dies are designated as " S " dies or BRASS dies. e.g. 1/2 BPS(S) or 20 x 1.5 (Brass).

2. We also manufacture dies for stainless steel application with HSS M 35 steel. M 35 steel contain Cobalt which increases wear resistance, hardness, strength and higher temp. stability. Generally these dies are designated as " SS " dies or " M 35 " dies. e.g. 12 x 1.75 (SS) or 3/32 BSW (M35).

3. We have observed that some customer needs threads up to next of the component. Hence customer grinds off the surface of the dies to reduce relived portion. This is a wrong practice as uneven grinding of lands to uneven cutting. Looking at this requirement we have developed special dies with lesser relieving. These dies gives up to neck threading. Generally these dies are designated as "J" dies. e.g. 3/16 BSF(J).

General Complaints of Taps And Dies :

1. Threads chipped off.
2. Threads worn out / got blunt.
3. Broken.
4. Less life.
5. Gauging problem / Incorrect geometry.
6. Cutting in reverse.

Some Of The Probable Causes For Such Complaints Are Listed Below -

1. Cutting torque is exceeding the torsional strength of the tap or die i.e. due to excessive torque.
2. The Tap is hitting the bottom of the hole for blind holes.
3. Thread depth is more than the thread length.
4. Used on hard material.
5. Used on machine at high RPM.
6. Used without coolant where necessary.
7. Chip clogging.
8. Used in punched hole.
9. Jerk is given while reversing.
10. Improper machine alignment.
11. Improper clamping of the tool.
12. Incorrect drill used / job diameter is not correct.
13. Too much axial load while threading.
14. Cold welding on flank.
15. Incorrect positioning / fixturing of tap or die.
16. Incorrect lead selected.
17. Uncalibrated gauges / instruments used for thread gauging / inspection.

Tap Breakage :

1. Blind hole : Below 4 mm, if tapping depth is more than 3/4th thread length, tap may break by chip clogging or tap may break by hitting bottom.

2. Deep hole tapping : If tapping depth is more than thread length, tap may break by chip clogging.

3. Tap may break during tapping, if cutting load is going beyond torsional strength of the tap i.e. due to excessive torque.

4. In punched hole the diameter of the hole reduces from the top to bottom hence material tend to grip the tap resulting in breakage of tap.

5. The tap may break while reversing if jerk is given while reversing.

6. It is recommended to use proper coolant while tapping as use of coolant reduces friction during tapping and reduces heat generation during tapping which subsequently reduces the property of welding built up edge formation.

7. Basically CS taps are hand taps and should not be used on machine at high RPM which may cause breakage.

8. Improper drill size - only recommended drills should be used.

Tungsten Carbide Rotary Burrs

1.) When a cutter is not in use, it should be stored on a wooden-stand. It is essential to protect the cutter teeth from impact with other tools during storage.

2.) A cutter should never be allowed to lock, or wedge in the work-piece profiles or cavities since this will almost certainly result in shattered teeth and/or a broken shank. Select the proper shape of cutter for the job.

3.) Insert the shank well down in the collet, but not so far as to engage the shoulder below the head. If over hang is necessary, it is essential that at least one half of the shank is securely gripped in the collet.

4.) Check that the collet is of the correct size and not worn or eccentric. The cutter must run true. Rotational eccentricity produces a type of hammering that will affect finish of the work and jeopardize the life of the teeth and shank.

5.) Keep the tool moving all the time; apply it firmly to fill the teeth with metal; do not ‘tickle’. Remove high spots first and then traverse the tool on the work.

6.) The work - piece should be securely held either in a vice, or a jig, so that it cannot move while cutting.

7.) Use each cutter at its recommended speed and ensure that the driving tool has enough power.

8.) If the teeth of a cutter tend to get clogged(i.e. When cutting aluminum, brass, etc.), They should be cleared periodically by cutting into a spare piece of soft cast iron. Also the occasional application of tallow is helpful.

9.) Face shields to protect the full face, neck, ears, etc. Should be used in preference to eye gaggles. Use of hand gloves is also recommended.

10.) Do not try to work with over-worn cutters since this will only damage them and produce poor work.

SPEED CHART - Approx. r.p.m. In 1000s

Material / Head Dia.	3 mm	4 mm	6 mm	8 mm	10 mm	13 mm	16 mm
Alum. Zinc base plastics	20-90	20-90	10-60	10-50	10-50	10-40	10-30
Cast Iron, Brass, Copper	40-90	40-90	20-50	20-50	15-40	10-30	10-25
Glass Fibre	40-60	30-50	20-30	15-20	15-20	10-15	8-10
Mild Steel, Bronze	60-90	50-80	30-45	25-40	20-30	15-20	10-15
Nimonic, Titanium, Stainless Hardened Steels, Ceramics	50-90	40-80	30-60	25-50	20-40	15-30	10-20