

## 迴轉沖削孔加工使用說明

**ADONAI** 沖削刀具是一個操作簡單的轉削孔刀具，可在盲孔或通孔上，銳邊或倒角邊製作有規則或不規則的多邊形，除了標準的六角形與正方形孔，也可製作溝槽，星形或其他特殊外形。



**ADONAI** 沖削刀具適用於一般車床、CNC自動車床、綜合加工機、鑽床以及特殊功能機器。也適用各種標準或特殊內外孔沖孔。

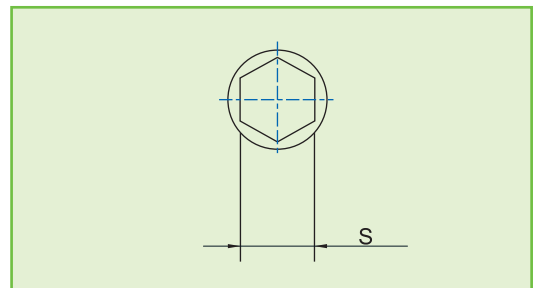
在正確的配合下，這個特殊角度  $\theta$  的結構可以節省80%的機械推力(如圖一)。這個重要的加工力量縮減是依賴特殊角度  $\theta$  的傾斜使機械推力集中於外形區域的一小部份去沖孔。外形的每一部份被沖刀的先端在同步迴轉中連續加工。輪廓精密度取決於沖刀的精密度，但也與正確的中心調整有關，如此也可增進刀刀的壽命。

在大部份轉塔車床的案例，夾頭本體是靜止的而組合在一起的沖刀卻可正、逆旋轉。在一些多軸或專用機械，情況剛好相反，夾頭本體在旋轉而沖刀在工件擋住而靜止。當輪廓必須被定位到工件的形狀，此時需要一個在主軸鼻端的連接器將沖刀靜止在正確的角度位置上。

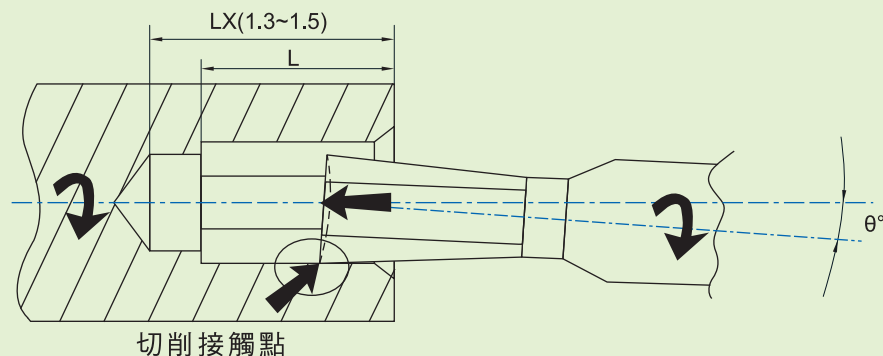
### 一、沖削孔加工之前的鑽孔與搪孔

例如沖六角孔其正常的鑽孔必須稍大於被沖孔兩相對平面的有效尺寸。在中碳鋼(60-80dN/mm)我們建議以下的容許公差值：

六邊形對邊尺寸S	鑽孔的容許公差值
1.5~3mm	S + 0.03~ + 0.06
3~6mm	S + 0.04~ + 0.08
6~10mm	S + 0.06~ + 0.12
10~16mm	S + 0.10~ + 0.20
> 16mm	S + 0.15~ + 0.30



(圖1) 工件



### 二、鑽孔的深度

通常我們建議鑽孔的深度是沖孔深度的1.3至1.5倍，(如圖一)碎片必須被沖刀充分的推擠而清除。如果工件必須熱處理，在這個狀況可能需要利用小於當初鑽孔0.1~0.2mm的鑽頭去鑽掉碎片。鑽孔的偏心最大不可超過沖孔尺寸的0.02~0.04mm。

### 三、沖削孔前注意事項

1、一個60度或90度倒角，倒角稍大於沖刀兩對角尺寸，這是一般沖孔的原則。如果在高迴轉速度下加工，則可能需要在沖孔後再修補倒角。

2、當需要高同心度時，預鑽一個0.5mm深的孔，直徑相同或稍小於沖刀兩對角尺寸，可提供解決的方案，這個預留的孔當開始沖孔時能撐住沖刀防止偏擺。加工完成後可將這個預留孔切削掉或倒角。

### 四、沖孔加工的迴轉速度

迴轉沖刀的原理允許應用於高轉速，從1500到3000轉/分鐘，因為迴轉速度不影響切削速度。然而，在非常高速下，當開始加工時沖刀會與工件摩擦，直到速度同步為止。這樣在時常使用小沖刀時非常容易提早磨損沖刀，為了避免這種現象，剛開始時用較慢的轉速或甚至不旋轉，當沖刀進入工作狀態在增加迴轉速度。

### 五、進刀

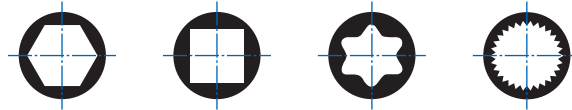
進刀量的選擇是根據材料的特性，在中碳鋼，如果機械有足夠的推力，我們建議每一迴轉進給率0.03~0.06mm。在銅、鋁材料能夠用到兩倍甚至三倍的進刀量。在大的輪軸尺寸和難切削材料加工，必須減少進給率0.01mm。

### 六、切削液

使用一般冷卻液或切削油。

## ROTARY BROACHING TOOLS OPERATING MANUAL

The **ADONAI** broaching tools is a simple, easy to operate attachment, which produces regular or irregular polygons in blind or through holes having sharp or chamfered edges. The attachment operates with a hunting rotary motion which, in addition to the standard hexagonal and square holes, can also produce grooves, Torx and other special profiles.

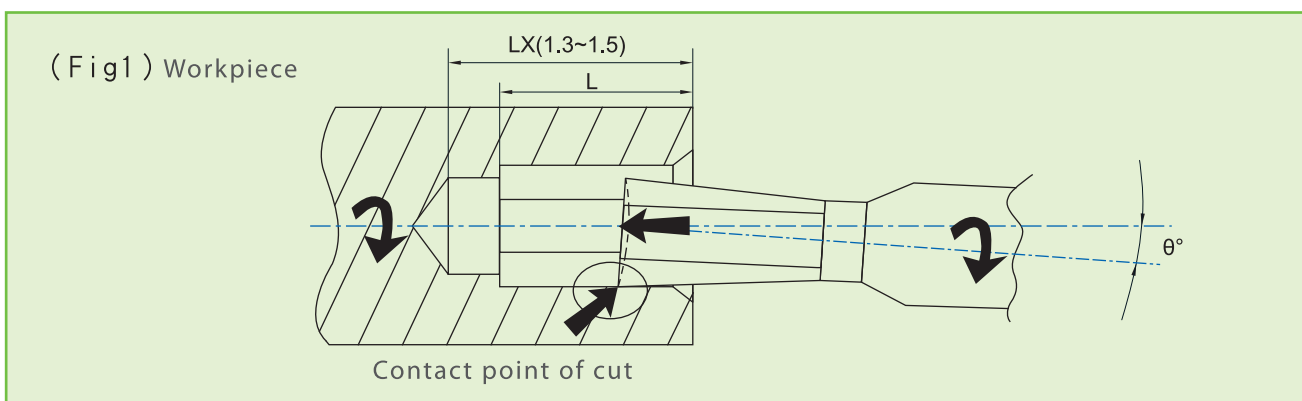
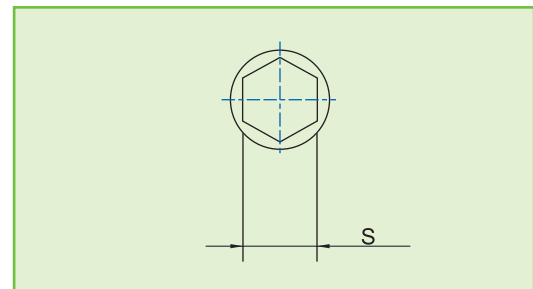


The **ADONAI** broaching tools can also be used on lathes, CNC automatic lathes, machining centers, drilling machines and special purpose machines for a wide variety of standard or special internal and external forms.

### 1.

Drilling The Hole - For internal broaching, the hole should be drilled approximately 1% bigger than the diameter across the flats of a hex shape. This percentage can be reduced in free cutting material and increased as machinability decreases, In mild steel we recommend the following tolerances:

Hex. Size S	Drill Hole Tolerance
1.5~3 mm	S + 0.03~ + 0.06
3~6 mm	S + 0.04~ + 0.08
6~10 mm	S + 0.06~ + 0.12
10~16 mm	S + 0.10~ + 0.20
> 16 mm	S + 0.15~ + 0.30



### 2.

Drill the hole as deep as possible to leave room for chip accumulation. A depth of 1.3 to 1.5 times the length of the profile is recommended. (Fig.1) If the chips need to be removed, re-drill the hole with a slightly smaller drill size.

3.

Preparing To Broach A Hole - The hole should first be prepared with a 60°-90° chamfer slightly bigger than the largest dimension of the broach tool. This ensures easy starting of the broach by guiding the tool in.

4.

Recommended Speed - The basic principle of the  $\theta^\circ$  offset allows high speed application from 1500 up to 3000 rpm. Surface footage has very little effect on cutting action or the tool life. The cutting edges of the broach tend to dig in to the face of the part as the tool comes in to contact with the rotating material. At higher speeds this dig mark will become more pronounced and tool life will suffer. For best results start the broach operation at a slow rotation and then increase the speed when it is in full contact. Reversing the spindle rotation half way into the part can reduce spiraling.

5.

Feed Rate - The feed choice mainly depends on the material characteristics. In a mild-steel, we recommend. 0.03~0.06mm per revolution. If the machine thrust force is sufficient, the feed can be doubled even.

6.

Coolant - Water soluble coolant or cutting oil are both sufficient for Rotary Broaching.