



## Speeds & Feeds

**Product Table:** Engraving Cutters - Tip Radius - 2 Flute - For Hardened Steels  
**Characteristics:** 2 Flutes

Series or Item	RPM	Chip Load (IPT) by Material		Axial DOC
		Hardened Steels		
		45 < 55 Rc	56 < 68 Rc	
7247xx-C6	6000+	.00022	.00017	< .008
7249xx-C6	6000+	.00020	.00016	< .009
7566xx-C6	6000+	.00031	.00025	< .010
7624xx-C6	6000+	.00043	.00035	< .010
7629xx-C6	6000+	.00038	.00031	< .010
7633xx-C6	6000+	.00022	.00017	< .010
8208xx-C6	6000+	.00032	.00025	< .010
8209xx-C6	6000+	.00029	.00023	< .010
8210xx-C6	6000+	.00020	.00016	< .010
8211xx-C6	6000+	.00032	.00025	< .010
8212xx-C6	6000+	.00020	.00016	< .010
8255xx-C6	6000+	.00024	.00019	< .010
8258xx-C6	6000+	.00027	.00022	< .010
8257xx-C6	6000+	.00031	.00024	< .010
8259xx-C6	6000+	.00037	.00030	< .010
8371xx-C6	6000+	.00040	.00032	< .010
8375xx-C6	6000+	.00026	.00020	< .010
8381xx-C6	6000+	.00034	.00027	< .010
8437xx-C6	6000+	.00025	.00020	< .010
8458xx-C6	6000+	.00036	.00029	< .010
8490xx-C6	6000+	.00036	.00029	< .010
8512xx-C6	6000+	.00025	.00020	< .010
8531xx-C6	6000+	.00035	.00028	< .010
8575xx-C6	6000+	.00022	.00017	< .010
8583xx-C6	6000+	.00022	.00018	< .010
8593xx-C6	6000+	.00029	.00023	< .010
8600xx-C6	6000+	.00032	.00026	< .010
8694xx-C6	6000+	.00040	.00032	< .010
8733xx-C6	6000+	.00019	.00015	< .010
8773xx-C6	6000+	.00036	.00029	< .010

**Please note:**

All posted speed and feed parameters are suggested starting values that may be increased given optimal setup conditions (minimal runout is required for best results).

Suggested speed is 6000 rpm or more. Choose an rpm value that creates the least amount of internal machine vibration. In many cases, a speed increaser is helpful.  
 Posted chip loads reflect axial depths of cut up to .009. For depths of cut = .010" -.015", reduce posted chip loads by 20%. For depths of cut = .016" -.020", reduce posted chip loads by 30%.  
 Posted chip loads reflect uncoated cutters. Coating is better suited to prolong tool life rather than decrease cycle times.  
 Posted chip loads reflect HORIZONTAL milling conditions. For VERTICAL plunge milling to depth, reduce posted chip loads by 70% (Do not plunge more than .009" depth, ramping is preferred to maintain tip integrity).

If you require additional information, Harvey Tool has a team of technical experts available to assist you through even the most challenging applications. Please contact us at **800-645-5609** or [Harveytech@harveyperformance.com](mailto:Harveytech@harveyperformance.com).

**WARNING:** Cutting tools may shatter under improper use. Government regulations require use of safety glasses and other appropriate safety equipment in the vicinity of use.