Name: _____ **Class:** _____ **Date:** _____ **ID: A**

Study Guide - Machining Parameters

Multiple Choice
Identify the choice that best completes the statement or answers the question.

____ 1. In optimizing machine efficienty, one of the primary trade-ofs is tool life Vs. _____.
a. Worker pay
b. Overhead
c. Machining time
d. Machine life

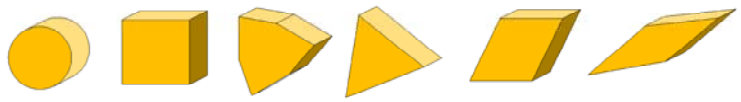
____ 2. As a rule of thumb, harder mateials are _____ to cut than softer ones.
a. Easier
b. More difficult
c. NA
d. NA

____ 3. Of the materials listed, which of the following is the oldest tooling material?
a. Carbon Steel
b. HSS
c. A2 Tool Steel
d. Carbides

____ 4. Because it can retain hardness at temperatures nearing 1,000°F, this type of steel was one of the first replacements for carbon steel in many cutting applications.
a. Carbon steel
b. HSS
c. Pearlite
d. Cementite

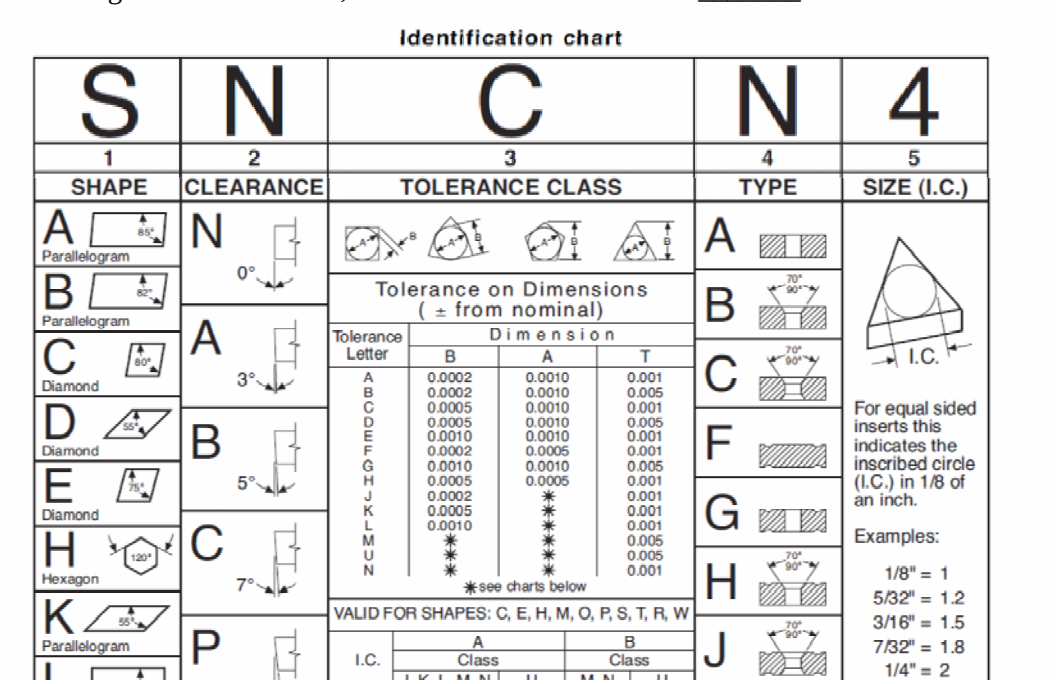
____ 5. Which of the following is the hardest tooling material?
a. Coated carbides
b. Ceramic
c. Polycrystalline diamond
d. HSS

____ 6. These insert shapes are arranged in order of _____ from left to right. The leftmost shape being the most _____.



a. Strength, strong
b. Versatility, versatile
c. Wavelength, radiant
d. Potential energy, powerful

____ 7. According to the chart below, a CNMG432ERP3 would be _____.



a. Hexagonal
b. Rectangular
c. Diamond Shaped
d. Round

____ 8. The three basic machining parameters are speed, feed, and _____.
a. Machine type
b. Tooling material
c. Depth of cut
d. Time

____ 9. This parameter has the least effect on tool life, so it should be maximized.
a. Depth of cut
b. Speed
c. Feed rate
d. RPM

Problem

10. Given the following parameters, use the formula $N=12V/\pi D$ to find the proper RPM.

Cutting Parameters:
Cutting Speed: 500 FPM
Depth of Cut: .040
Diameter: 1.25 Inches
Feed Rate: .005 IPR

(extra information provided at no additional cost)

Name: _____ **ID: A**

11. Given the following parameters, use the formula N=12V/πD to find the proper RPM.

    Cutting Parameters:
    Cutting Speed: 600 FPM
    Depth of Cut: .020
    Diameter: 3.22 Inches
    Feed Rate: .015 IPR

    (extra information provided at no additional cost)

12. Using the equation V = πDN/12 and the following parameters, find the cutting speed.

    Cutting Parameters:
    Spindle Speed: 600 RPM
    Depth of Cut: .020
    Diameter: 3.22 Inches
    Feed Rate: .015 IPR

13. If you are using 6061-T6 (ST & A) aluminum with a High Speed Steel tool, what would be the cutting speed?

Table 8. Cutting Feeds and Speeds for Turning Light Metals

Material Description	Material Condition	Tool Material: HSS Speed (fpm)	Uncoated Carbide (Tough) f = feed (0.001 in./rev), s = speed (ft/min) Opt.		Uncoated Carbide (Tough) Avg.		Polycrystalline Diamond Opt.		Polycrystalline Diamond Avg.	
All wrought and cast magnesium alloys	A, CD, ST, and A	800								
All wrought aluminum alloys, including 6061-T651, 5000, 6000, and 7000 series	CD	600	f	36	17					
	ST and A	500	s	2820	4570					
All aluminum sand and permanent mold casting alloys	AC	750								
	ST and A	600								
Aluminum Die-Casting Alloys										
Alloys 308.0 and 319.0	—	—	f	36	17		11		8	
			s	865	1280		5890[a]		8270	
Alloys 390.0 and 392.0	AC	80	f	24	11		8		4	
	ST and A	60	s	2010	2760		4765		5755	
Alloy 413	—	—	f	32	15		10		5	
	ST and A	100	s	430	720		5085		6570	
All other aluminum die-casting alloys including alloys 360.0 and 380.0	AC	125	f	36	17		11		6	
			s	630	1060		7560		9930	