Machining Guides for *DIA*bide™ Cutting Tools

Graphite

Machining graphite with diamond tools

CVD diamond tools are a perfect match for machining the graphite moldforms for EDM. The abrasive nature of EDM graphite grades severely limit the life of carbide tools, and PCD diamond tools are not available in the configurations required for detailed moldmaking.

Tools with diamond on the surface wear longer and have a lower coefficient of friction. These characteristics provide substantial benefit to the machining operation.

Because diamond tools last much longer —10 to 50 times the life of carbide—they:

– Improve the dimensional accuracy and consistency of the machined parts

– Greatly reduce the number of tool changes, increasing productivity

– Increase machine utilzation

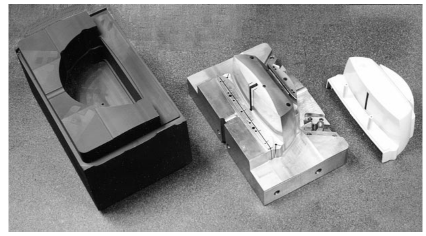
– Allow much longer periods of unattended machining, e.g., overnight

– Quickly pay for themselves

The low friction of CVD diamond tools permits using speeds 2 to 3 times higher than carbide—again contributing to productivity—with no degradation of surface quality. The consistently sharp edge and lower friction allows delicate, thin-wall sections to be machined quickly and precisely at high rpm settings with reduced feed rates.

Examples of graphite machining applications

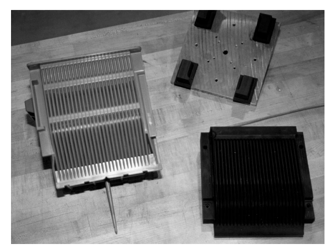
A plastic injection molding company serving the electronics and medical device industries had a problem making the mold for a component with a complicated contoured parting line. The double cavity mold involved four parting line burns, two for the cavity and two for the core. Each burn required four electrodes. The large electrode size required use of EDM 200 graphite to obtain 9" x 9" x 14" blocks. Using carbide ball endmills at 40 in / min required a 13-hour tool path. Cutters wore out before the end of one cut, necessitating three tool passes with a new cutter for each pass for a total machining time of 39 hours. The required accuracy of ±0.001" was never achieved.



Electrode, mold, and product.

Switching to an *sp³* DIAbide 3/8" 2-flute, ball nose Dapra-style insert and cutting at the same speed, each electrode was cut in one pass with no tool changes, saving 26 hours of shop time per electrode. With four electrodes per burn times four burns, 416 hours of shop time was saved in building this mold. In addition, the required electrode tolerance of ±0.001" was attained.

A leading supplier of molded plastic products for the microelectronics industry uses *sp³* endmills to machine the sinker EDM electrodes used to make injection molds. The electrodes are made from various grades of graphite which include EDM 200, Poco 2 and Poco 3. Previously, using uncoated carbide endmills at spindle speeds from 12,000 to 14,000 rpm and feed rates between 120 and 160 ipm, tool life averaged 4 hours. With *sp³* DIAbide 4-flute ball and square endmills in diameters from 1/8" to 3/8", operated at the same parameters, tool life is 60 hours—an improvement of 15 times over carbide. This allows long-term unattended machining, yielding further savings in machining costs.



Typical electrodes and a finished plastic wafer carrier.

Besides increasing productivity through fewer tool changes, the low wear rate of CVD diamond endmills has made it easier to obtain the required electrode accuracy of ±0.0005 in. because the CNC machinist doesn't have to fight tool wear.

General tool conditions for machining graphite

The graphite cutting process

In any machining process, a wear-resistant cutting edge separates material from the workpiece because of the velocity of the cutting tool edge relative to the workpiece. When cutting metals, intense heat causes plastic deformation, producing chips or a curl of material.

Graphite is different; it is a polycrystalline structure, and is machined by a process of fracturing this structure rather than a plastic deformation. The cutting edge *crushes* the graphite just ahead of the tool edge as it moves through the material. This forms small particles which result in graphite powder as an end product rather than chips or a curl, as when cutting metal. The graphite cutting process does not generate elevated temperatures.

Attention must be paid to the avoidance of chipping at the edge of a workpiece and development of internal cracking caused by compressive stresses. High quality, fine grain types of graphite should be used whenever possible due to the superior machining and handling characteristics.

Material abrasion

When cutting graphite, most tool wear is caused by the abrasive nature of the graphite particles rather than by material temperature or cutting speed. This places emphasis on selecting the most abrasion-resistant tool surface material, specifically, diamond.

Cutting speed

When cutting metals, the intense heat that is generated results in tool wear increasing rapidly with increase in cutting speed. The absence of elevated temperatures when machining graphite essentially eliminates speed as a contributor to tool wear.

Feed and depth of cut

Because small feeds and depths of cut do not lead to increasing the crushing effect in machining graphite, tool wear will advance rapidly with light feed, but stabilize as feed is increased. As the cutting action moves toward producing larger cracks and particles of removed material, flank wear of the cutting tool edge stabilizes. Therefore, in addition to increasing the volume of material removed, increasing feed can extend tool life.

The depth of cut should not exceed one-half of an insert's leg length or one-third of an endmill's diameter. This precaution will minimize material breakage at the exit of a cut.

Determining tool life

Tool life is determined by the quality of the cutting edge and the thickness of the diamond layer at the cutting edge. Typically the tool will go through a break-in period that serves to refine the tool surface at the cutting edge, resulting in an improved surface finish. This will be followed by a long period of consistent performance, with a very gradual thinning of the diamond due to workpiece abrasion.

End-of-life occurs when the diamond finally wears through, revealing the tungsten carbide substrate, or when the diamond surface develops a discontinuity. In either event the tungsten carbide wears rapidly at the cutting edge and the tool quickly becomes dull, resulting in poor surface finish of the workpiece.

Care and handling of diamond tools

Use the same care practiced with carbide tools when measuring or setting up. This care should include optical measurements when possible, and the avoidance of any contact of the tool's cutting edges with the hard surfaces of micrometer anvils, gauge blocks and any other inspection or setup equipment. Machine setup, including “touching off”, must be accomplished with a soft material (plastic, paper, etc.) between the tool and the touch off plate.

When storing *sp*^{3} tools, place them in the original containers provided. This will prevent the tools from touching each other or other hard surfaces in the storage area, or from rolling off the bench, which could result in chipping of the cutting edges.

Controlling dust and chips

The residue from machining graphite will range from small particles (0.001" – 0.005") to fine dust. It is important to install a quality system for control of graphite residue. A dust collection system employing high velocity air is commonly used for graphite. There are two aspects to be considered: initially capturing the dust and then suspension of the dust via the exit ducts. Depending upon the application, vacuum and compressed air, or a combination of both methods, are effective.

A minimum air velocity of 500 ft / min is needed to capture the dust at the machining location. After capturing the dust an even higher velocity of 2000 ft / min should be used to prevent dust from settling in the exhaust ducts. Screening should be used in advance of the dust filters to catch large particles which might damage the filter system.

Fluids can be used as a collection method for graphite particles. While a fluid is effective in trapping the graphite dust, filtering and re-use of the fluid is usually an expensive process which requires constant attention.

Clamping, fixturing and supports

Conventional clamping and / or fixturing systems can be used to secure graphite workpieces. The primary precaution is to avoid excessive clamping pressure, especially with lower strength materials. Just as with metals (aluminums, coppers, etc.) graphite can take on permanent warpage and deformation, especially with thin walls, large thin surfaces, and rods with a high length / diameter ratio.

Clamping systems from standard toe and strap clamps, through bolts, locator pins, and special pocketed fixtures, are commonly used. Hydraulic clamping systems must include line restrictors and air control valves to avoid clamps chipping the graphite parts.

Coolants

Generally, cutting fluids and coolants are not recommended for machining graphite. In some cases, for example, centerless grinding, coolants are used. Use care to avoid retention of coolant or water in exposed graphite parts. Because of the porous nature of graphite it will absorb any liquid-borne contaminants. If cooling is used in a machining operation, the graphite should subsequently be heated in a 300–400°F circulating heat oven to remove any retained fluids.

Endmilling, drilling and profiling graphite

Endmilling

Tool configuration: use square endmills with a small radius whenever possible. Diamond tools are more brittle than carbide and sharp corners may break upon entry into a cut at high feed rates. A radius of 0.010" to 0.015" will greatly strengthen the tool, providing extra durability.

For roughing at high feed rates 2-flute endmills should be used to minimize the possibility of tool breakage from flute packing. For general purpose and finish cutting use 4 flutes; CVD diamond endmill cost is the same, regardless of flute count. Improved surface finish and longer life usually result from multiple flutes in finishing operations.

Chipping: to avoid chipping, several techniques can be employed. Milling a short distance at the exit side of the part before starting the cut is very effective in avoiding breakout, just as chamfering the end of a cylinder is for turning. Lowering feed rates will lessen chipping upon exit, but directly affects productivity. Tool rotation can be used to lessen exit edge chipping for flat surfaces by using "down" milling rather than "up" milling rotation. Elimination of "up" milling rotation for contour milling will generally adversely affect tool life.

Feed rate: It is important to keep the tool engaged. If the feed rates drop too low (<.0001 to .0005 in. or <.00025 to .013 mm) the tool tends to burnish the part, rather than cut or crush. This can cause rapid tool wear.

When calculating the correct rpm for chip load at a given traverse speed it is important to consider if the machine tool is ever reaching the optimum traverse speed. It can take 1/2" or more to reach a high traverse speed. If the tool path has a lot of small adjustments, reduce rpms as the tool is never reaching the full traverse speed.

Machining parameters: starting conditions vary considerably; 2000 sfm and 0.004 inches per flute per revolution is a conservative starting point for 1/4" and larger endmills.

Starting parameters for endmilling graphite

Endmill dia. in. (mm)	Machine speed rpm	Cutting speed sfm (m/min)	Operation	Feed rate fpt (mm)
1/64	20,000 to 50,000	80 to 200 (25 to 60)	Finish	.0002–.0005 (.005–.013)
1/32 (1.0)	18,000 to 50,000	150 to 400 (45 to 120)	Finish	.0005–.001 (.013–.025)
1/16 (2.0)	12,000 to 50,000	200 to 1,000 (60 to 300)	General	.001–.002 (.025–.050)
			Finish	.0005–.001 (.015–.025)
1/8 (3.0)	6,000 to 50,000	200 to 2,000 (60 to 600)	General	.001–.002 (.025–.050)
			Finish	.0005–.001 (.015–.025)
3/16 (5.0)	4,000 to 50,000	200 to 2,500 (60 to 760)	General	.001–.002 (.025–.050)
			Finish	.0005–.001 (.015–.025)
1/4 (6.0)	3,000 to 45,000	200 to 3,000 (60 to 900)	General	.002–.004 (.050–.100)
			Finish	.001–.002 (.025–.050)
5/16 (8.0)	2,500 to 37,000		General	.002–.004 (.050–.100)
			Finish	.001–.002 (.025–.050)
3/8 (10.0)	2,000 to 31,000		General	.003–.005 (.075–.130)
			Finish	.001–.003 (.025–.075)
1/2 (12.0)	1,500 to 23,000		General	.003–.005 (.075–.130)
			Finish	.001–.003 (.025–.075)

rpm – revolutions per minute
sfm – surface feet per minute; m/min – meters per minute.
fpt – feed per tooth (feed per cutting edge/flute) in inches or (mm).
Feed per revolution (fpr) = feed rate (fpt) x no. of teeth (flutes).
Feed per minute (fpm) = fpr x rpm.

Drilling

Dust removal: special care should be used to clear the machining dust from holes during drilling. Proper removal will allow using higher spindle speeds as well as reducing drill wear.

Machining parameters: the table shows starting machining parameters for drilling graphite. As in all applications, these conditions will vary according to the grade of graphite being machined and the set-up and dust removal practices.

Starting parameters for drilling graphite

Drill diameter inches (mm)	Cutting speed sfm (m/min)	Feed rate ipr (mm/rev)
1/32–3/16 (1.0–5.0)	200–3000 (60–900)	.001–.004 (.025–.100)
3/16–1/4 (5.0–6.0)		.002–.005 (.050–.130)
1/4–5/16 (6.0–8.0)		.002–.006 (.050–.150)
5/16–3/8 (8.0–10.0)		.002–.008 (.050–.200)
3/8–1/2 (10.00–12.0)		.002–.010 (.050–.250)

ipr – inches per revolution. mm/rev – millimeters per revolution

Profiling

The table shows parameters for Dapra-style ball nose, flat bottom and back draft profiling cutters. *sp*^{3} profiling inserts are also available for Valenite Widia and Millstar tools.

Starting parameters for profiling graphite

Cutting dia. inches (mm)	Machine speed rpm	Cutting speed sfm (m/min)	Operation	Feed rate fpt (mm)
5/16 (7.94)	7,500 to 16,000	640 to 1,320 (195 to 400)	General	.005 – .008 (.130 – .200)
			Finish	.001 – .004 (.025 – .100)
3/8 (9.53)	6,500 to 13,500		General	.005 – .008 (.130 – .200)
			Finish	.001 – .004 (.025 – .100)
1/2 (12.7)	4,900 to 10,000		General	.009 – .015 (.230 – .400)
			Finish	.002 – .008 (.050 – .200)
5/8 (15.9)	3,900 to 8,000		General	.009 – .015 (.230 – .400)
			Finish	.002 – .008 (.050 – .200)
3/4 (19.1)	3,200 to 6,700		General	.009 – .015 (.230 – .400)
			Finish	.002 – .008 (.050 – .200)
1 (25.4)	2,400 to 5,000		General	.013 – .020 (.330 – .500)
			Finish	.004 –.012 (.100 – .300)
1-1/4 (31.8)	2,000 to 4,000		General	.013 – .020 (.330 – .500)
			Finish	.004 –.012 (.100 – .300)

rpm– revolutions per minute.
sfm – surface feet per minute; m/min – meters per minute.
fpt – feed per tooth (feed per cutting edge/flute) in inches or (mm).

Turning and milling graphite with inserted cutters

General

Tool configuration: Disposable inserts with a $^{1}/_{64}$" to $^{1}/_{32}$" nose radius are most effectively used for turning and milling graphite. A positive rake insert with a ground flank is preferred.

Surface finish: finish can be improved by selecting the appropriate tool geometry and feed rates. Larger nose radii will improve finish, but with increased tool pressure. A smaller nose radius will relieve pressure, but feed must be reduced to achieve comparable surface finish. DOC will not affect surface finish unless it causes excess tool pressure resulting in vibration, or if it is too light (under 0.005") to remove an adequate amount of material.

Breakout: breakout at the end of a pass is always a concern. This can be avoided by having a chamfer cut at the end of the part to ease exit of the tool, or provide stock which can later be cut off. Avoid square-nosed cut-off tools to prevent breaking prior to completion of the cut. A 20° relief angle is recommended.

Turning

Workpiece configuration: when machining long rods and cylinders, higher speeds and depths of cut can be employed with higher strength graphite materials.

Depth-of-cut: DOC should always be maximized when possible without incurring distortion of the part. When distortion or “whip” are present, feed and DOC must be adjusted. Lower feed rates will allow holding deeper cuts. Feed rates of 0.005" per revolution for roughing and between 0.001" and 0.003" for finishing might be necessary. Deeper cuts always generate higher pressures and larger fracturing particles, thereby producing rougher surface finish.

Machining parameters: The following starting parameters are recommended for general purpose and finish turning.

Starting parameters for turning graphite

Operation	Cutting speed sfm (m/min)	Feed rate ipr (mm/rev)
General purpose	2000–3000 (600–900)	.010–.020 (.250–.510)
Finish	3000–4000 (900–1200)	.003–.010 (.075–.250)

sfm – surface feet per minute; m/min – meters per minute.
ipr – inches per revolution; mm/rev – millimeters per revolution

Milling

Workpiece configuration: When milling large surfaces or volumes, higher speeds and depths of cut can be employed. Use higher strength graphite materials when there are thin walls involved.

Depth-of-cut: DOC should always be maximized when possible, to reduce multiple passes. Lower feed rates will allow holding deeper cuts. Feed rates of 0.004" / tooth / revolution for roughing and between 0.0005" and 0.002" / tooth / revolution for finishing might be necessary.

Multiple cutters: for multiple-pocket milling cutters it is recommended that axial alignment be used to align all inserts within ±0.0002" for best results. This will improve surface finish and reduce insert wear, as all the inserts will be cutting equally.

Machining parameters: The following starting parameters are recommended for general purpose and finish milling.

Starting parameters for milling graphite

Operation	Cutting speed sfm (m/min)	Feed rate in/tooth (mm/tooth)
General purpose	2000–4000 (600–1200)	.003–.008 (.075–.200)
Finish		

sfm – surface feet per minute; m/min – meters per minute.

Diamond tools available from *sp³*

Endmills

Fractional inch size endmills are available from *sp³* in diameters from 1/64" to 1/2". Over 150 styles include 2, 3 and 4 flutes, in standard and extended lengths. Metric endmills are available in diameters from 1.0 to 12.5 mm, in various lengths. End styles offered are square (with an end grind angle 1° to 3° concave) and ball end. Square-end endmills with a corner radius of .015" and .030" are available in several standard sizes. Optionally, any standard endmill may be ordered with a corner radius. *sp³* also offers **profiling endmills.**

For deep, zero draft graphite machining, *sp³* offers **premium endmills** with a relieved shank allowing a typical reach of 10 diameters. The premium endmills are available in fractional inch diameters from 1/64" (0.16" reach) to 1/4" (2.5"reach), and metric diameters of 1.0, 2.0, and 3.0 mm. Standard end styles are square and ball nose; corner-radiused tools can be ordered.

Drills

sp³ drills are industry-standard 2-flute spiral, with a 118° four-facet point. Drills are available in over 150 diameters in fractional inch, wire and letter sizes from #70 (0.0280") to 1/2", and in metric sizes from 1.0 to 12.5 mm. *sp³* will supply custom drills in diameters from 1/32" to 1/2" (1 mm to 12.5 mm) and lengths to 6 inches (150 mm).

sp³ also offers **spotting drills** and **center drills**.

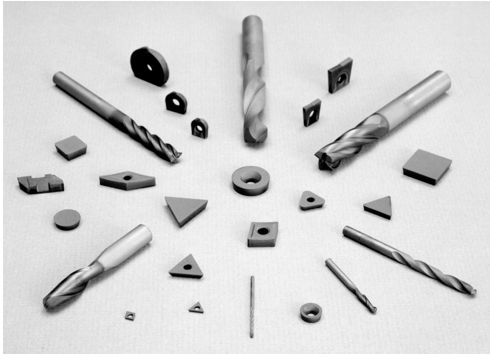
Routers

If you are using diamond pattern routers for side cutting fiberglass, consider switching to *sp³* multi-flute routers. These routers will give significantly longer tool life coupled with higher feed rates. A 10° helix angle ensures that there will be minimal material deflection force on thin panels. Use our standard endmill table for machining guidelines and adjust spindle speeds and feeds for 8 flutes.

Saws

sp³ offers diamond saws for slotting and other applications in a broad range of materials. The saws come in four standard widths and a coarse tooth pitch suitable for nonmetallic materials. Machining parameters are basically the same as for carbide saws although significantly higher speeds can be used in some materials, such as graphite.

Material	**Cutting speed** sfm (m/min)	**Chip load per tooth** thickness 0.02 – 0.0312" (0.508–0.792 mm)	**Chip load per tooth** thickness >0.0312" (>0.792 mm)
Graphite	1000 (300)	0.000025/0.0002" (0.00064/0.0051 mm)	0.0001/0.0007" (0.0025/0.018 mm)
Graphite/epoxy	800 (240)	0.000025/0.0002" (0.00064/0.0051 mm)	0.0001/0.0007" (0.0025/0.018 mm)



sp³'s offering of diamond tools encompasses over 500 styles of endmills, drills, insertable cutters, routers, saws, and other tools.

Inserts

sp³ offers over 240 insert styles, including multiple cutting edges, chip breakers, and geometries not possible with PCD tooling in many cases. The corner-to-corner coating uniformity of *sp³* inserts allows indexing without machine readjustments.

Insert shapes available include diamond (35°, 55° and 80°), triangle, square and round. All inserts are ANSI standard sizes and tolerances. Sizes (inscribed circle) range from 1/4" to 1 1/4". Styles include screw-down, pin-held and clamp.

Profiling inserts

Styles available: profiling inserts are available in three styles: ball nose, flat bottom and backdraft.

Ball nose cutters are designed to ramp up or down, and center cut, allowing for blending complex contours, shapes, cores and cavities.

Flat bottom cutters are designed for draft walls with only the corner radius of the insert engaged in the cut. This style reduces cutting pressure and deflection.

Back draft cutters are designed for straight walls and long extension in milling hard to reach areas in cores and cavities.

sp³ offers as standard products Dapra-style profiling inserts in sizes ranging from 5/16" to 1 1/4". The standard corner radius is 1/16"; tools may be ordered with a radius of 1/32". Millstar and Valenite Widia styles are available on special order.

Custom coating

sp³ will coat customer-supplied tungsten carbide tools, such as inserts, endmills, drills, and grooving inserts. These may be standard tools purchased by the customer or tools manufactured to a special configuration. *sp³* will also have tools ground to a customer-supplied print. Consult the *sp³* catalog or website for acceptable carbide grades for diamond coating.

About *sp*^{3}

Founded in 1993 and located in California's Silicon Valley, *sp*^{3} specializes in the development and manufacture of diamond cutting tools.

These products encompass a wide range of cutting tools for abrasive non-metallic materials and non-ferrous metals, as well as seals and other wear parts. Polycrystalline diamond is is grown on the surface of endmills, drills, inserts and other tools in a chemical vapor deposition or “CVD“ reactor. The result is a pure diamond surface, with no metallic binder. *sp*^{3} uses patented manufacturing processes and deposition systems designed and built by *sp*^{3}.

The MAC system

sp^3 offers its DIAbide tools in different surface configurations to optimize performance with specific workpiece materials and machining operations. These configurations are identified by Material Application Codes, or “MACs.” MACs are expressed as a 1- or 2-digit suffix to the part number, for example 12085-**7** or TPG-432-**18**.

Higher numbered MACs represent a thicker diamond surface and a more robust tool; lower numbered MACs tend to have a sharper edge. For less abrasive and less brittle materials, a smoother finish may be obtained with a lower MAC. *sp*^{3} recommends using the lowest MAC that is suitable for the material to be machined.

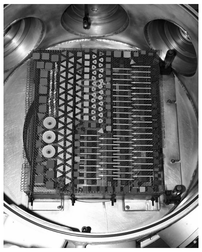
A complete guide to the MAC system is included in the *sp*^{3} diamond tool catalog and on the *sp*^{3} website.

sp^3's website

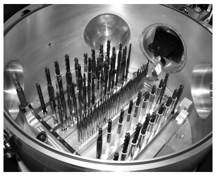
The website offers a wealth of information:

- A complete on-line catalog
- Guidance on selecting the optimum tool for your application via the Material Application Code
- New product information
- Information on custom coating services
- Information on CVD technology
- Machining parameters
- Descriptions of tool applications in various workpiece materials
- Comparative tool tests
- Current and archive copies of the newsletter *Spotlight on sp*^{3}
- Magazine articles featuring diamond tools
- Notice of tradeshows at which *sp*^{3} will be exhibiting
- The ability to request literature, request sample tools, and order tools
- Company information, including employment

With the deposition process completely under computer control, the *sp*^{3} Model 600 Reactor produces tools and other products with a uniform, consistent diamond surface, in high volume.



The reactor design allows large quantities of diamond tools and other diamond items to be produced per deposition cycle.



Evaluation sample tools

If you have an application that is unproven with diamond tools, *sp*^{3} will provide samples for your evaluation.

We realize there is a cost on your part to test a new tool, just as there is a cost to *sp*^{3} in supplying samples. However, without testing it is hard for both the tool supplier and the tool user to make advancements. We hope that your efforts will be rewarded with a success that reduces your overall costs or helps you solve a machining problem that hasn't had a satisfactory answer in the past.

Testing is also very important for *sp*^{3}. Shared results from your testing help *sp*^{3} to better understand how our tools performed in your application, allowing us to optimize our products to better meet your needs.

When we supply evaluation samples, we ask that you complete the brief Field Evaluation Report we supply with the samples and return it, along with the tested tools, when you have completed your evaluation. We learn a great deal about our tools’ performance by inspecting them after they have been tested, and we can often recommend adjustments to your application or improve our process to provide your company with even greater productivity gains or tooling cost reductions.

To request an evaluation sample do one of the following:

- Call your *sp*^{3} tool distributor
- Call *sp*^{3} at 800-773-9940
- Use the sample request on our website, www.sp3inc.com

For assistance call: 800-773-9940
Visit our website: www.sp3inc.com

sp^3 Inc., 505 East Evelyn Avenue, Mountain View, CA 94041. Tel (650) 966-0630; fax (650) 966-0633; www.sp3inc.com.