Nakreslený spoj otvorím v Protely 99.



👥 Design	Explorer	the state in some	
	File View Help		
12 🚔	New		
Explorer	Open		
Desi	Exit		

陰 🛩 ?	
Explorer	
👌 Design Desktop	
Active Design Stations	New Design Database
	Location Password
	The document you selected to open must be imported into a new design database.
	Protel 99 SE stores all design documents in an integrated design database. Depending upon the storage type, documents are either stored in a single design database or as stand-alone files and folders on your disk drive.
	Design Storage Type MS Access Database
	MS Access type stores all design documents in a single MS Access database drive.
	Database File Name TEST.DDB
	Database Location
	E:\Projekt\Private\ATMEGA328\Export Browse
	OK Cancel Help



Skontrolovať polohu spoja. Pokiaľ je spoj zbytočne posunutý do stredu, koordináty sú príliš veľké. Mal som problémy pri exportovaní takéhoto spoja. Preto odporúčam jednoducho spoj vyznačiť (Select All) a manuálne presunúť vľavo dole (pomocou ikony v tvare strelky).





Kliknúť na CAM Manager a následne vygenerovať doň dáta pre Gerber.













CAM Outputs for TEST.cam	If you enable the 'Plot u PCB will produce drill guide present on the board. O	ised drill guide layer pairs' option plots for all layer pairs which are Otherwise you can specify which
	TopLayer BottomLa	yer ☐ Minor plots
	Cancel	Back Next>



V ďalšom kroku sa vygenerujú dáta pre vŕtačku.



xplorer	TEST.DDB Documents TEST.PCB Recycle E	Bin Design Team 📓 CAM Outputs for TEST.cam
Design Desktop	Name	Туре
K Active Design Stations W HOME-PC TEST.DDB Gr Design Team Gr Recycle Bin	Gerber Output 1	Gerber
Documents CAM Outputs for TEST.cam	Output Wizard	8
TEST.PCB	What	is the name for this NC Drill output?
		rill Output 1
		Cancel < Back Next> Einish



lorer	TEST.DDB Documents TEST.PCB Recycle Bin	Design Team 🔹 CAM Outputs for TEST.cam
Design Desktop Active Design Stations HOME-PC TEST.DDB Design Team Revuele Bin	Meme Gerber Output 1 NC DHII Output 1	Type Gerber NC Drill
CAM Outputs for TEST.cam	Gerber Setup General Layers Dril Drawing Dril (Film Size	Guide Mechanical Layers Apertures Advanced
Dvojklik a zmenit	∑ (horizontal) 20000mil ∑ (vertical) 16000mil Border size 1000mil Aperture Matching Tolerances Plus 0.005mil Migus 0.005mil Batch Mode	<u>Keep leading and trailing zeroes</u> Suppress reading <u>zeroes</u> Suppress trailing zeroes Position on Film <u>Reference to shoulde origin</u> <u>Reference to relative origin</u> <u>Seriouse file</u> Plotter Type
nastavenia	 Sgparate file per layer Panelize layers Other 	
	☐ <u>G</u> 54 on aperture change	☑ Optimize change location commands

Skontrolovať štandardne prednastavené údaje pre vŕtačku a zmeniť ich.

xplorer TE	ST.DDB Documents TEST.PCB Recycle Bin	Design Team 📓 CAM Outputs for TEST.cam
Design Desktop	ame I Control 1 I NC Drill Output 1	Type Gerber NC Drill
	NC Drill Setup	<u>ହ</u> 🗙
TEST.PCB	Options Advanced Leading/Trailing Zeroes Eeep leading and trailing Stoppers trailing zeroes	zeroes Coordinate Positions Reference to absolute origin Reference to relative origin
Dvojklik a zmenit nastavenia	Other	n commands

Následne sa vygenerujú potrebné súbory.





Súbory sú vygenerované v prednastavenom adresári protelu. Následne sa môžu presunúť na potrebné miesto.



Ako posledné sa môžu vygenerované súbory skontrolovať pomocou programu CAM350.

