



# DragonFly2020 Pro 3D Printer Spec Sheet\*

## INTRODUCTION

The following spec sheet is for users of the Nano Dimension DragonFly™ 2020 Pro 3D Printer for the production of professional multilayer Printed Circuit Boards (PCBs).

FEATURE	DESIGN RULE
PCB size	Recommended: 10x 10cm (4" x 4") Maximum: 20x20 cm (8"x 8")
Space	1/2 oz./ft <sup>2</sup> (17 μm) - min. 125 μm (5 mil) 1 oz./ft <sup>2</sup> (35 μm) - min. 125 μm (5 mil) 2 oz./ft <sup>2</sup> (70 μm) - min. 150 μm (6 mil)
Trace	1/2 & 1 oz. (17 μm)- min. 100 μm (4 mil) 2 oz./ft <sup>2</sup> (35 μm) - min. 125 μm (5 mil)
Through holes (TH)	min. 400 μm (16mil)
PTH	min. 400 μm (16mil)
Via size <sup>1</sup>	min. 200 μm (8mil)
Signal layer thickness <sup>2</sup>	6 μm (0.25mil)      17 μm (1/2 oz.) 9 μm (1/4 oz.)      35 μm (1 oz.) 12 μm (1/3 oz.)      70 μm (2 oz.)
Dielectric layer thickness	9 μm-2 mm (0.35-80mil)
Edge spacing <sup>3</sup>	1.25 mm (50mil)
Number of layers	No known limitations within 2mm thickness
Component Assembly & soldering <sup>4</sup>	Stencil – Yes, on mechanical fixture Pick and place – Yes Manual tip soldering – Yes Low temp. automatic soldering – Yes
Marking and labeling	Yes, with clearance as per Space

## BENCHMARK ADDITIONAL REQUIREMENTS

The following information is required, when sending a design for benchmarking:

1. Gerber x274 design files. Please indicate which EDA tools and version were used.
2. Stack-up description with layer names, orders and thicknesses.
3. Dedicated mechanical layer (route).



### Footnotes:

1. Clearance is lower than in traditional PCB, as per Space above
2. Any thickness above 6 μm is possible, in 3 μm increments
3. Electrical clearance including internal GND surface
4. Currently do not support press fit, wave soldering and high temperature reflow

### Notes:

Soldermask is printed with dielectric material, and is usually 30-50 μm (1.2-2mil) thick  
\*Data is subject to change without notice

## CONFIDENTIAL

Further questions and queries:

robert@nano-di.com +972-54-5893250