ICS 31.180, 49.140 SLOVENSKÁ TECHNICKÁ NORMA Apríl 2017

STN	**Zabezpečovanie výrobkov kozmického programu. Pravidlá na návrh dosiek s plošnými spojmi.**	**STN EN 16602-70-12** 31 0542

Space product assurance - Design rules for printed circuit boards

Táto norma obsahuje anglickú verziu európskej normy.
This standard includes the English version of the European Standard.

Táto norma bola oznámená vo Vestníku ÚNMS SR č. 03/17

Obsahuje: EN 16602-70-12:2016

124313

EUROPEAN STANDARD **EN 16602-70-12**

NORME EUROPÉENNE

EUROPÄISCHE NORM

October 2016

ICS 31.180; 49.140

English version

Space product assurance - Design rules for printed circuit boards

Assurance produit des projets spatiaux - Règles de conception des circuits imprimés

Raumfahrtproduktsicherung - Designregeln für Leiterplatten

This European Standard was approved by CEN on 22 May 2016.

CEN-CENELEC Management Centre:
Avenue Marnix 17, B-1000 Brussels

Ref. No. EN 16602-70-12:2016 E

Table of contents

European foreword **12**

Introduction **13**

1 Scope **14**

2 Normative references **15**

3 Terms, definitions and abbreviated terms **16**

3.1 Terms from other standards 16

3.2 Terms specific to the present standard 16

3.3 Abbreviated terms 24

4 Principles **26**

4.1 Qualified PCBs 26

4.2 Manufacturing tolerances 26

4.3 Reliability of design 26

5 Design review and MRR **28**

5.1 Overview 28

5.2 Documentation 28

6 General design and production requirements **30**

6.1 Reliability of design 30

6.2 Choice of materials and build-up 30

6.2.1 Overview 30

6.2.2 Material selection 33

6.3 Selection of the PCB manufacturer 33

6.4 Traceability and marking 33

7 Rigid PCBs **34**

7.1 PCB build-up 34

7.1.1 General 34

7.1.2 Copper styles 34

7.1.3 Dielectric thickness 36

7.2 PCB dimension 39

7.3 Thickness of PCB 39
7.3.1 General 39
7.3.2 Polyimide PCB 40
7.3.3 Epoxy PCB 40
7.3.4 Number of copper layers in PCB 40
7.3.5 Aspect ratio of vias 40
7.4 Track width and spacing 41
7.4.1 General 41
7.4.2 Manufacturing tolerances for width and spacing 41
7.4.3 External layers 42
7.4.4 Normal pitch tracks on internal layers 43
7.4.5 Fine pitch tracks on internal layers 44
7.4.6 Routing to AAD footprint on internal layers 45
7.5 Pad design 46
7.5.1 Non-functional pad removal 46
7.5.2 Pad dimensions 46
7.5.3 Non-circular external pads 48
7.6 Copper planes in rigid PCB 49
7.7 Design considerations for the prevention of sliver and peelable 50
7.8 PCB surface finish 50
7.8.1 Metallization 50
7.8.2 Solder mask 51

8 Flex PCBs 52

8.1 Overview 52
8.2 Dynamic applications 52
8.3 PCB build-up 52
8.3.1 General 52
8.3.2 Dielectric materials 52
8.3.3 Copper cladding 53
8.3.4 Copper planes in flex PCB 53
8.4 Track design 54
8.5 Through holes 55
8.5.1 Annular ring 55
8.5.2 Vias and pads 56
8.5.3 Tear drop pad for flex PCB 56
8.6 Bending radius 57
8.6.1 Overview 57

8.6.2 General 57
8.7 Sculptured flex PCB 58
8.7.1 Overview 58
8.7.2 General 58
8.7.3 Copper foil dimensions for build-up 58
8.7.4 Connection finger 59
8.7.5 Through-holes 60
8.7.6 Bending radius 61

9 Rigid-flex PCBs 62
9.1 Overview 62
9.2 General 62
9.3 Build-up 63
9.4 Cover layer 64
9.5 Interface of rigid part and flexible part 64
9.6 Pads 64

10 Thermal rules and heat sinks 65
10.1 Overview 65
10.2 General requirements 65
10.3 Specific requirements for external heat sink 65
10.3.1 Overview 65
10.3.2 Construction of the interface between PCB and heat sink 65
10.3.3 Dimensional requirements 66
10.4 Specific requirements for internal heat sink 68
10.4.1 General 68
10.4.2 Cu thickness and type 68
10.4.3 CIC and Molybdenum inserts 69
10.4.4 Dimensional requirements 69

11 HDI PCBs 71
11.1 Overview 71
11.2 Justification 71
11.3 Microvia technology 71
11.4 Microvias 72
11.4.1 Build-up of microvia layers 72
11.4.2 Design of microvias 74
11.4.3 Pad design for microvia 74
11.4.4 Annular ring for microvias 75

11.5 Core PCB for HDI 75
11.5.1 General build-up 75
11.5.2 Annular ring on vias for fine pitch footprint 76
11.5.3 Track width and spacing on external layers 77
11.5.4 Track width and spacing on internal layers for impedance control and routing to AAD 77
11.5.5 Track width and spacing on internal layers for differential pair routing within the footprint of 1,0 mm pitch AAD 78
11.5.6 Aspect ratio of vias for footprint of AAD with 1 mm pitch 79

12 PCBs for high frequency applications 81
12.1 Material selection 81
12.2 Build-up of RF PCB 81
12.3 Embedded film resistors 81
12.4 Thickness of RF PCB 83
12.5 Track width and spacing 83
12.5.1 External layers 83
12.5.2 Internal Layers 84
12.6 Pad design 84
12.6.1 Pad dimensions 84
12.6.2 Non-functional pads 84
12.7 Surface finish 84
12.8 Profiled layers and vias 84

13 Electrical requirements for PCB design 86
13.1 Overview 86
13.2 General 87
13.3 PCB drying 87
13.4 Electrical characteristics 87
13.5 Floating metal 88
13.6 Current rating 88
13.6.1 Overview 88
13.6.2 Requirements for temperature increment 89
13.6.3 Requirements for the model IPC-2152 for current rating 89
13.6.4 Amendments to the model from IPC-2152 90
13.7 Provisions to prevent open circuit failure on critical tracks 91
13.7.1 Overview 91
13.7.2 Routing 92
13.8 Voltage rating 92

13.8.1 Overview 92
13.8.2 General requirements 92
13.8.3 Spacing on flex and rigid-flex laminate 94
13.8.4 Conformal coating 95
13.9 Double insulation design rules for critical tracks 96
13.9.1 Overview 96
13.9.2 Critical nets 96
13.9.3 Prevention of short circuit 96
13.10 Insulation distance of combined requirements on rigid PCB 101
13.11 Controlled impedance tracks 106
13.11.1 Definitions specific to controlled impedance 106
13.11.2 General rules 106
13.11.3 Microstrip and stripline 106
13.11.4 Line impedance termination for end-to-end configuration 107
13.11.5 Line impedance termination for multidrop configuration 107
13.12 Digital PCB 108
13.12.1 Overview 108
13.12.2 Zone management and routing 108
13.12.3 Criticality of digital signals 109
13.13 Analog PCB 110
13.13.1 Overview 110
13.13.2 Criticality of analog signals 110
13.13.3 Routing and shielding 111
13.14 Mixed analog-digital PCB 112

14 Design for assembly 113

14.1 Overview 113
14.2 General 113
14.3 Placement requirements 114
14.3.1 Conductive patterns 114
14.3.2 Components 115
14.3.3 Component pads 117
14.3.4 Fan out of SMT pads 118
14.3.5 Fan out of PTH 120
14.4 Specific requirements for fused tin-lead finish 121
14.5 Dimensional requirements for SMT foot print 121
14.5.1 Overview 121
14.5.2 General 122

14.5.3 Bipolar components 123
14.5.4 SOIC components 124
14.5.5 J-leaded components 125
14.5.6 LCC components 126
14.5.7 Flat pack components 128
14.5.8 AAD components 129

15 Design of test coupon 130
15.1 Design rules for test coupon 130
15.2 Test coupon design 130

Annex A (normative) PCB definition dossier - DRD 133
A.1 DRD identification 133
A.1.1 Requirement identification and source document 133
A.1.2 Purpose and objective 133
A.2 Expected response 133
A.2.1 Scope and content 133
A.2.2 Special remarks 139
A.2.3 Example figures 139

Annex B (normative) PCB manufacturing dossier – DRD 144
B.1 DRD identification 144
B.1.1 Requirement identification and source document 144
B.1.2 Purpose and objective 144
B.2 Expected response 144
B.2.1 Scope and content 144
B.2.2 Special remarks 145

Annex C (informative) Example of capability list of PID 146

Annex D (informative) Track current rating computation methodology 148
D.1 Introduction of the three models 148
D.1.1 Overview 148
D.1.2 Formulae for the three models 149
D.1.3 Example of current rating 150
D.2 Track current rating computation based on IPC-2152 150
D.3 Track current rating computation based on CNES/QFT/IN.0113 154
D.4 Track current rating computation based on IPC-2221A 158

Annex E (informative) Example of calculation of PTH pad dimensions 162

Annex F (informative) Prevention of resin starvation and cracks 164

F.1 Prevention of resin cracks....164
F.2 Prevention of resin starvation....164

Annex G (informative) Example of MRR checklist....166

Bibliography....170

Figures

Figure 3-1: Simplified build-up of HDI PCB....20
Figure 3-2: Projected peak-to-peak insulation distance....23
Figure 7-1: Example of automated fine pitch routing and possible improvements....45
Figure 7-2: Comparison between circular and oblong pads showing annular ring and the centre of the hole misregistered with the centre of the pad....48
Figure 7-3: Grid copper plane with openings....49
Figure 7-4: Example of peelable (left) and sliver (right)....50
Figure 8-1: Clearance of tracks on flex PCBs....54
Figure 8-2: Tracks on flex, defining termination and bending zones....55
Figure 8-3: Teardrop reinforcement of terminal pads in flex PCB....56
Figure 8-4: Bending radius of assembled flex....57
Figure 8-5: Sculptured flex circuit....58
Figure 8-6: Build-up of sculptured flex circuit....59
Figure 8-7: Connection finger of sculptured flex circuit....60
Figure 8-8 Side view of a component hole for sculptured flex....61
Figure 9-1 Example of a build-up of a 6 layer symmetric rigid-flex....62
Figure 10-1: Lay out of heat sink and PCB....67
Figure 10-2: drilling holes and slots in internal heat sink....70
Figure 10-3: Angle of intersection of overlapping holes....70
Figure 11-1: Build-up of HDI PCBs with staggered (left) and stacked (right) microvia....72
Figure 11-2: Dimple on a microvia....74
Figure 11-3: Tear drop pad design....76
Figure 11-4: Example of routing two differential pair tracks between vias in a footprint of an AAD with 1,27 mm pitch....78
Figure 11-5: Example of routing two differential pair tracks between vias in a footprint of an AAD with 1 mm pitch....80
Figure 13-1: Cross section of PCB with insulation distances....87
Figure 13-2: Example of double insulation by increasing distance in X,Y and by not superimposing copper on adjacent layers....100
Figure 13-3: Example of double insulation by increasing distances in X,Y and by using two insulators in Z direction....100
Figure 13-4: Edge coupled differential striplines....107

Figure 14-1: Illustration of surface pattern to adjacent surroundings....115
Figure 14-2: Illustration of component placing on PCB w.r.t. adjacent surrounding....116
Figure 14-3: Illustration of pad placing on PCB with respect to adjacent surroundings 118
Figure 14-4: Fan out from SMT pad to via 119
Figure 14-5: Example of point symmetric AAD footprint....120
Figure 14-6: Track width (I_c) ratio to PTH pad (D) diameter. 120
Figure 14-7: Track length (Lc) between soldering pad of PTH (D_1) and via pad (D_3) 121
Figure 14-8:Illustration of bipolar component pads 123
Figure 14-9: Illustration of SOIC component pads 124
Figure 14-10: Illustration of J-leaded component pads 125
Figure 14-11: Illustration of LCC component pads....126
Figure 14-12: Illustration of FP and QFP component pads 128
Figure 14-13: Illustration of AAD component pads....129

Figure A-1 : Example of PCB mechanical layout 139
Figure A-2 : Example of a drilling drawing 140
Figure A-3 : Example of a table indicating as-designed plated hole diameters 141
Figure A-4 : Example of build-up data 142
Figure A-5 : Example of the archive file with CRC 143
Figure D-1 : Comparison between CNES and IPC-2152 current rating at 5°C increase 149
Figure D-2 : IPC-2152: Current rating [A] vs cross sectional area [mm^2] in double log scale 151
Figure D-3 : IPC-2152: Track width [mm] vs cross sectional area [mm^2] 151
Figure D-4 : IPC-2152: Current rating based on Figure D-2, range 0-25 A 152
Figure D-5 : IPC-2152: Current rating based on Figure D-2, range 0-10 A 152
Figure D-6 : IPC-2152: Current rating based on Figure D-2, range 0-5 A 153
Figure D-7 : IPC-2152: Current rating based on Figure D-2, range 0-2 A 153
Figure D-8 : CNES/QFT/IN.0113: Current rating [A] vs cross sectional area [mm^2] in double log scale 155
Figure D-9 : CNES/QFT/IN.0113: Track width [mm] vs cross sectional area [mm^2] 155
Figure D-10 : CNES/QFT/IN.0113: Current rating based on Figure D-8, range 0-25 A 156
Figure D-11 : CNES/QFT/IN.0113: Current rating based on Figure D-8, range 0-10 A 156
Figure D-12 : CNES/QFT/IN.0113: Current rating based on Figure D-8, range 0-5 A 157
Figure D-13 : CNES/QFT/IN.0113: Current rating based on Figure D-8, range 0-2 A 157
Figure D-14 : IPC-2221A: Current rating [A] vs cross sectional area [mm^2] in double log scale 159
Figure D-15 : IPC-2221A: Track width [mm] vs cross sectional area [mm^2] 159
Figure D-16 : IPC-2221A: Current rating based on Figure D-14, range 0-25 A 160

Figure D-17 : IPC-2221A: Current rating based on Figure D-14, range 0-10 A 160

Figure D-18 : IPC-2221A: Current rating based on Figure D-14, range 0-5 A 161

Figure D-19 : IPC-2221A: Current rating based on Figure D-14, range 0-2 A 161

Figure E-1 : PCB manufacturing tolerances for registration and annular ring for HDI PCB 162

Figure E-2 : PCB manufacturing tolerances for registration and annular ring for rigid-flex sequential PCB 163

Tables

Table 6-1: Characteristics of some example dielectric materials for laminates 32

Table 7-1: As-designed versus as-manufactured copper foil thickness 36

Table 7-2: Example of worst case as-manufactured insulation distance for laminate with double-sided copper cladding 39

Table 7-3: Tolerance on track width and spacing of internal and external layers 42

Table 7-4: Minimum as-manufactured track width and spacing for external layers as a function of copper thickness 43

Table 7-5: Minimum as-manufactured track width and spacing for internal layers as a function of copper thickness 44

Table 8-1: Minimum as-manufactured dimensions of sculptured flex conductor 59

Table 12-1: Minimum spacing and width as-designed for RF elements on external layers 84

Table 13-1: Legend of terms 86

Table 13-2: Current overload limits 88

Table 13-3: Minimum insulation distance as function of voltage on as-manufactured PCB for rigid laminate. 94

Table 13-4: Minimum insulation distance as function of voltage on as-manufactured PCB for flex laminate 95

Table 13-5: Minimum double insulation distance as function of voltage on as-manufactured PCB for rigid laminate. 98

Table 13-6: Minimum double insulation distance as function of voltage on as-manufactured PCB for flex laminate. 98

Table 13-7: Minimum insulation distances on rigid PCB as function of all combined requirements (*part 1 of 4*) 102

Table 14-1: Minimum distance as-manufactured of the surface pattern to the adjacent surroundings 114

Table 14-2: Minimum distance as-manufactured of components to adjacent surroundings 116

Table 14-3: Minimum distance as-manufactured to component pads 117

Table 14-4: Minimum track length to component pads 120

Table 14-5 Legend for dimensions of components and footprint 122

Table 14-6 As-manufactured pad sizes for bipolar components 123

Table 14-7: As-manufactured pad sizes for SOIC components 124
Table 14-8: As-manufactured pad sizes for J-leaded components 125
Table 14-9: As-manufactured pad sizes for LCC components.....127
Table 14-10: As-manufactured pad sizes for FP and QFP components 128
Table 14-11: As-designed pad diameter for AAD components 129

Table C-1 : Example of technical capabilities of PCB manufacturer as specified in the PID.....146

European foreword

This document (EN 16602-70-12:2016) has been prepared by Technical Committee CEN-CENELEC/TC 5 “Space”, the secretariat of which is held by DIN.

This European Standard (EN 16602-70-12:2016) originates from ECSS-Q-ST-70-12C.

This European Standard shall be given the status of a national standard, either by publication of an identical text or by endorsement, at the latest by April 2017, and conflicting national standards shall be withdrawn at the latest by April 2017.

Attention is drawn to the possibility that some of the elements of this document may be the subject of patent rights. CEN [and/or CENELEC] shall not be held responsible for identifying any or all such patent rights.

This document has been prepared under a mandate given to CEN by the European Commission and the European Free Trade Association.

This document has been developed to cover specifically space systems and has therefore precedence over any EN covering the same scope but with a wider domain of applicability (e.g. : aerospace).

According to the CEN-CENELEC Internal Regulations, the national standards organizations of the following countries are bound to implement this European Standard: Austria, Belgium, Bulgaria, Croatia, Cyprus, Czech Republic, Denmark, Estonia, Finland, Former Yugoslav Republic of Macedonia, France, Germany, Greece, Hungary, Iceland, Ireland, Italy, Latvia, Lithuania, Luxembourg, Malta, Netherlands, Norway, Poland, Portugal, Romania, Slovakia, Slovenia, Spain, Sweden, Switzerland, Turkey and the United Kingdom.

Introduction

PCBs are used for the mounting of electronic components to produce PCB assemblies that perform electrical functions. The PCBs are subjected to thermo-mechanical stress during assembly such as soldering of components, rework and repair under normal terrestrial conditions. In addition the assembled PCBs are exposed to the launch and space environment. The reliability of the circuit depends on the robustness of the design, among other factors. Moreover, PCB design with high technological complexity enables the use of complex components with advanced functionality.

1
Scope

This European Standard specifies the requirements for the supplier and PCB manufacturer for PCB design.

This European Standard is applicable for all types of PCBs, including sequential, rigid and flexible PCBs, HDI and RF PCBs.

This European Standard can be made applicable for other products combining mechanical and electrical functionality using additive or reductive manufacturing processes, as used in PCB manufacturing. Examples of such products are slip rings and bus bars.

This European Standard may be tailored for the specific characteristics and constraints of a space project in conformance with ECSS-S-ST-00.

2 Normative references

The following documents, in whole or in part, are normatively referenced in this document and are indispensable for its application. For dated references, only the edition cited applies. For undated references, the latest edition of the referenced document (including any amendments) applies.

EN reference	Reference in text	Title
EN 16602-00-01	ECSS-S-ST-00-01	ECSS system – Glossary of terms
EN 16602-70-02	ECSS-Q-ST-70-02	Space product assurance - Thermal vacuum outgassing test for the screening of space materials
EN 16602-70-08	ECSS-Q-ST-70-08	Space product assurance - Manual soldering of high-reliability electrical connections
EN 16602-70-10	ECSS-Q-ST-70-10	Space product assurance - Qualification of printed circuit boards
EN 16602-70-11	ECSS-Q-ST-70-11	Space product assurance - Procurement of printed circuit boards
EN 16602-70-38	ECSS-Q-ST-70-38	Space product assurance -High-reliability soldering for surface-mount and mixed technology
EN 16603-20	ECSS-E-ST-20	Space engineering – Electrical and electronic
EN 16603-20-06	ECSS-E-ST-20-06	Space engineering - Spacecraft charging
	IPC-2152, August 2009	Standard for determining current carrying capacity in printed board design
	IPC-4101D, April 2014	Specification for base materials for rigid and multilayer printed boards
	IPC-4562A, April 2008	Metal foil for printed wiring applications