

Literatura za učenje programskog paketa Altium Designer 6 nalazi se na web stranicama proizvođača:

<http://www.altium.com/community/support/learningguides/altiumdesigner6learningguides/>

Obzirom na veliki opseg dostupnih materijala, dokumentaciju je preporučljivo proučavati prema sljedećem rasporedu:

Projektna radna okolina
AR0137 Introduction to Altium Designer
AR0101 An Overview of Electronic Product Development in Altium Designer
GU0112 Welcome to the Altium Designer Environment*
AP0129 Project Essentials
TU0117 Getting Started with PCB Design
Crtanje električne sheme
AP0109 Schematic Editing Essentials
AR0123 Connectivity and Multi-Sheet Design*
TU0115 Editing Multiple Objects*
TU0104 Generating a Custom Bill of Materials*
Definiranje novih komponenata (za električne sheme i tiskane pločice)
AR0104 Component, Model and Library Concepts
TU0103 Creating Library Components
AR0105 Enhanced Library Management Using Integrated Libraries*
TU0111 Building an Integrated Library*
Projektiranje tiskane pločice
TU0110 Preparing the Board for Design Transfer*
AP0135 Interactive and Differential Pair Routing ¹
AR0111 Specifying the PCB Design Rules and Resolving Violations*
AP0101 Polygon Pours and Copper Regions*
TR0127 OutputJob Editor Reference*
AP0137 Working with the Board Insight System*

* Označava dokumente koje ili obrađuju neke naprednije teme ili sadrže dosta detalja koji se mogu preskočiti prilikom početnog upoznavanja s programom.

¹ dio "*differential pairs routing*" može se preskočiti (od str. 14)