

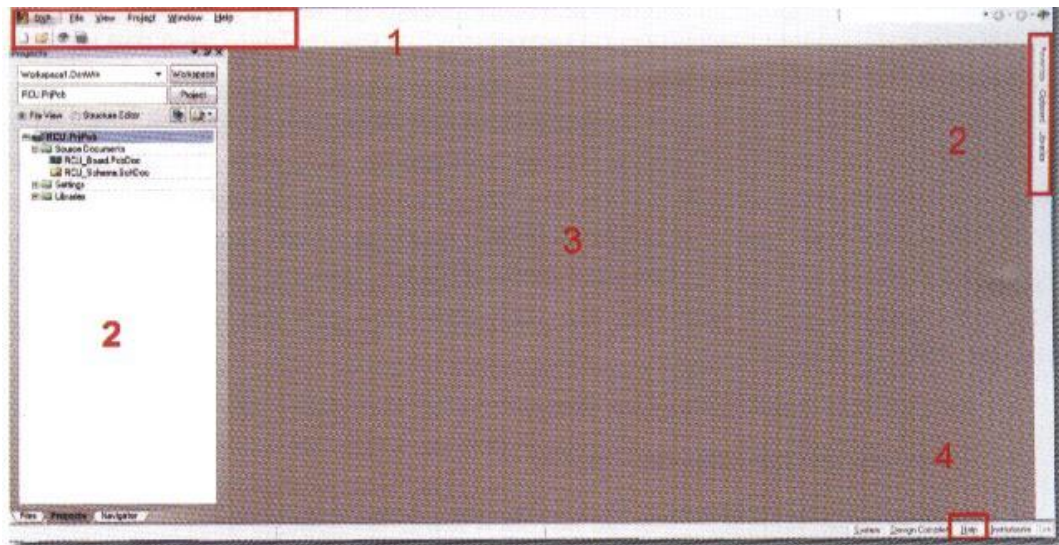
FEATURES OF DESIGN OF
AIRBORNE RADIO
ELECTRONIC FACILITIES IN
THE ALTIUM DESIGNER
ENVIRONMENT

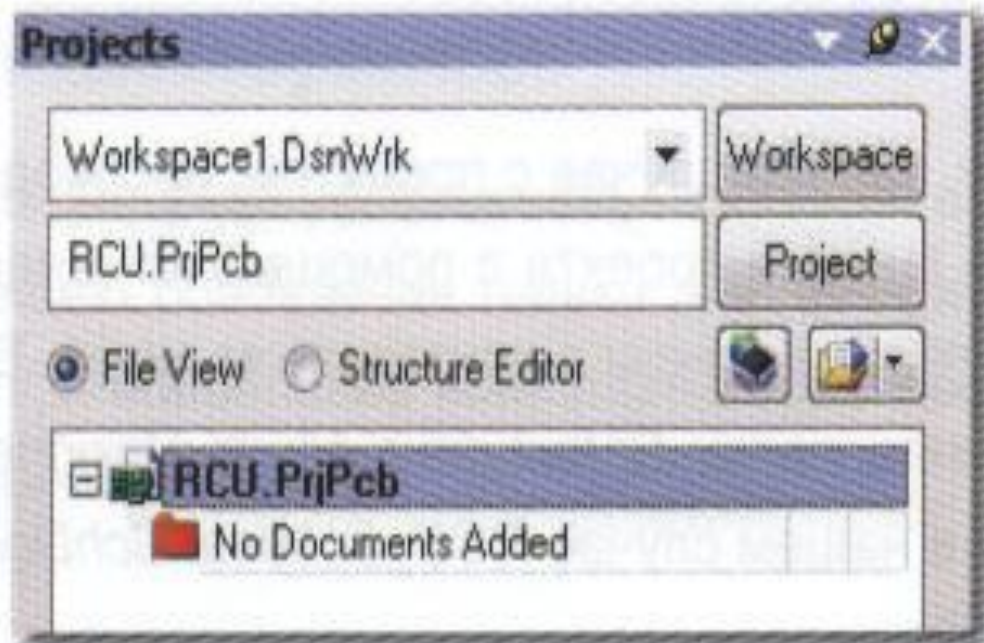
STRUCTURAL MODULES OF ALTIUM DESIGNER

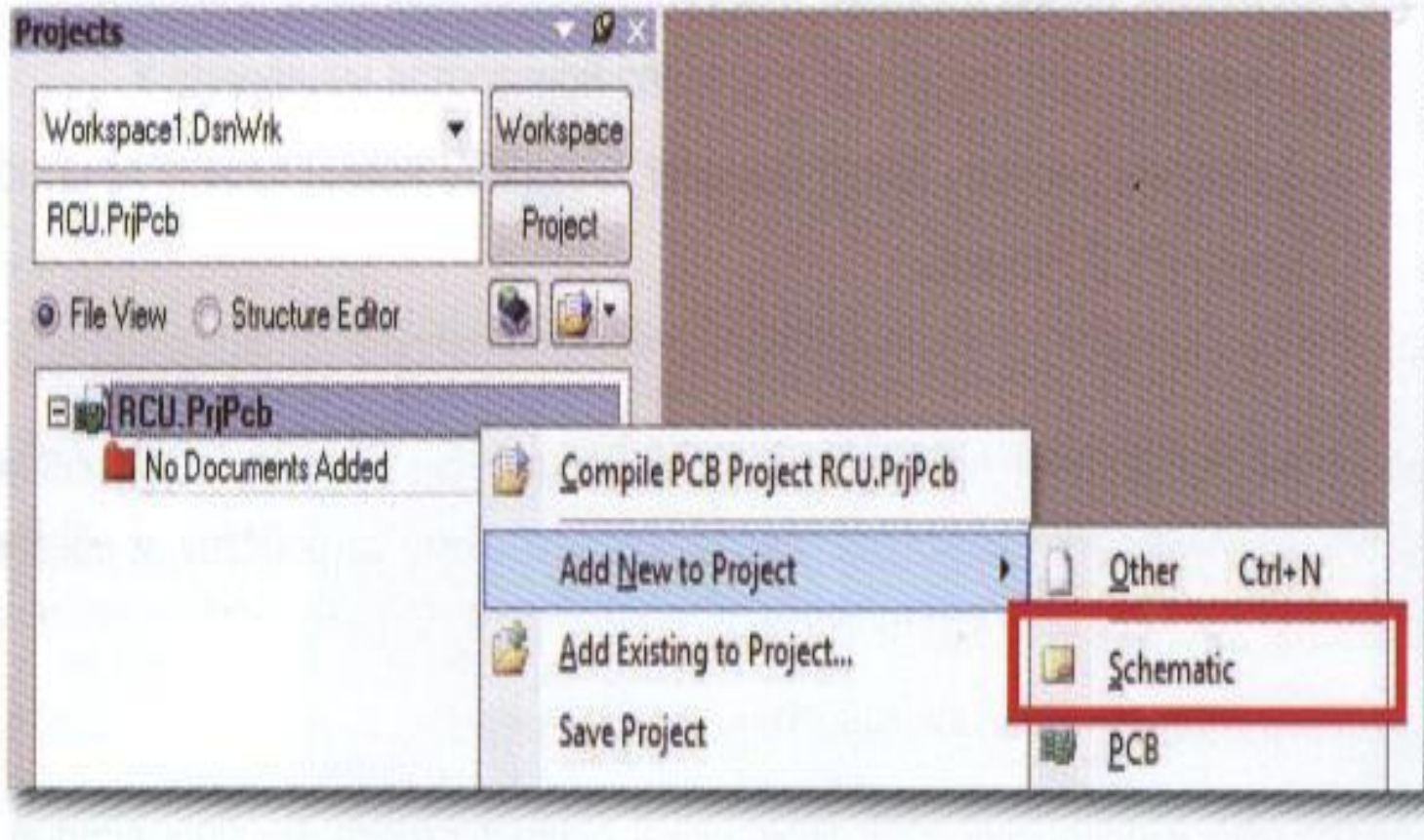
- development of electronic circuits;
- design of printed circuit boards;
- development of embedded software;
- mixed analog-digital simulation;
- signal integrity analysis;
- technological preparation of manufacturing;
- design of systems based on PLIC and their debugging with the use of the Altium NanoBoard breadboard.

THE ALTIUM DESIGNER WINDOW CONTAINS THE FOLLOWING BASIC ELEMENTS:

1. System menu and toolbars, content and composition of which varies depending on the type of active document;
2. Auxiliary panels, which have multiple display modes;
3. Workspace;
4. Integrated support for Altium Designer, which provides access to the built-in help pages and resources located in the Internet (Altium Wiki).

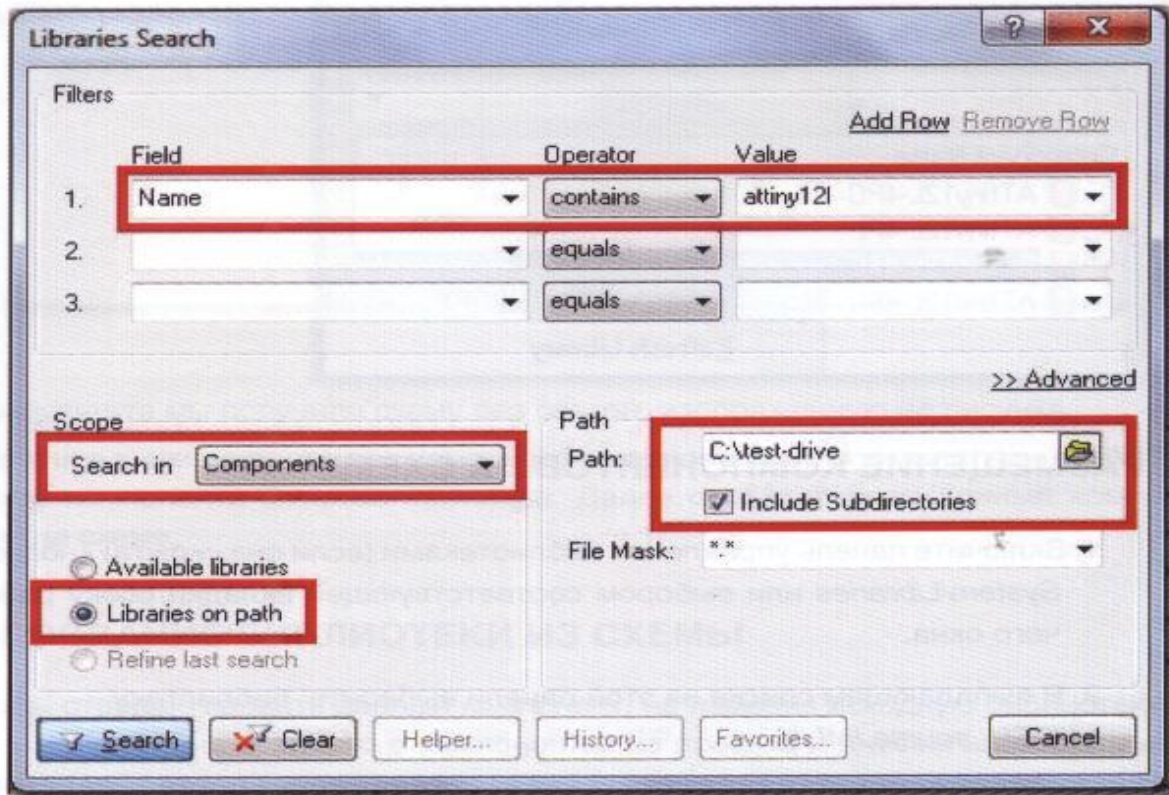






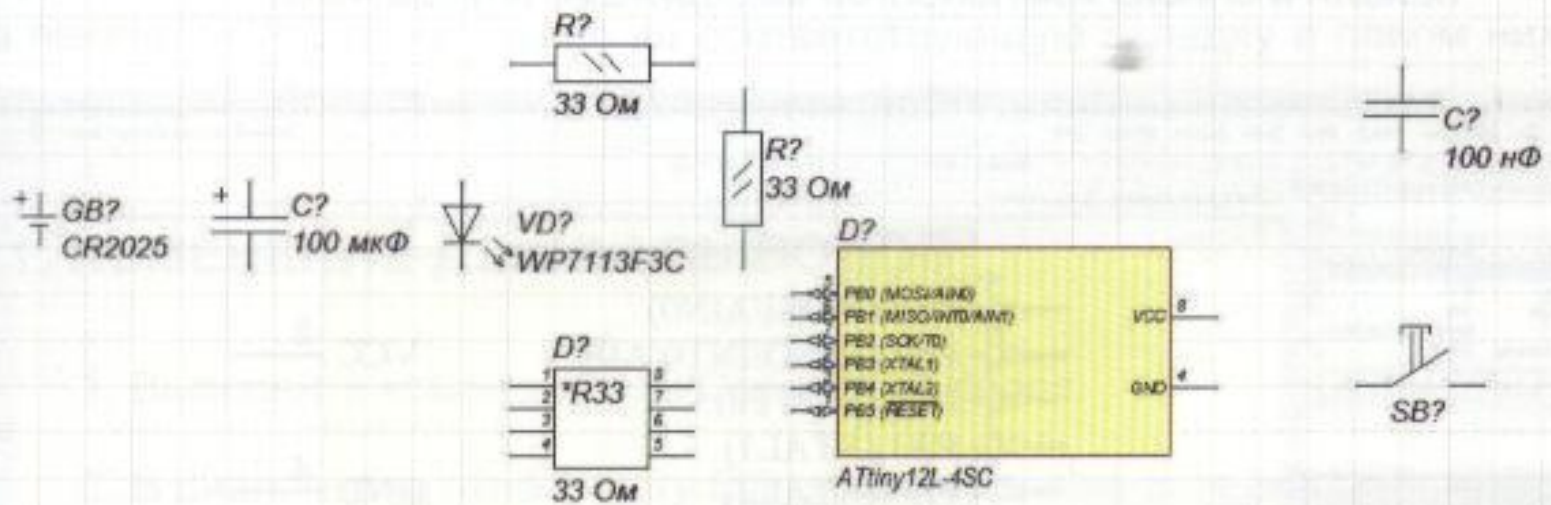
POSSIBLE TYPES OF THE CONNECTED DOCUMENT:

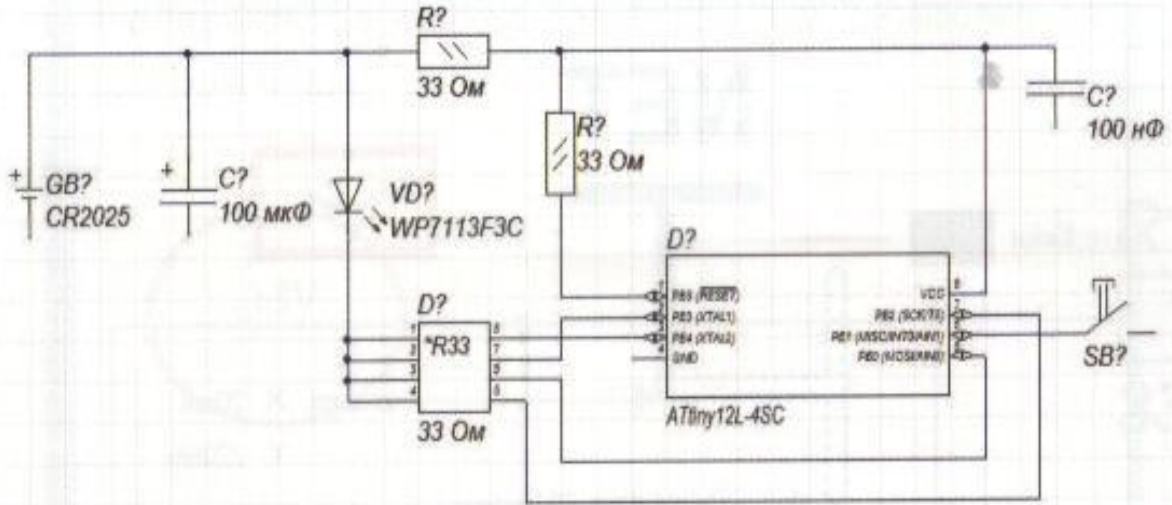
1. Schematic – a schematic document;
2. PCB – a file of a printed circuit board;
3. Schematic Library – a library of schematic symbols;
4. PCB Library – a library of topological sites;
5. CAM Document — a document of a CAM-program;
6. Output Job File – a file of output data for processing;
7. Database Link File – a file pointer to the link to the database;
8. Text Document – a text document;
9. Other – other documents.

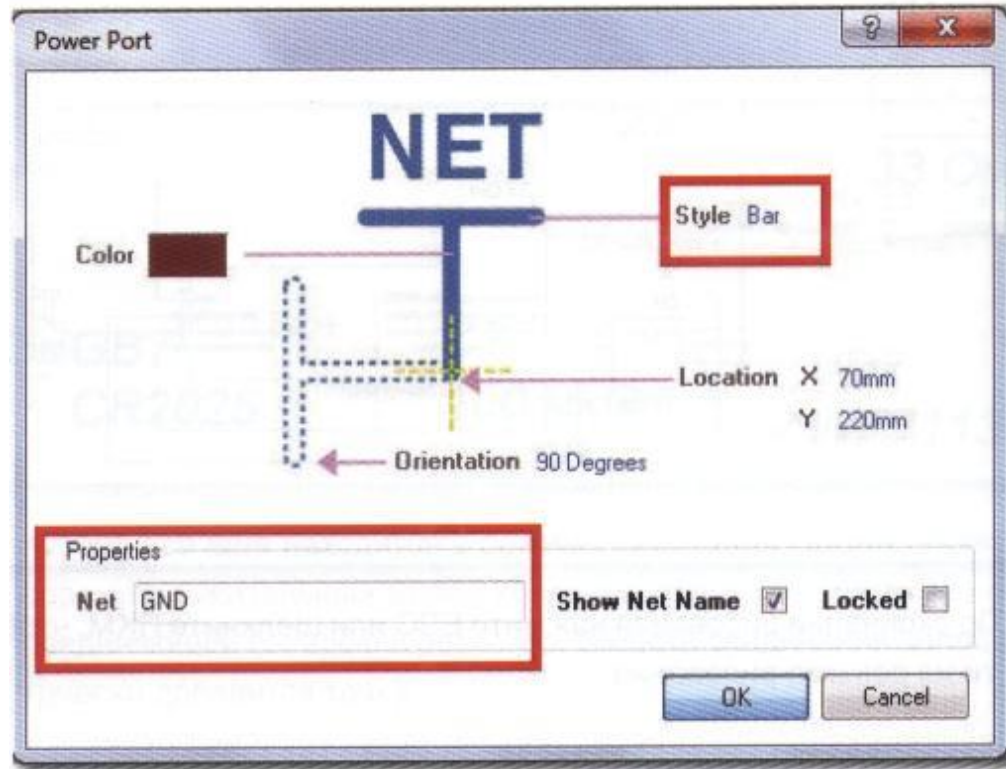


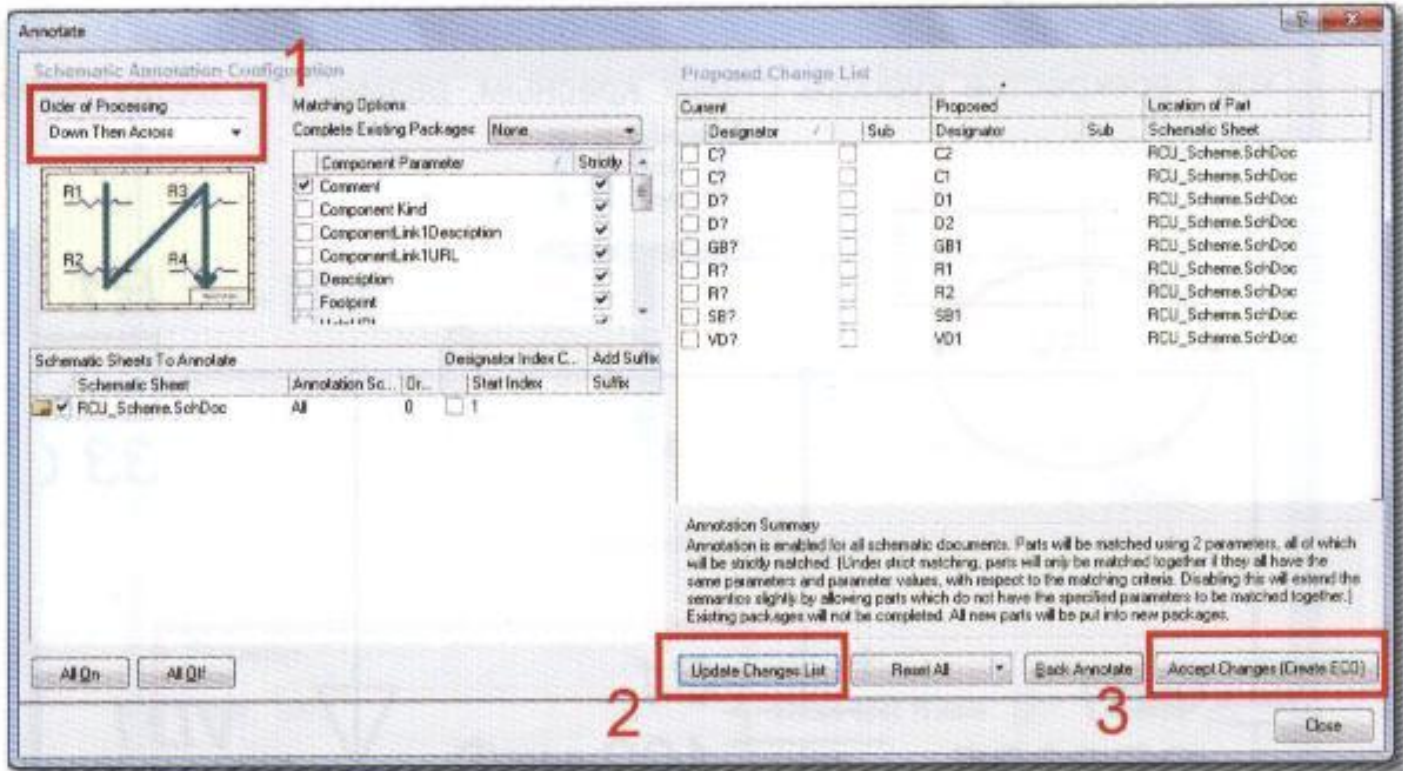
THE COMPONENT PROPERTIES WINDOW

- Properties – the main component properties (Designator – the positional notation, Comment – the component name);
- Library Link – the link to the library;
- Graphical – the information about the component graphic display;
- Parameters – the component attributes (hence the information about the component will be used to form the text part of design);
- Models – the component models (sites);
- Edit Pins – the table of component pins.









Order of Processing
Down Then Across



- Matching Options
- Completes Existing Packages: None
- Component Parameter
 - Comment
 - Component Kind
 - ComponentLinkIDescription
 - ComponentLinkURL
 - Description
 - Footprint

Schematic Sheets To Annotate	Annotation Sc...	Dr...	Start Index	Add Suffix
RCU_Scheme.SchDoc	All	0	1	

Current		Proposed		Location of Part
Designator	Sub	Designator	Sub	Schematic Sheet
<input type="checkbox"/> C?		C2		RCU_Scheme.SchDoc
<input type="checkbox"/> C?		C1		RCU_Scheme.SchDoc
<input type="checkbox"/> D?		D1		RCU_Scheme.SchDoc
<input type="checkbox"/> D?		D2		RCU_Scheme.SchDoc
<input type="checkbox"/> GB?		GB1		RCU_Scheme.SchDoc
<input type="checkbox"/> R?		R1		RCU_Scheme.SchDoc
<input type="checkbox"/> R?		R2		RCU_Scheme.SchDoc
<input type="checkbox"/> SB?		SB1		RCU_Scheme.SchDoc
<input type="checkbox"/> VD?		VD1		RCU_Scheme.SchDoc

Annotation Summary
Annotation is enabled for all schematic documents. Parts will be matched using 2 parameters, all of which will be strictly matched. (Under strict matching, parts will only be matched together if they all have the same parameters and parameter values, with respect to the matching criteria. Disabling this will extend the semantics slightly by allowing parts which do not have the specified parameters to be matched together.) Existing packages will not be completed. All new parts will be put into new packages.

All On All Off Update Changes List Reset All Back Annotate Accept Changes (Creates ECO) Close

THE COMPILATION PROCESS CONSISTS OF THE FOLLOWING STAGES:

- Setting project options;
- Starting compilation;
- Search and fix errors.

THE PREFERENCES IN THE PRINTED BOARDS EDITOR

- Local preferences of the current document.
- Global preferences of the editor.
- Display preferences.