

CR-8000™

Top features and benefits

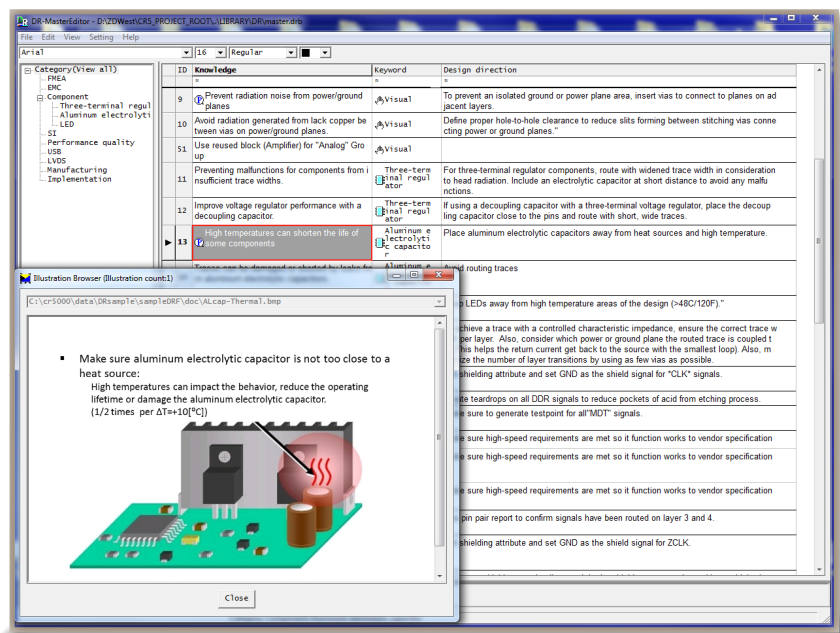
- Connect process documents with integrated engineering knowledge base
- Reduce costly errors and design iterations by specifying engineering guidelines and expertise early in the design process
- Associate images, documents and URLs to improve communication of design instructions
- Apply standard and design-specific guidelines to logical circuit designs and automatically create a design checklist
- Save time creating guideline checklists through automatic assignment of design guidelines to circuits, components, nets and pins
- Cross-probe logical and physical designs, with design viewers to enhance and ease the design review process
- Validation and traceability of engineering requirements through support for automatic digital sign-off in checklists
- Easy addition of new guidelines into knowledge base improves quality of future designs and provides growth in expertise across engineering organizations

Circuit DR Navi - Engineering knowledge base and design review checklist

Introduction

Engineering teams often maintain process requirements and best practice documents to support the PCB design process (which is a best practice in itself). These documents are typically stored outside the design project and are not intelligently associated with the design, resulting in inconsistent communication of engineering specifications. During design reviews, engineers provide feedback and guidelines to address design issues, but this is often provided on paper, PDF or by email. This makes it difficult for design teams to manage and confirm each instruction provided during design reviews.

CR-8000 Circuit DR Navi helps design teams consolidate engineering expertise and best practices into a central repository integrated with the design process. During logical circuit design, engineers can access a standard set of requirements, or create project-specific instructions to create a structured checklist to drive design guidelines across the PCB design flow. These checklists aid layout designers and confirm that engineering guidelines have been observed. They are also used as an intelligent platform for engineers to communicate feedback during design reviews. With Circuit DR Navi, new information can easily be added to the engineering knowledge base and integrated within the design process to drive engineering intent early on, validate and trace requirements, and expand and share design expertise across the organization. This leads to shorter PCB design cycles, reduced product development costs, and faster time-to-market.



Engineering knowledge base helps define engineering guidelines

DR-Master Editor – Engineering knowledge base

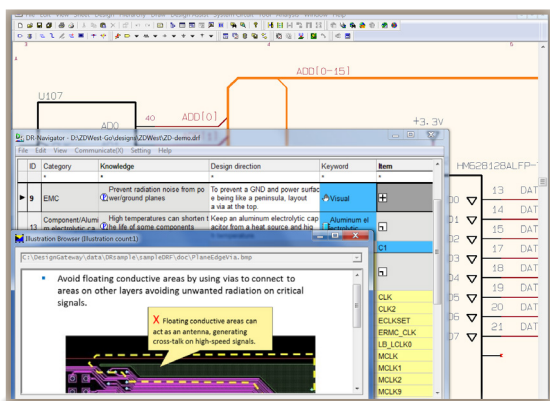
Creating your corporate knowledge base for electronic design is straightforward using DR-Master Editor. Take your existing design process guidelines or checklists from word documents, spreadsheets, or engineering experts and directly apply or import them into the engineering knowledge base. Within DR-Master Editor, users can define technology or application-specific categories, such as high-speed design or manufacturing, and apply guidelines to each of the categories.

For each guideline or knowledge item defined in DR-Master Editor, users may associate design instructions and links to visual guides to help the engineer and PCB designer during the course of the design process. The visual guides can be graphic images, documents, web links or even video files. Each guideline can also be assigned a unique keyword, which allows circuit objects such as components, component groups, nets and pins to be associated automatically during logical circuit design, to save time and effort. Templates of standard or application-driven guidelines can be created for engineers to download and apply to their design.

DR-Navigator – Assigning engineering guidelines

During the logical circuit design, engineers use the DR-Navigator to start creating their design checklist with a template of guidelines, or to access the master knowledge base to individually select the knowledge items for their design. The checklist can be stored with the design or in a central location. Engineers also define new project-specific rules and associate design instructions and visual guides as available in the DR-Master Editor. This allows flexibility in ensuring that standard guidelines are realized during the design flow, and in identifying new guidelines for consideration in future designs. The Illustration Browser within DR-Navigator provides access to key documents or visual guides to aid users during circuit design.

With DR-Navigator, engineers can link and cross-probe with their schematic for interactive association of guidelines



Assign knowledge items rules automatically and export instructions directly into the circuit

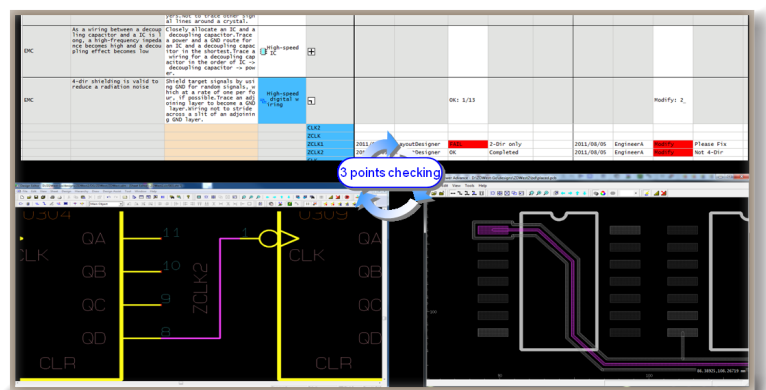
to design objects. By using keywords defined for each knowledge item, users can automatically extract design objects and associate them to the guidelines for fast and easy creation of their checklist. DR-Navigator supports bidirectional exchange of design instructions so users can access them during the design, and reference them in the checklist.

DR-Manager – Checklist for engineering guidelines

Once the checklist has been created and reviewed by engineers, it can be shared during physical circuit design using DR-Manager. PCB designers use the DR-Manager to start design placement and routing with an upfront set of rules and engineering guidelines to avoid rework identified during design reviews. During board layout, designers cross-probe between the checklist items and the physical design to highlight and address each object assigned with a guideline. DR-Manager also provides access to the Illustration Browser for supplemental instruction on each design item.

As the layout designer works through the checklist and the board design, they can apply a sign-off to each item with a digital signature, along with a status and comment regarding the compliance of each design instruction. DR-Manager is also used during the design review process so design teams can have a controlled sign-off process and provide follow-up instructions if necessary. This helps avoid the error-prone management of multiple print or PDF copies when implementing design changes.

DR-Manager includes support to cross-probe with CR-8000 Design Gateway, Design Force and Board Viewer Advance, along with Zuken's CR-5000. Design teams can conduct three-point checking between the logical and physical circuit design and the engineering checklist. With Circuit DR Navi, design teams can collaborate intelligently with an integrated method to validate and trace requirements anytime during the design process to ensure engineering intent is observed.



Three-point checking between the checklist, schematic, and layout

Expanding the knowledge base

As design practices evolve and new technologies are adopted, knowledge items added during circuit design process can be transferred to the master engineering knowledge base. All users can access the master knowledge base anytime to review current design guidelines and best practice.