IDA-STEP User Guide

Contents

1. Overview
2. About
3. Register, Update and Install
4. Preferences
4.1. 3D View
4.1.1. Display Styles
4.2. GD&T
4.2.1. Annotation Style
4.3. General
4.3.1. Keys
4.3.2. Language
4.3.3. Network Connections
4.3.4. Perspectives
4.3.5. Virtual Machine Memory
4.4. Help
4.5. Numbers
4.5.1. Quantities and Units
4.6. PCB Stratums Cross-section
4.6.1. Color Schemas
5. General User Interface Elements
5.1. Workbench Window Layout
5.2. Views
5.2.1. Opening View
5.2.2. Moving and Docking View
5.2.3. Detaching View
5.2.4. Minimizing and Maximizing View
5.2.5. Fast Viewing
5.2.6. Resizing View
5.2.7. Find Text
5.2.8. Sorting
5.3. Perspectives
5.3.1. Open Perspective
5.3.2. New Perspective
5.3.3. Customize Perspective
5.3.4. Reset Perspective
5.3.5. Switching Between Perspectives
5.4. Toolbars
5.4.1. File buttons
5.4.2. GD&T buttons
5.4.3. Datum buttons
5.4.4. Tool buttons
5.5. Statusbar
5.6. Workbench Menus
5.6.1. File menu
5.6.2. Edit menu
5.6.3. GD&T menu
5.6.4. Constructive
5.6.5. Window menu
5.6.6. Help menu

6. File Handling
6.1. New File
6.2. Opening File
6.3. Import
6.3.1. Import Wizard
6.3.2. Importing Preferences
6.3.3. Importing EAGLE
6.3.4. Importing Data from CSV File
6.3.5. Importing Board Station
6.3.6. Importing Allegro
6.3.6.1. Importing Allegro Designs
6.3.6.2. Importing Example Data
6.3.7. Importing CircuitCAM
6.3.8. Importing PADS
6.3.9. Importing CR-5000
6.3.10. Importing Expedition
6.3.11. Importing CADIF
6.4. Export
6.4.1. Export Wizard
6.4.2. Exporting Preferences
6.4.3. Exporting STEP: 2D PCA/PCB data Extruded to 3D
6.4.4. Exporting STEP (ISO 10303 - part 21)
6.4.5. Exporting to CSV
6.4.6. Exporting to STL
6.4.7. Exporting to VRML
6.4.8. Exporting to CircuitCAM
6.5. Manage Libraries
6.5.1. 3D Part Libraries
6.5.2. E-CAD to M-CAD Mapping
6.6. Language
7. Installed Features
7.1. Electronics
7.2. GD&T Editor
7.3. IDA-STEP Framework
7.4. Import Allegro
7.5. Import Board Station
7.6. Import CADIF
7.7. Import CircuitCAM
7.8. Import CR-5000
7.9. Import EAGLE
7.10. Import Expedition
7.11. Import PADS
7.12. Layer Stack Editor
7.13. Viewer Basic
7.14. Viewer Pro 3D
8. Pre-defined Perspectives
8.1. GDT
8.2. Parts 3D
8.3. PCB Layer Library
8.4. PCB Layer Stack
8.5. PCB/PCA
8.6. Tasks
9. Available Views
9.1. 2D Viewer
9.1.1. Opening 2D Data View

9.1.2. 2D View
9.1.3. Zoom In
9.1.4. Zoom Out
9.1.5. Zoom to Fit
9.1.6. Zoom to Highlighted
9.1.7. Zoom Window
9.1.8. Highlight Object
9.1.9. Display Settings
9.2. 3D Viewer
9.2.1. Displaying Geometry in 3D View
9.2.2. Overview of 3D View
9.2.3. Predefined Orientations
9.2.4. Save Current 3D View
9.2.5. Appearance Style
9.2.6. Style/Annotation Model
9.2.7. Selection Modes
9.2.8. View Control
9.2.9. Front and Back Clipping
9.2.10. Inspecting Model
9.2.11. Measuring
9.2.12. Optimize Geometry
9.2.13. Re-triangulate
9.2.14. Management of GD&T Elements
9.2.14.1. Datum Feature
9.2.14.2. Datum Target
9.2.14.3. Datum System
9.2.14.4. Geometric Tolerances
9.2.14.5. Dimensions
9.2.14.6. Annotation Plane
9.2.14.7. Text Annotation
9.2.14.8. Shape Aspect
9.2.14.9. Constructive Geometry Elements
9.2.14.9.1. Apex
9.2.14.9.2. Symmetry Axis
9.2.14.9.3. Symmetry Plane
9.2.14.9.4. Symmetry Point
9.2.14.9.5. Common Line
9.2.14.9.6. Common Plane
9.2.14.9.7. Intersection
9.2.14.9.8. Offset
9.2.14.9.9. Perpendicular Line
9.2.14.9.10. Perpendicular Plane
9.2.14.9.11. Extension
9.2.14.9.12. Tangent Plane
9.2.14.9.13. Point
9.2.14.9.14. Line
9.2.14.9.15. Plane
9.2.14.10. Align Offset
9.3. Datasets
9.3.1. Datasets View
9.3.2. Creating Dataset
9.3.3. Renaming Dataset
9.3.4. Removing Dataset
9.4. Design Stack Models
9.4.1. Design Stack Models View

9.4.2. Creating Design Stack Model
9.4.3. Editing Design Stack Model Data
9.5. Documents
9.5.1. Documents View
9.5.2. Document Hierarchy
9.5.3. Creating New Document
9.5.4. Adding Version
9.5.5. Adding File
9.5.6. Adding File Reference
9.6. FMECA
9.6.1. FMECA View
9.6.2. Creating Failure Mode
9.6.3. Creating Failure Cause
9.6.4. Editing Failure Mode Data
9.6.5. Editing Failure Cause Data
9.6.6. Removing
9.7. GD&T
9.7.1. GD&T View
9.7.2. Shape Aspects
9.7.2.1. Composite Shape Aspect
9.7.2.2. Constructive Geometry
9.7.2.2.1. Create Apex
9.7.2.2.2. Create Symmetry Axis
9.7.2.2.3. Create Symmetry Plane
9.7.2.2.4. Create Symmetry Point
9.7.2.2.5. Create Common Line
9.7.2.2.6. Create Common Plane
9.7.2.2.7. Create Intersection
9.7.2.2.8. Create Offset
9.7.2.2.9. Create Perpendicular Line
9.7.2.2.10. Create Perpendicular Plane
9.7.2.2.11. Create Extension
9.7.2.2.12. Create Tangent Plane
9.7.3. Annotation Plane
9.7.4. Datum Feature
9.7.5. Datum Target
9.7.6. Datum System
9.7.7. Dimensions
9.7.8. Geometric Tolerances
9.7.9. Text Annotation
9.8. Geometry Details
9.8.1. Geometry Details View
9.8.2. Selection
9.8.3. Construction Geometry Models
9.8.4. Style/Annotation Models
9.8.5. Saved Views
9.8.6. Editing Visibility
9.8.7. Editing Colors
9.8.8. Editing Transparency
9.8.9. Editing Transformation Data
9.9. Header
9.9.1. Header View
9.10. Log
9.10.1. Log View
9.11. Maintenance Activities

9.11.1. Maintenance Activities View
9.11.2. Creating Maintenance Activity
9.11.3. Editing Maintenance Activity Data
9.11.4. Removing Maintenance Activity
9.12. Part Details
9.12.1. Part Details View
9.12.2. Selection
9.12.3. Visibility and Transparency
9.12.4. Editing Part Details
9.12.5. Create Assembly from Parts
9.12.6. Adding Component
9.12.7. Adding 3D Part to Library
9.12.8. Copying and Pasting
9.12.9. Removing
9.13. Parts
9.13.1. Parts View
9.13.2. Part Classifications
9.13.3. Creating Part Version
9.13.4. Creating New Version
9.13.5. Copying and Pasting
9.13.6. Editing Data
9.13.7. Assigning Part Version to Classification
9.13.8. Removing Part Version from Classification
9.13.9. Creating Classification System
9.13.10. Creating Classification
9.13.11. Creating Classification Attribute
9.13.12. Deleting
9.13.13. Moving
9.14. Passage Technologies
9.14.1. Passage Technologies View
9.14.2. Creating Passage Technology
9.14.3. Editing Passage Technology Data
9.15. Passages
9.15.1. Passages View
9.15.2. Passages Hierarchy
9.15.3. Creating Passage
9.15.4. Editing Passage Data
9.16. Persons and Organizations
9.16.1. Persons and Organizations View
9.16.2. Editing Person Data
9.16.3. Editing Organization Data
9.16.4. Editing Position Data
9.16.5. Editing Address Data
9.16.6. Creating New Element
9.16.7. Copying and Pasting
9.16.8. Deleting
9.16.9. Moving Element
9.16.10. Removing from Organization
9.17. Products
9.17.1. Products View
9.17.2. Creating Product
9.17.3. Editing Product Data
9.17.4. Removing Product
9.18. States
9.18.1. States View

9.18.2. Creating State
9.18.3. Adding Sub State
9.18.4. Editing State Data
9.18.5. Removing
9.19. Stratum Technologies
9.19.1. Stratum Technologies View
9.19.2. Creating Stratum Technology
9.19.3. Editing Stratum Technology Data
9.20. Stratums
9.20.1. Stratums View
9.20.2. Stratums Hierarchy
9.20.3. Creating Layer
9.20.4. Creating Sequential Group
9.20.5. Changing Layer Position
9.20.6. Moving Autorow to Root
9.20.7. Editing Layer Data
9.21. Stratums Cross-section
9.21.1. Stratums Cross-section View
9.21.2. Toggling Collapsed/Exploded view
9.21.3. Changing Color Schema
9.21.4. Highlighting Components
9.22. Tasks
9.22.1. Tasks Window
9.22.2. Task Hierarchy
9.22.3. Editing Task Data
9.22.4. Creating Task
9.22.5. Moving
9.22.6. Removing
9.22.7. Assigning Task Invocation
9.22.8. Assigning Zonal Breakdown
9.22.9. Assigning Part
9.22.10. Assigning Position
9.22.11. Assigning Auxiliary Materials
9.22.12. Assigning Tools
9.23. Zonal Breakdowns
9.23.1. Zonal Breakdown View
9.23.2. Zonal Breakdown Hierarchy
9.23.3. Creating Zonal Breakdown
9.23.4. Editing Zonal Breakdown Data
9.23.5. Assigning Position
9.23.6. Moving
9.23.7. Removing
10. Dynamic columns
10.1. Customizing views
10.2. Inserting Column
10.3. Removing Column
10.4. Configuring Column
10.5. Insert Column Wizard
10.6. Column types
10.6.1. Activity
10.6.2. Approval
10.6.3. Assembly component definition
10.6.4. Certification
10.6.5. Classification
10.6.6. Chosen method

10.6.7. Curve Color
10.6.8. Description
10.6.9. Date
10.6.10. Date and Time
10.6.11. Design
10.6.12. Document
10.6.13. Element
10.6.14. External data exists
10.6.15. External file location
10.6.16. ID
10.6.17. Magnitude
10.6.18. Material property
10.6.19. Name
10.6.20. Network assignment
10.6.21. Person / Organization
10.6.22. Position
10.6.23. Security
10.6.24. Subject
10.6.25. Surface Color
10.6.26. Text property
10.6.27. Transformation
10.6.28. Transparency information
10.6.29. Quantity
10.6.30. FMECA Specific
10.6.30.1. Enumeration property
10.6.30.2. Failure detection method
10.6.30.3. Failure effect
10.6.30.4. FMECA RPN
10.6.31. LSE Specific
10.6.31.1. Allocated technology
10.6.31.2. Allowed component terminal extent
10.6.31.3. Conductive layers
10.6.31.4. Connected minimum annular ring
10.6.31.5. Finished deposition thickness
10.6.31.6. Finished passage extent
10.6.31.7. From
10.6.31.8. Laminate stiffness class
10.6.31.9. Layer position
10.6.31.10. Maximum aspect ratio
10.6.31.11. Maximum feature size requirement
10.6.31.12. Minimum aspect ratio
10.6.31.13. Minimum finished feature size
10.6.31.14. Minimum finished feature spacing
10.6.31.15. Minimum finished size
10.6.31.16. Minimum interior extent
10.6.31.17. Passage deposition material attribute
10.6.31.18. Passage deposition material identification
10.6.31.19. Passage filling material attribute
10.6.31.20. Passage filling material identification
10.6.31.21. Passage terminus condition
10.6.31.22. Plated passage
10.6.31.23. Stratum purpose
10.6.31.24. Stratum technology material designation
10.6.31.25. Stratum technology occurrence base
10.6.31.26. Stratum technology occurrence definition

10.6.31.27. Stratum technology occurrence number
10.6.31.28. Stratum thickness
10.6.31.29. Technology type
10.6.31.30. To
10.6.31.31. Total thickness
10.6.31.32. Unconnected minimum annular ring
10.7. Columns perspective
10.7.1. Open columns perspective
10.7.2. Create columns perspective
10.7.3. Rearange columns
10.7.4. Restet default columns perspective
11. Printing
12. Errors and Warnings
12.1. Reporting Errors
12.2. Installation Errors
12.2.1. Error: Insufficient disc space on drive C:
12.2.2. Error: Failed to connect to https://services.ida-step.net/ through the Internet
12.3. Licensing Errors
12.3.1. Error: License ID not found
12.3.2. Error: License is blocked
12.3.3. Error: No rehostings left
12.4. Internal error

1. Overview

IDA-STEP is an extensible software system for handling many different aspects of industrial product model data. It is very tightly linked with STEP, the "STandard for the Exchange of Product model data", ISO 10303.
The underlying infrastructure of IDA-STEP consists of:

- Java programming language
- Eclipse for all GUI and installation aspects (see www.eclipse.org)
- JSDAI for handling product data (see www.jsdai.net)

The functionality of IDA-STEP is wrapped up in *software components* that are called *features* by the underlying Eclipse system. We use both terms in this documentation. By licensing and installing more software components the functionality of IDA-STEP can be increased step by step. You can check for the licensed software components together with an optional expiration date and optional installation status in the Register, Update and Install dialogue that you can start from the **Help** menu.
You can check for the currently installed software components and the exact version number of the component by starting About IDA-STEP dialogue from the **Help** menu and then clicking on **Feature Details**. The following kind of features are available:

- **IDA-STEP Framework** for handling the main aspects of the Graphical User Interface (GUI) and all aspects on registration, installation and update.
- **Viewer Basic** for being able to view the contents of a STEP File.
- **Viewer Pro xxx** for extended viewing capabilities in a particular area, e.g. for 3D or PCB/PCA.
- **Editor xxx** for editing capabilities in some area xxx.
- **Import xxx** for being able to import files of format/type xxx.
- **Export xxx** for being able to export files in some other format.

This help documentation will cover only those features that are currently installed on your PC.

Free annual licenses are available for the features **IDA-STEP Framework** and **Viewer Basic**. They don't need any special license-id to run on your computer. For all other features a license-id is needed which

consists of 5 by 5 characters separated by a "-". They look like this:

XXXXX-XXXXX-XXXXX-XXXXX-XXXXX

Features may be licensed for an unlimited time or for a restricted time; e.g. for one year. When licensed for a restricted time the actual termination day of each feature is listed in the **Registration, Install and Update** dialogue, available via the **Help** menu. Also some licenses are constrained for non-commercial use only. If this is the case it is indicated in the title bar of IDA-STEP.

The installation of IDA-STEP is quite easy, but requires a good Internet connection:

- you first download the **IDA-STEP Framework** installer from www.ida-step.net and install it on your computer.
- next you start IDA-STEP and then you have to decide whether to use the free viewer or to enter a license-id.
- depending on what you have chosen all needed software is then loaded into your PC and immediately afterwards you can start working
- whenever a new update becomes available, or you have upgraded your license for additional features or you want to switch from the free license to a commercial one simply run the **Register, Install and Update** dialogue again.

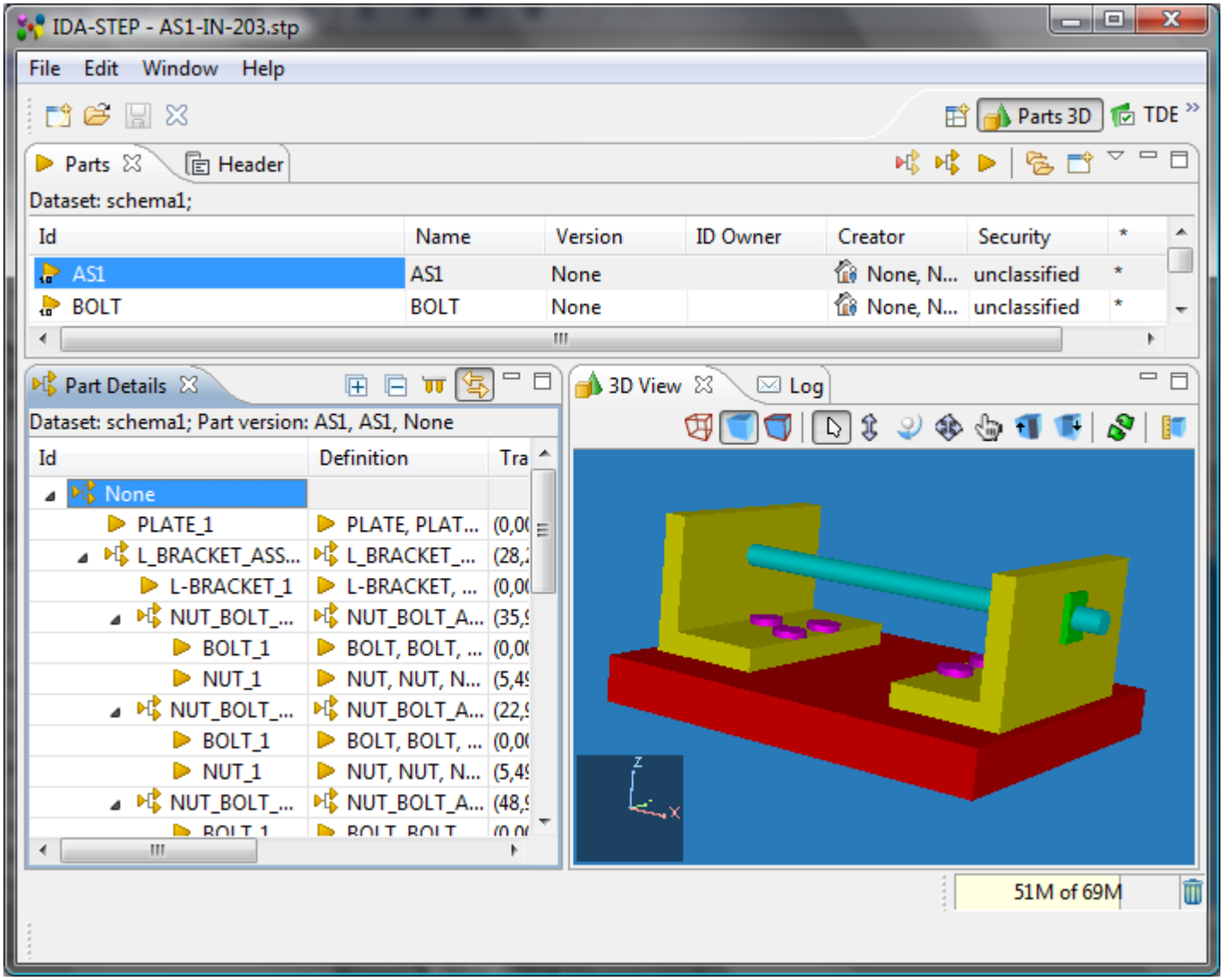

Figure 1. IDA-STEP Viewer Basic window

The internal data model used by IDA-STEP is called XIM which stands for Extended Integrated Model. It is a vendor specific combination and arrangement of the data models in STEP Application Modules called Application Reference Model (ARM) and Modular Integrated Model (MIM), the integrated resources and IDA-STEP specific extension.

IDA-STEP is able to process data from various applications and based on different STEP Application Protocols (APs). Most APs are supported during Import and for Export the focus is on:

- AP203 edition 2, Configuration controlled 3D designs of mechanical parts and assemblies;
- AP210 Electronic assembly, interconnect and packaging design;
- AP214 Core data for automotive mechanical design processes;
- AP239 Product life cycle support.

Data according the XIM model is stored in binary files with the extension ".xim". XIM files can be loaded significantly faster than standard STEP Files with the extension ".stp" and can also contain additional information such as rendered 3D models for fast viewing.

To top

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2. About

The **About IDA-STEP** dialogue provides legal information about IDA-STEP and details about the installed version and configuration. (see Figure 1). To open this dialog, select in the main menu bar **Help -> About IDA-STEP**.

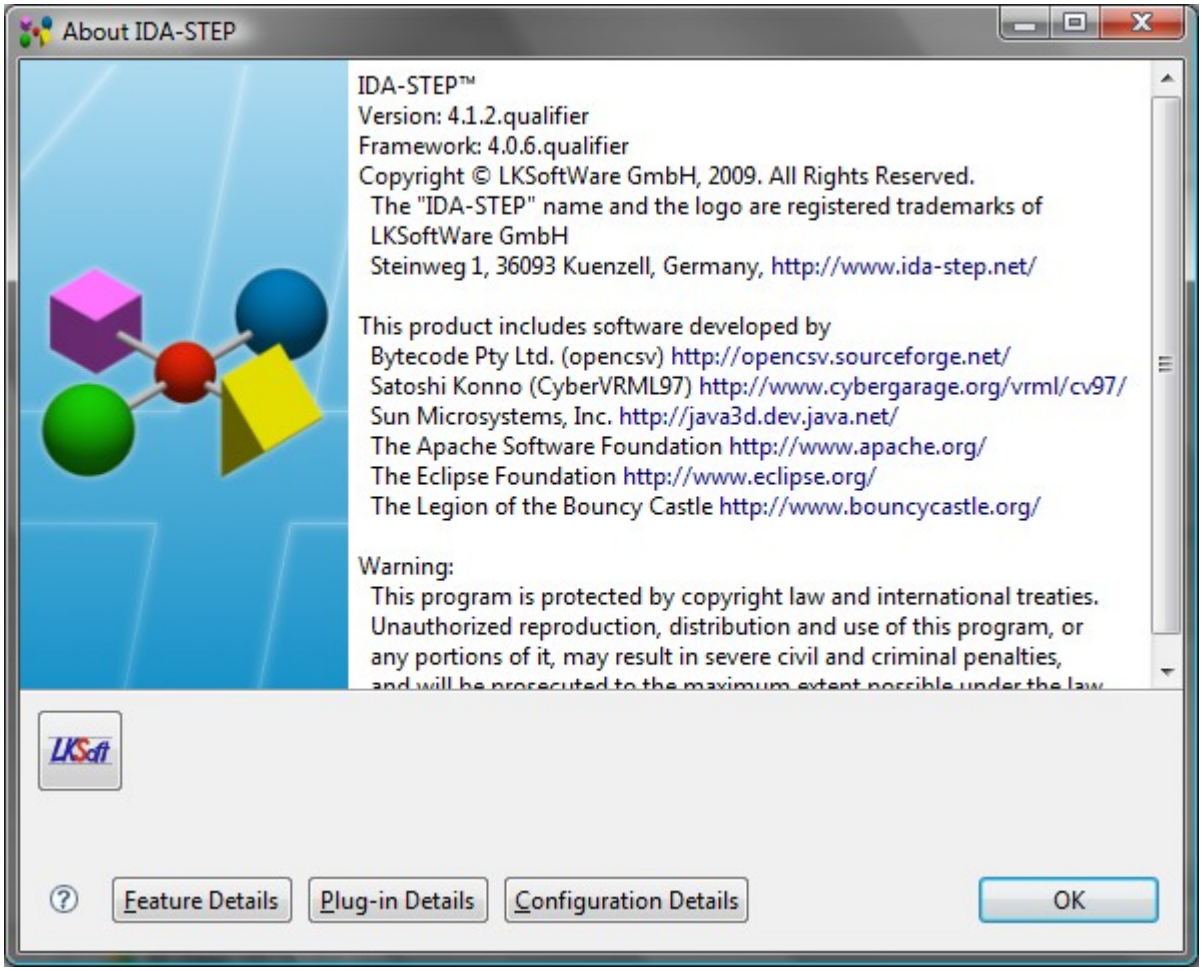

Figure 1. About IDA-STEP dialogue page

When pressing the Feature details button, the About IDA-STEP Features dialog is opened. It displays table, that lists the installed features. The columns are:

- **Provider** - the organization, which created the feature. In most cases this is LKSoftWare GmbH.

- **Feature Name** - the group of words by which the feature is known. The same names are used in the online shop.
- **Version** - each features has it's own version, that is listed here. The IDA-STEP installation system ensures that only mating versions of different fetaures are installed together.
- **Feature Id** - the identifier under which the feature is known by the underlying Eclipse system.

Note. The Register, install and Update page also lists the licensed feature together with optional expiration date and installation status.

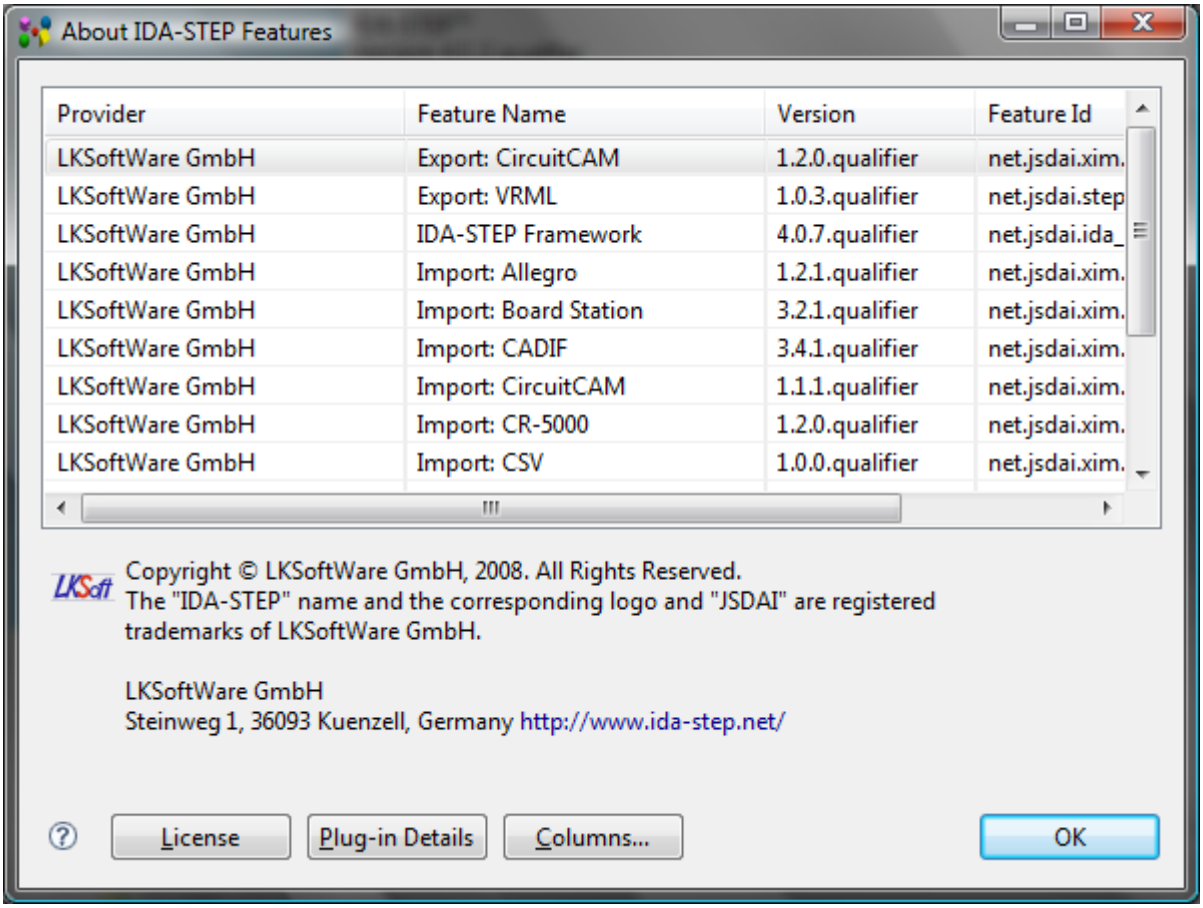

Figure 2. About IDA-STEP Features page

To top

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3. Register, Update and Install

For the installation of IDA-STEP you have to choose to either go for a free license for the Viewer Basic or to go for a named license that you've either purchased or given otherwise from LKSoft.
If the software was already installed before, you may just want to get an update or an upgrade of your license. An *update* is getting the most actual version of the licensed features. An *upgrade* means to enhance the license for new features. So your choices are:

- **New free license** - select this radio button, if you want to get free licensed features.
- **New license ID** - select this radio button, if you want to enter a new license.
- **Update current license** - select this radio button to update already installed features according to the indicated license-id.

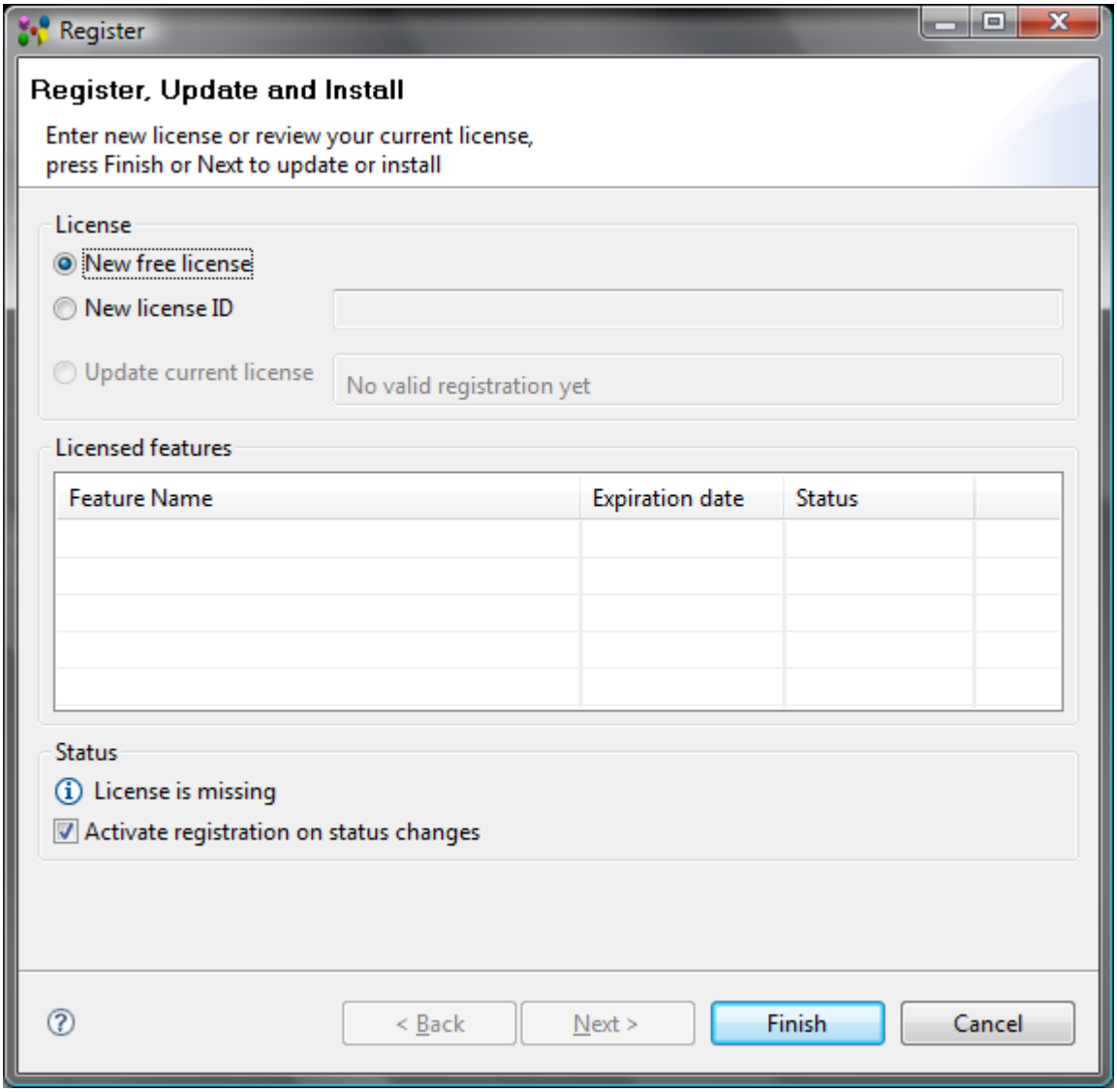

Figure 1. Register, Install and Update wizard dialogue window

The other controls in the **Register, Update and Install** wizard page are:

- **Licensed features** - presents the list of licensed features with their expiration date:
  - **Feature name** - the group of words by which the feature is known. The same names are used in the online shop. Here all licensed features are listed, independent of their installation status.
  - **Expiration date** - depending on the license condition, a particular feature may expire at a given date or is timely unlimited. This information is given here. Expired licensed can be updated via online shop.
  - **Status** - the installation status of the particular license. The possible states are:
    - **OK** - the licensed feature is installed and works properly,
    - **Not installed** - the licensed feature is not installed,
    - **Expired** - the license of the feature has expired, but the feature is still installed.
- Overall **Status** - presents the status of the license as a whole:
  - The possible states are:
    - License is valid.
    - License is missing.
    - License is invalid. Renew your license.
    - License is not valid for this computer. Update your license or ask for another one.
    - Some features are missing or are not installed correctly. Try updating IDA-STEP.
    - Some feature licenses have expired. Update your license.
    - Some feature licenses will expire in 30 or less days, consider updating your license.
  - **Activate registration on status changes** option - if this option is selected, the **Register,**

Update and Install window is pop upped every time you start IDA-STEP when the state of the license is changed:

You can switch this option *Off*, by unselecting the check box. The **Registration and Licensing Status** dialogue window appears on the screen (see Figure 2), where you can select whether to disable the future activation of the registration on status changes or not. If you agree with the affirmation, press **Yes** button.

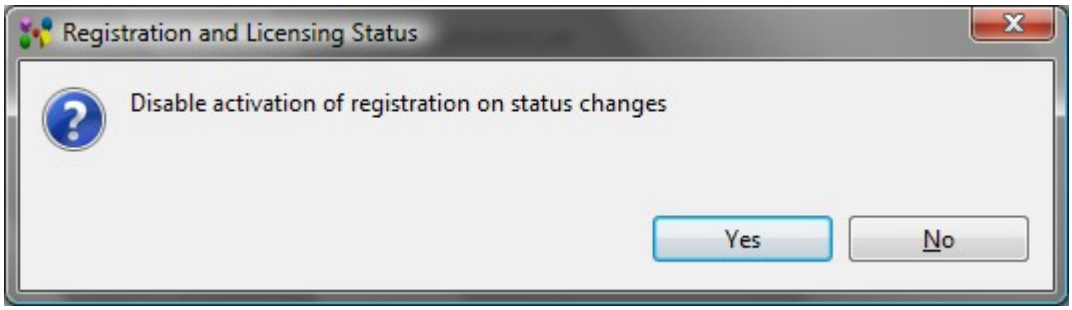

Figure 2. Registration and Licensing Status dialogue window, proposing to disable activation of registration on status changes

If you want to enable the activation of a registration on status changes in the future - select the **Activate registration on status changes** check box. The **Registration and Licensing Status** dialogue window appears on the screen with the message "Enable activation of registration on status changes" (see Figure 3). If you agree with the affirmation, press **Yes** button.

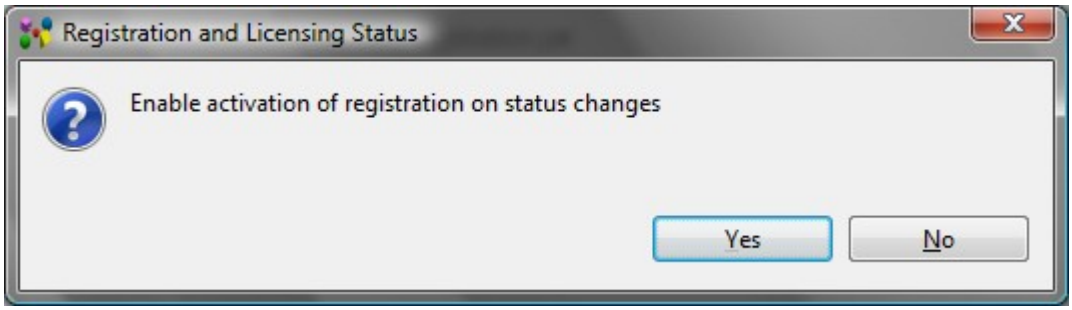

Figure 3. Registration and Licensing Status dialogue window, proposing to enable activation of registration on status changes

Register

After **IDA-STEP Framework** is started for the first time, the **Register, Update and Install** wizard pop ups on the screen. If you want to update license-id, or update installed features select the **Register, Update and Install** menu item in the **Help** menu.

- Select **New free license** radio button to get free IDA-STEP Viewer (see Figure 1) and press **Finish** button. The IDA-STEP Viewer will be installed.
  Note. To install Free IDA-STEP viewer, the current computer should be with the Internet access.
  Or
- Select **New license ID** radio button and enter a new license-id into the text field (see Figure 4).
- Press **Finish** button to install new licensed features or press **Next** button to open the **Registration and Installation Method** dialogue to choose how to register and install IDA-STEP.
  Note. The **Next** button is disabled for the free license installation.

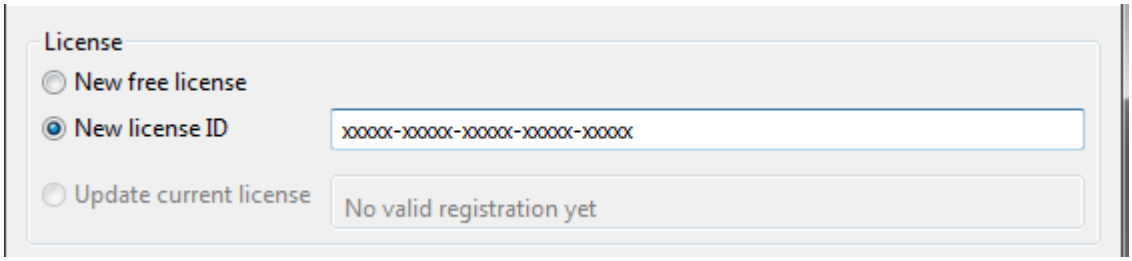

Figure 4. Register new license

To top

Update

If you want update already licensed features:

- Select the **Update current license** radio button in the **License** layout (see Figure 1).
- The licensed features are indicated in the **Licensed features** table.
- Click **Finish** button or press **Next** button to open the **Registration and Installation Method** dialogue to choose how to register and install IDA-STEP.

If no new features or updates are available for the currently activated license, the particular message window appears on the screen (see Figure 6).

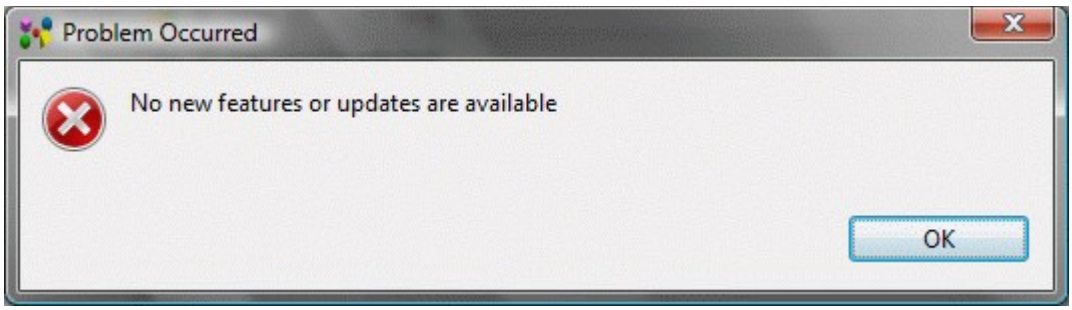

Figure 6. Problem Occurred message window

To top

Registration and Installation Method

The user can choose how to register a new license or update licensed features - online or offline - in the **Registration and Installation Method** page (see Figure 7).

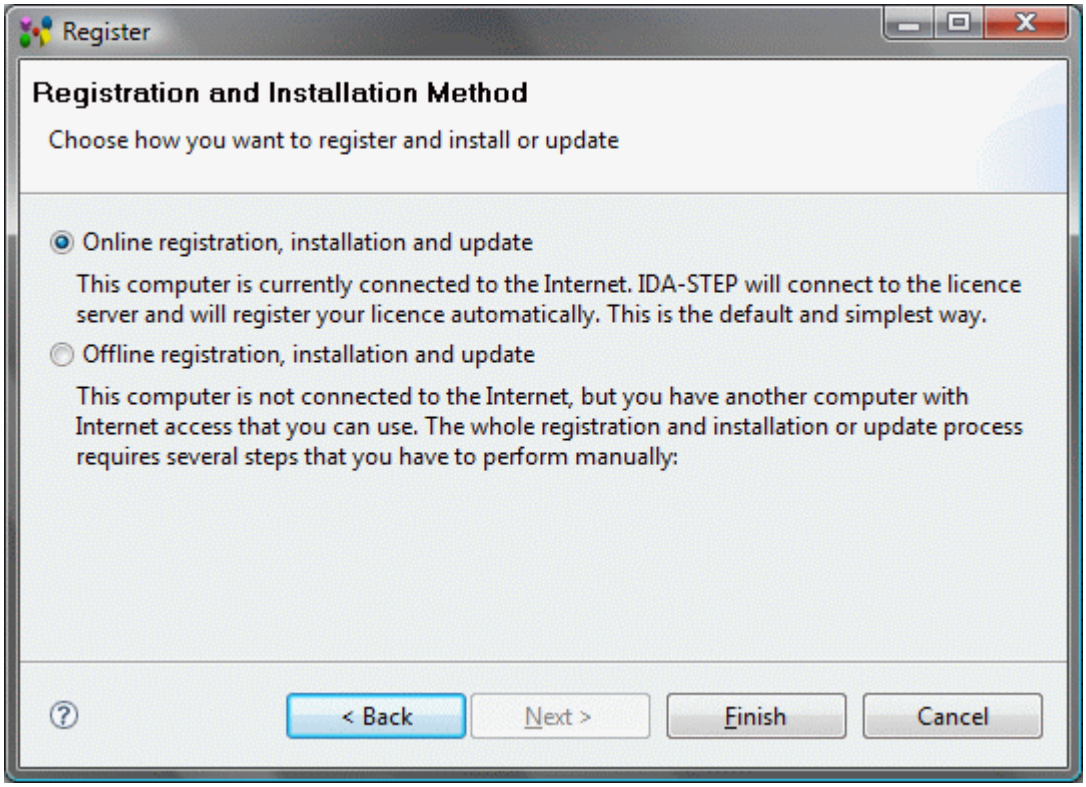

Figure 7. Registrations and Installation Methods page

Online registration and installation

The default registration and installation method is **Online registration and installation**. This option must be selected if your computer is currently connected to the Internet. IDA-STEP will connect to the license server and will register your license automatically. This is the default and simplest way.
If any problem occurred while the registration and installation, please see Troubleshooting for details.

Offline registration, installation and update

If your computer is not connected to the Internet, you can use offline installation method.

- Select the **Offline registration, installation and update** radio button in the **Registration and Installation Method** page (see Figure 7).
- Press **Next** button. The **Offline Registration and Installation** page appears on the screen to choose the action for offline registration, installation or update (see Figure 8):

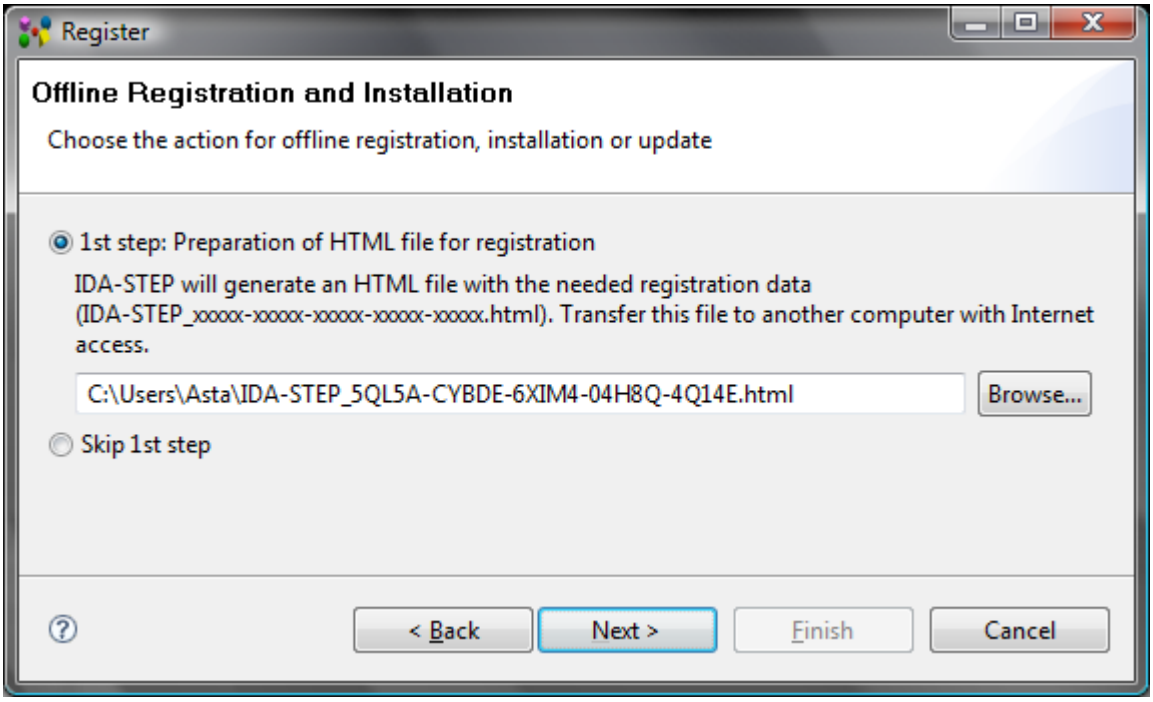

Figure 8. Offline Registration and Installation page (displayed license here: 5QL5A-CYBDE-6XIM4-04H8Q-4Q14E)

- Select the **1st Step: Preparation of HTML file for registration** radio button. IDA-STEP will generate an HTML file with the needed registration data (IDA-STEP_xxxxx-xxxxx-xxxxx-xxxxx-xxxxx.html). Transfer this HTML file to another computer with internet access.
  Select a location of the file to be generated in your file system by using **Browse...** button next to the text field.
  Or
- Select the **Skip 1st step** radio button if the HTML file is already created during previous registration or update sessions.

- Press **Next** button to go to the next page of the wizard (see Figure 9). After the **Next** button is pressed the HTML file is generated and located in the selected place (in our example 5QL5A-CYBDE-6XIM4-04H8Q-4Q14E.html).

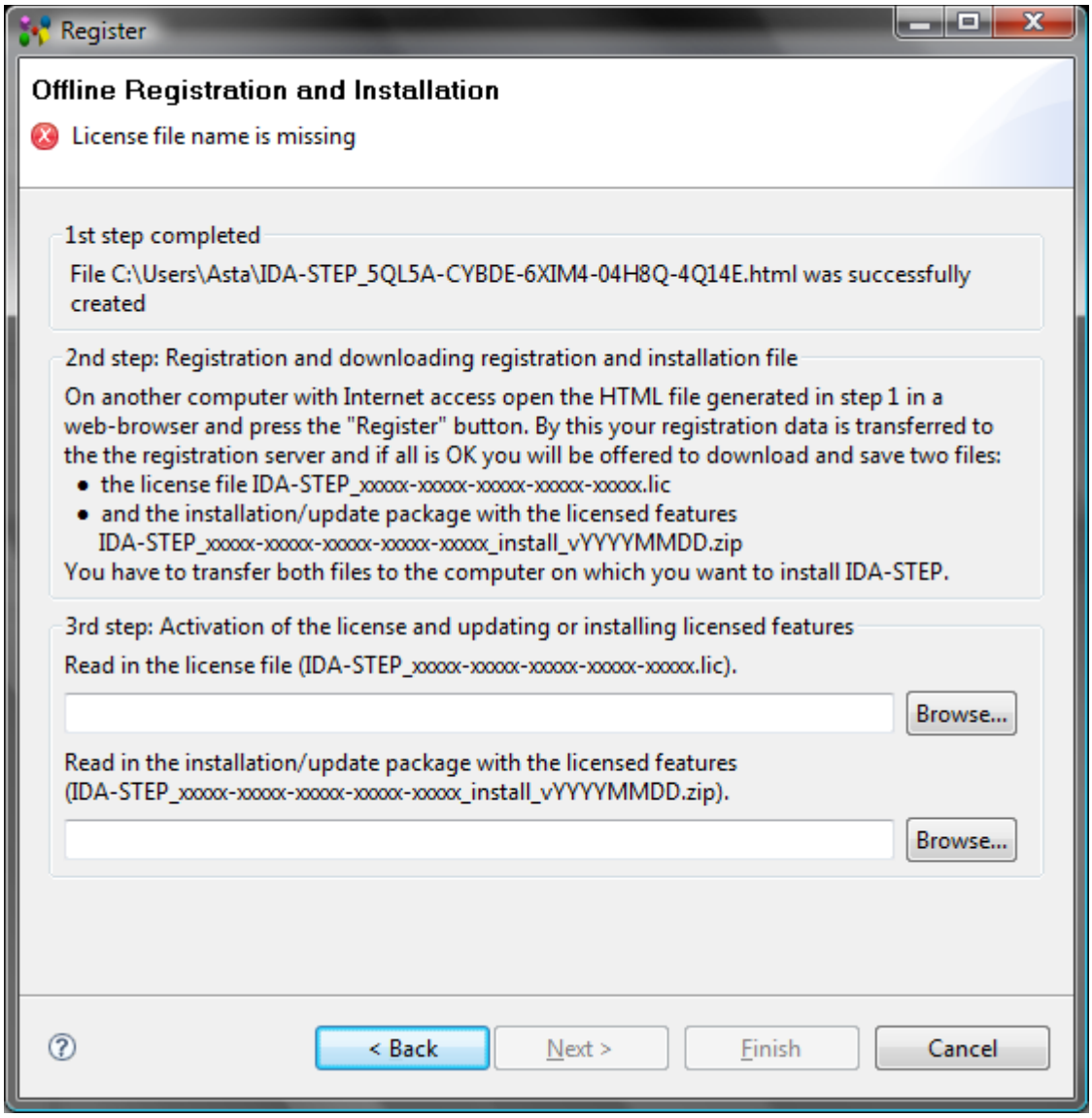

Figure 9. Offline Registration and Installation wizard page to select license file and installation/update package

- **1st step completed** layout presents information, that HTML file is successfully created. **Note.** This layout is skipped, if the **Skip 1st step** radio button was selected in the previous page.

- **2nd step: Registration and downloading registration and installation file** layout
  On another computer with Internet access access open the HTML file generated in step 1 in a web-browser and press the **Register** button.By this your registration data is transferred to the the registration server and if all is **OK** you will be offered to download and save two files:
  - the license file IDA-STEP_xxxxx-xxxxx-xxxxx-xxxxx-xxxxx.lic
  - and the installation/update package with the licensed features IDA-STEP_xxxxx-xxxxx-xxxxx-xxxxx-xxxxx_install_vYYYYMMDD.zip.

You have to transfer both files to the computer on which you want to install IDA-STEP.

Meanwhile you will get a license file and installation/update package, you can close the wizard page by pressing **Cancel** button. Later you can open it by selecting **Help -> Register, Update and Install** menu item in the main menu bar and using **Next** button go to the needed page..

After you receive the license file (.lic), you can activate your license on your computer, where IDA-STEP should be installed.

- **3rd step: Activation of the license and updating or installing licensed feature** layout:
  - Specify the path to the license file (IDA-STEP_xxxxx-xxxxx-xxxxx-xxxxx-xxxxx.lic) by

using **Browse** button next to the **Read in the license file (IDA-STEP_xxxxx-xxxxx-xxxxx-xxxxx-xxxxx.lic)** text field (see Figure 10).

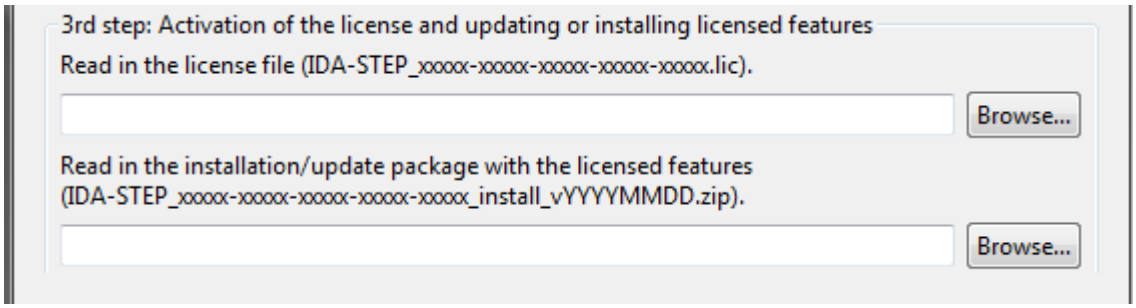

Figure 10. Installation and Registration 3rd step

- Specify the path to the installation/update package IDA-STEP_xxxxx-xxxxx-xxxxx-xxxxx-xxxxx_install_vYYYYMMDD.zip by using **Browse** button next to the **Read in the installation/update package with the licensed features (IDA-STEP_xxxxx-xxxxx-xxxxx-xxxxx-xxxxx_install_vYYYYMMDD.zip)** text field.

- Press **Finish** button to activate the license and update/install licensed features.

To top

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4. Preferences

The **Preferences** dialog is the dialog used to set user preferences (see Figure 1). The **Preferences** dialog pages can be searched using the filter function. To filter by matching the page title, simply type the name of the page you are seeking and the available pages will be presented below. The filter also searches on keywords such as appearance and java. The history controls allow you to navigate through previously viewed pages. To step back or forward several pages at a time, click the drop down arrow and a list of the most recently viewed preference pages will appear.

The **Preferences** dialog can be found from the main workbench **Window** menu under **Window > Preferences**. Preference pages contributed by plug-ins will be found in this dialog.

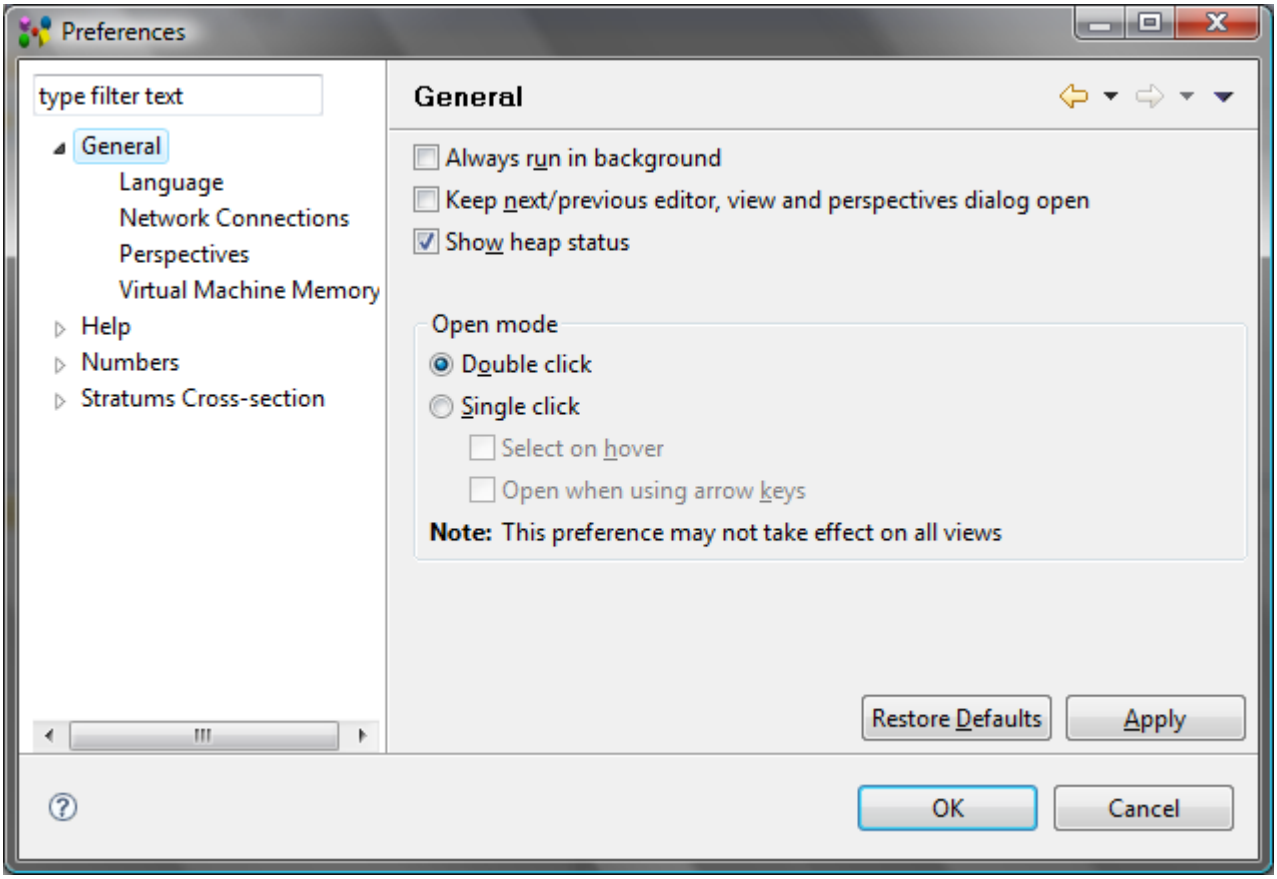

Figure 1. Preferences page

The IDA-STEP preferences can be exported in EPF (preferences file) format and imported into some other installation of IDA-STEP (reference Importing and Exporting Preferences).

To top

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4.1. 3D View

The **3D View** preference page defines settings for the **3D View** page (see Figure 1):

- Front side (right, up) combo box,
- Projection type,
- Zooming direction,
- Point size for dimension inspection,
- Point size for construction geometry,
- Precision used by the system,
- Coordinate system size.

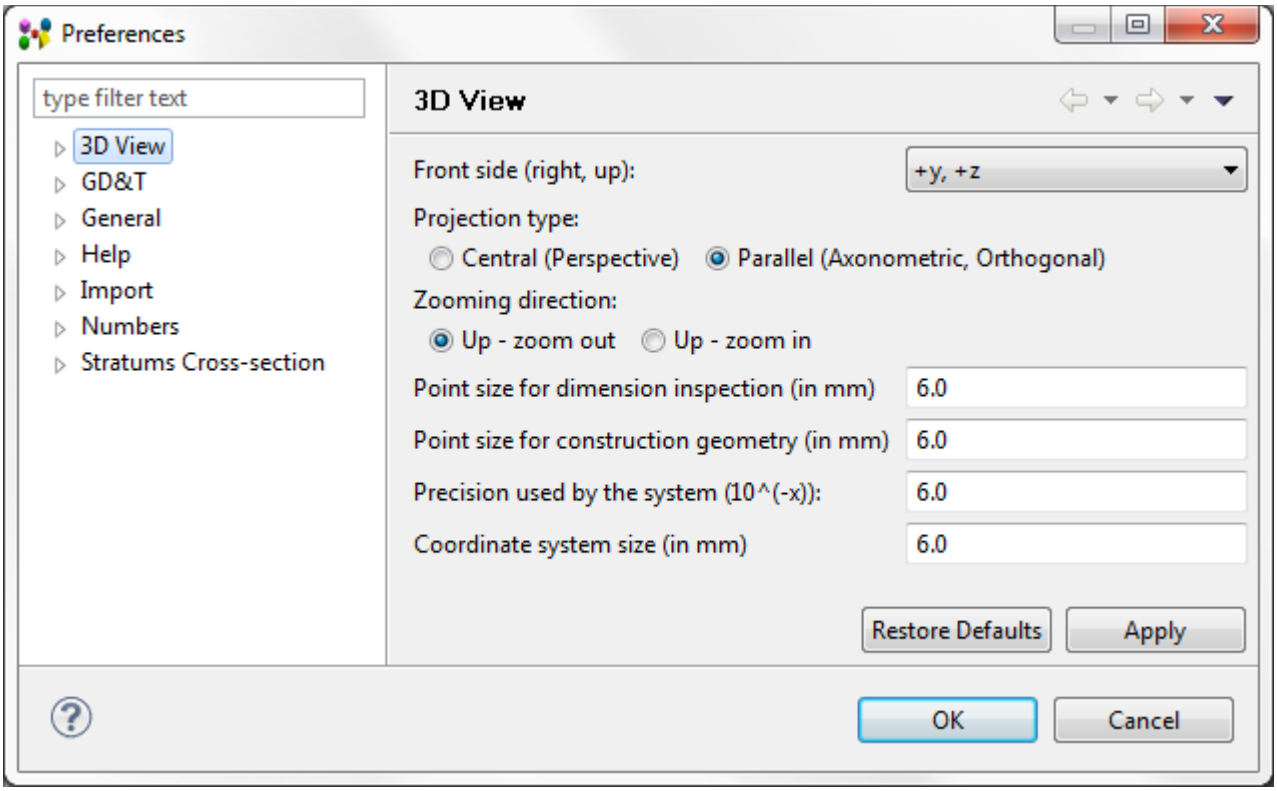

Figure 1. 3D View preferences page

Front side (right, up)

The *Front side* is a default orientation in the **3D View**. All other predefined orientations, such as Top, Bottom, Right and etc. are derived from the Front orientation data (e.g. Top is 90 degrees rotation about axis, which points to the right, etc.).
The **Front side (right, up)** combo box lists all 24 possible combinations. They are indicated by which axis go to the right and which axis go to the top, e.g. "+x,+z", meaning that the positive x axis points to the right and the positive z axis points to the top (see Figure 2).

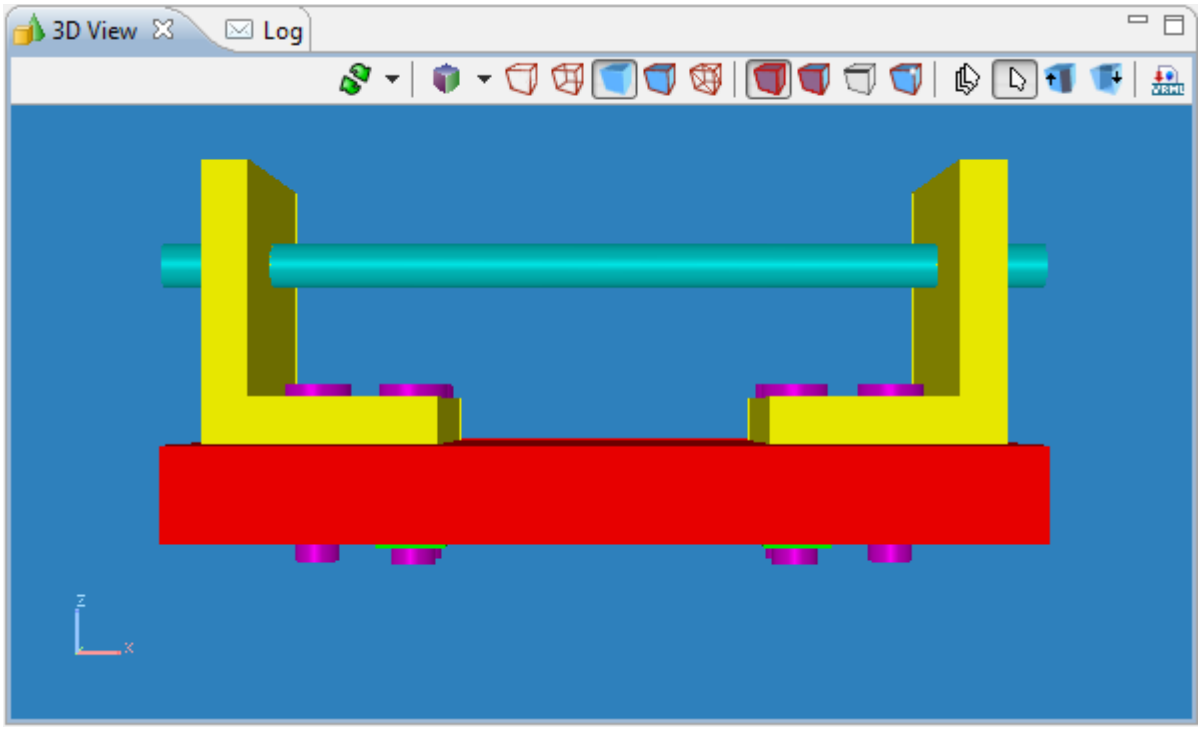

Figure 2. 3D object Front side view, right axis X, up axis Z (+x, +z)

To restore the default Front side orientations, press the **Restore Defaults** button on the bottom section of

the preferences page.

Projection type

The *Projection type* determines whether geometry in the **3D View** is displayed using an orthogonal or perspective projection. To switch between projection types select one of radio buttons:

- **Central (Perspective)** - this projection incorporates foreshortening (the size of an object's dimensions along the line of sight are relatively shorter than dimensions across the line of sight) (see Figure 3). Parallel lines in the data may not appear parallel on screen.
- **Parallel (Axonometric, Orthogonal)** - this projection maintains the correct relative dimensions of graphics objects with regard to the distance a given point is from the viewer and draws parallel lines in the data parallel on the screen (see Figure 4).

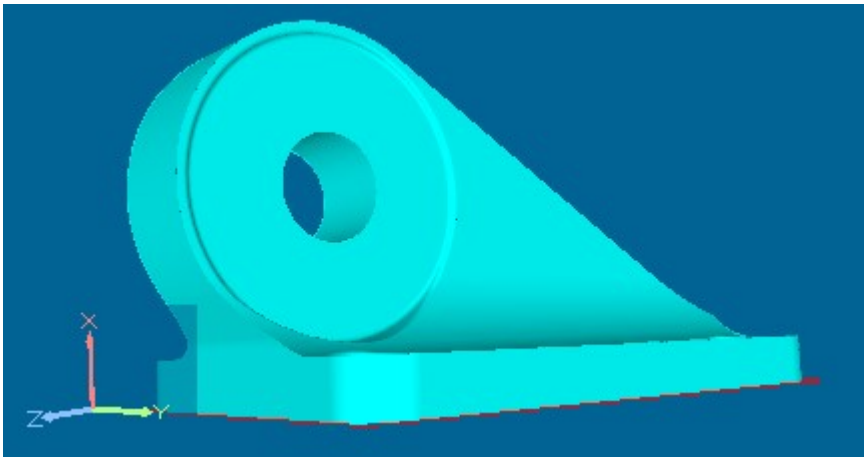

Figure 3. Geometry presentation, when Central (Perspective) projection type is selected

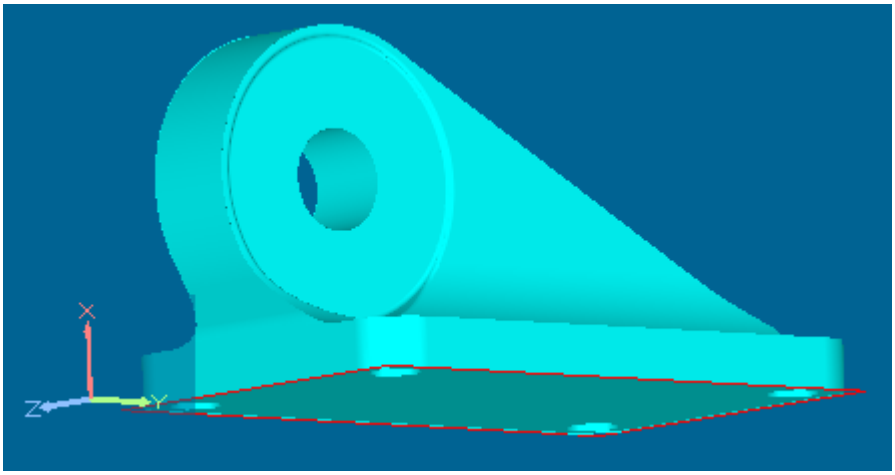

Figure 4. Geometry presentation, when Parallel (Axonometric, Orthogonal) projection type is selected

Zooming direction

The default default zoom direction in the **3D View**:

- Up - zoom out. Scroll mouse wheel or press CTRL + move mouse towards you to zoom out and away from you to zoom in.
- Up - zoom in . Scroll mouse wheel or press CTRL + move mouse towards you to zoom in and away from you to zoom out.

Point size for dimension inspection

The default point size for dimension inspection points.The default size value is 6 mm.

Point size for construction geometry

The default point size for construction geometry points. The default size value is 6 mm.

Precision used by the system

The default precision used by the system (10^(-x)) in millimetres. The default size value is 6 mm.

Coordinate system size

Size of coordinate system for annotation plane. The default size value is 6 mm. **Note.** Axes are visible for an annotation plane only when it is selected.

See also Display Styles.

To top

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4.1.1. Display Styles

For displaying 3D geometry models by default IDA-STEP uses the default style schema, which is present in STEP data. A user can create a new default style and make it active for a geometry model in the **3D View**, remove the redundant or incorrect style, modify a set of properties of the selected style, such as background, ambient light, diffuse light colours, element/component colours, transparency, highlighting and dark-lighting colours. The **Display Styles** settings are stored in a preferences file.
The **Display Styles** preferences configuration page is composed of several sections:

- Top section of the window - to select, create, remove, make active a particular style;
- **General tab** - to set background, ambient light and diffuse light colours (see General tabbed panel);
- **Element Colors** tab - change display preferences for a geometry model, construction geometry and annotation (see Element Colors tabbed panel);
- **Component Colors** tab - to change colour and transparency for a surface, colour for a curve and vertex point for an assembly component. Also highlighting and dark-lighting colours can be changed in this tab (see Component Colors tabbed panel).

Note. The **Display Style** default settings would be overridden by any available styling data in a styling model.

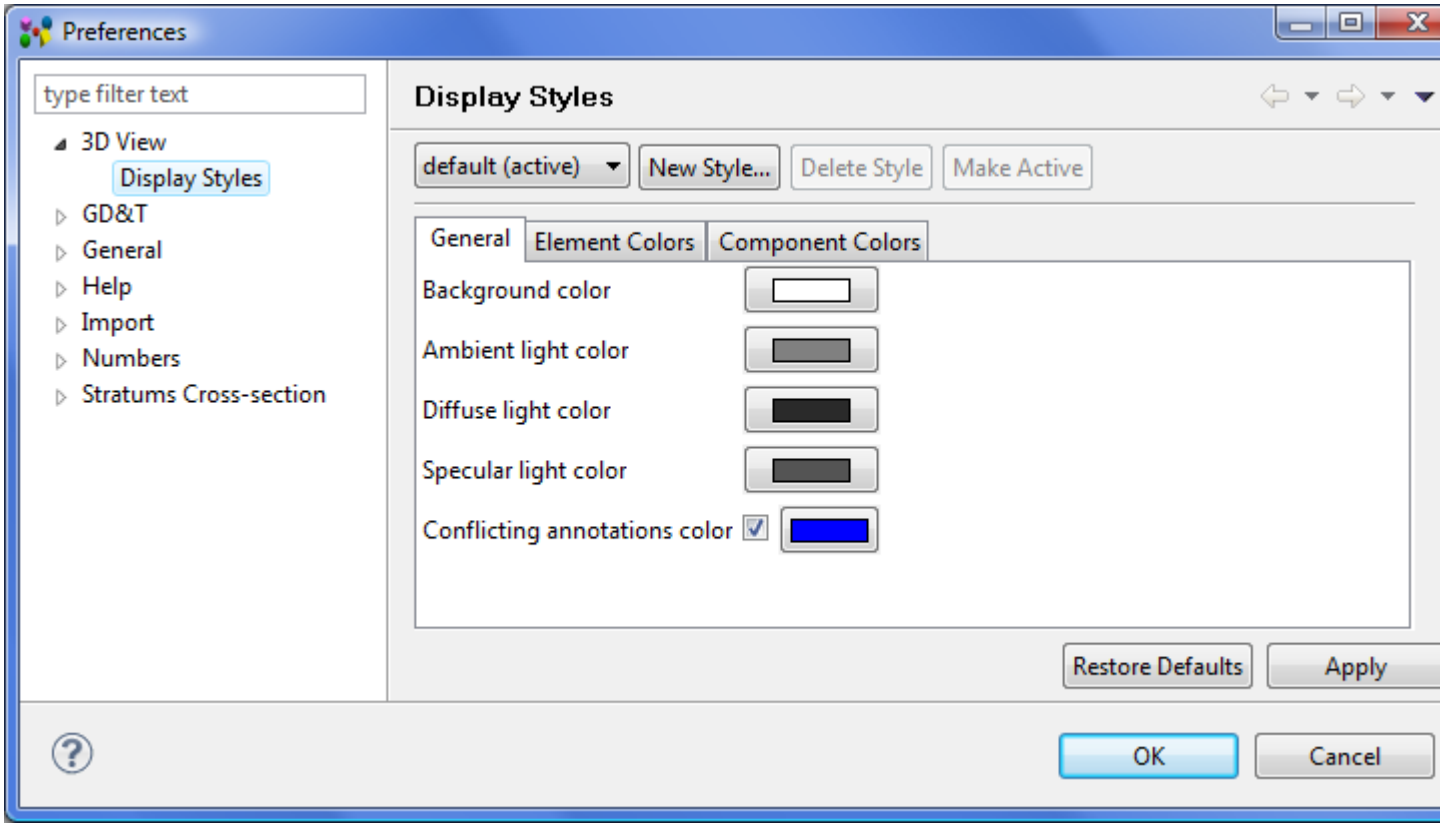


Figure 1. Display Styles preferences page

The top section of the **Display Styles** preferences page contains:

- Combo box - to select a style schema. The active schema is identified with the word 'active' in the brackets.
- **New Style** button - to create a new style schema;
- **Delete Style** button - to remove a redundant or incorrect style schema. The button is enabled if the selected schema is not active.
- **Make Active** button - to activate the selected schema. If the selected style schema is active, the button is disabled.

To select style schema:

- Press on the combo box arrow.
- Select an applicable item in the appeared drop down list.

You can find the same list of style schemas in the **Styling Preference**s list of the **3D View**.

To create new style schema:

- Press **New Style...** button. The **New Style** wizard window appears on the screen (see Figure 2).
- Enter a name of the style schema to be created into the text field.
- Press **OK** button.

The created style schema name appears in the combo box and in the **Styling Preference**s list of the **3D View**.

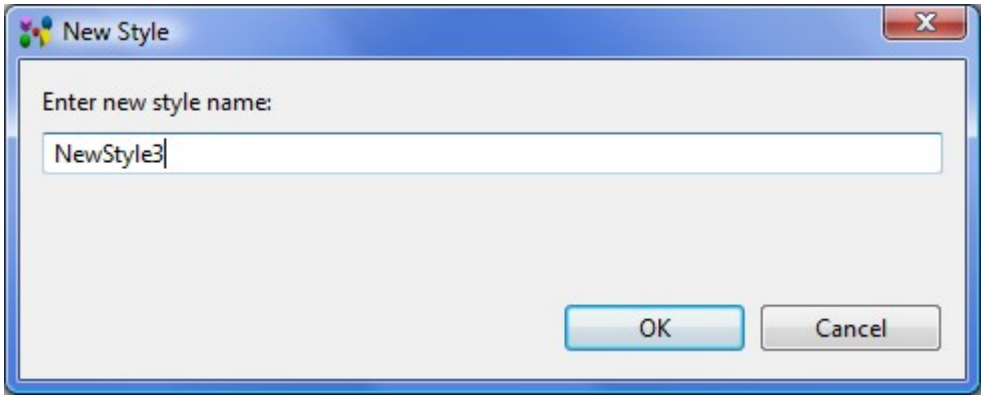


Figure 2. New Style wizard window

To remove style schema:

- Select a redundant or incorrect style schema in the combo box.
  Note. The style schema has to be not activated (without label '(active)').
- Press **Delete Style** button.

The style schema is removed from the combo box list and from the **Styling Preference**s list in the **3D View**.

To make style schema active:

- Select an applicable style schema in the combo box list.
  Note. The selected schema has to be not activated (without label '(active)').
- Press **Make Active** button.

The selected schema is activated (the label '(active)' appears next to the style schema name).

To top

General tabbed panel

The **General** tab (see Figure 1) allows a user to select colour for a:

- background,
- ambient light,
- diffuse light,
- specular light, and
- conflicting annotations color (if the option is checked).

Note. The background, ambient light, diffuse light properties are in the bounds of a currently selected style and not a global property.

Background color

The Background parameter defines the colour of the area around geometry in the **3D View**. The default background colour is white.

To change 3D View background color:

- Press on the coloured square button next to the **Background** label in the **Display Styles** preferences page and select an applicable colour in the appeared **Color** window. Confirm your selection by pressing **OK** button.
- Back to the preferences page. Press **Apply** or **OK** buttons to apply the selected colour.

The newly defined background colour is be displayed in the **3D View** (see Figure 3).

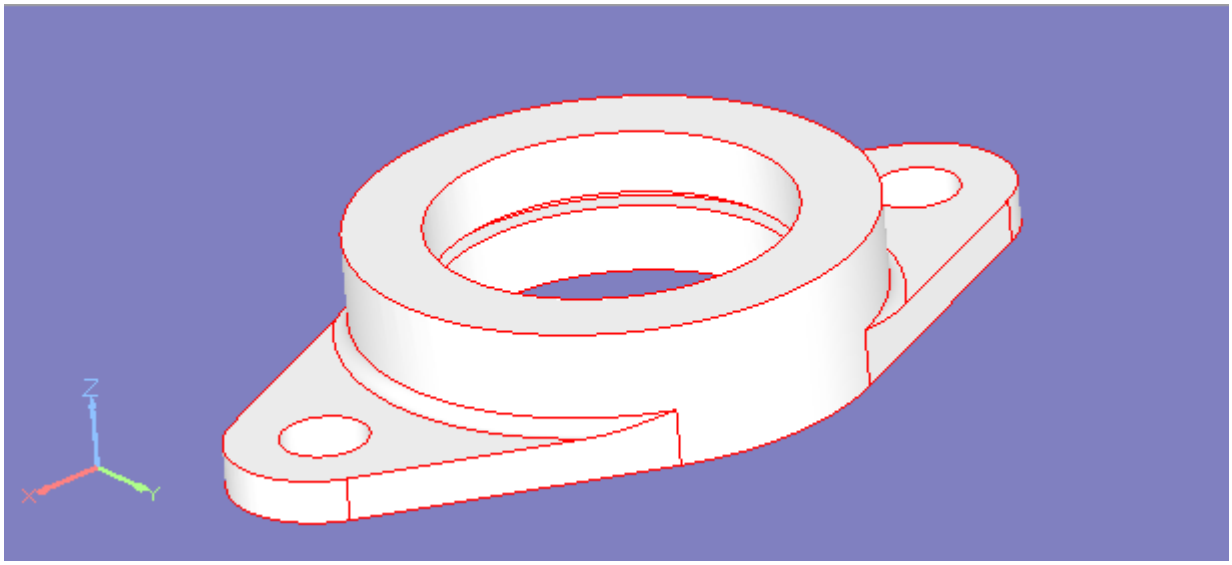


Figure 3. 3D View with changed background colour

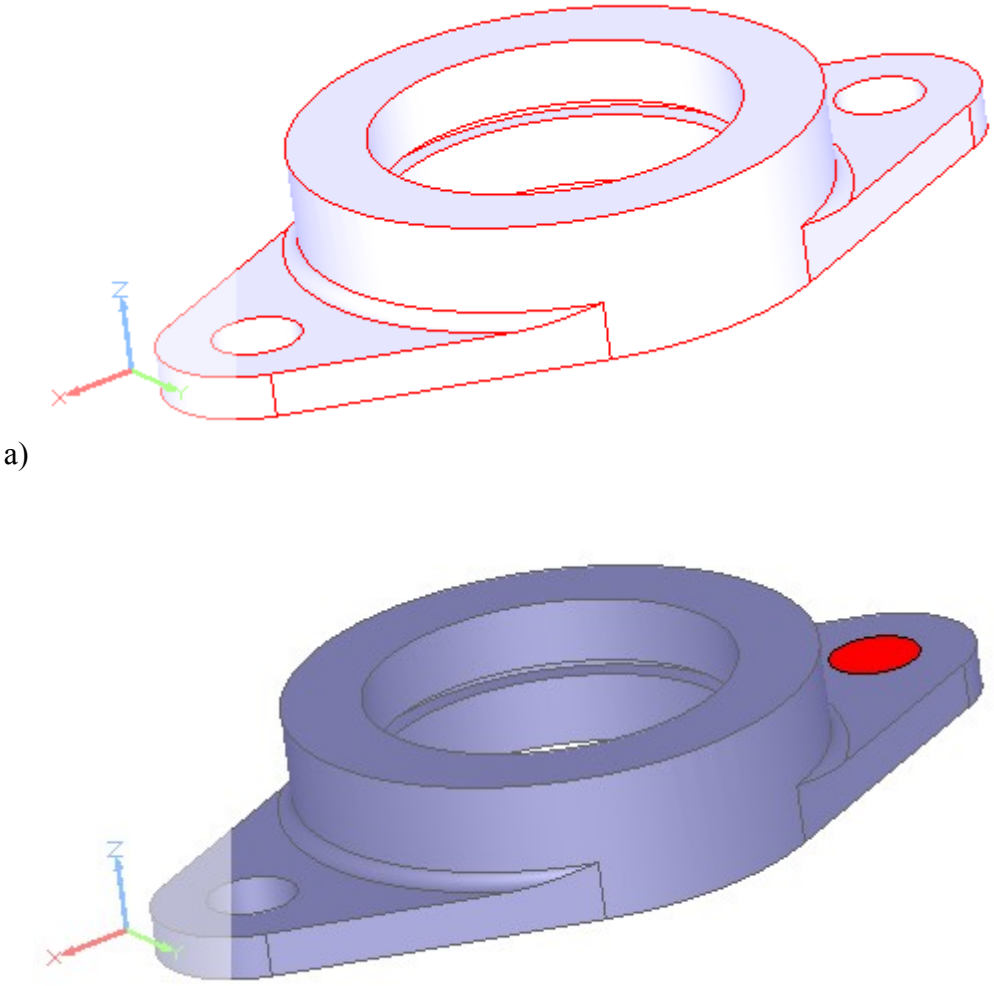
To restore the default Front side orientations and Background colour, press the **Restore Defaults** button on the bottom section of the preferences page.

Ambient light color

Ambient light is the general light that illuminates the entire geometry. It has no discernible source and no discernible direction. Ambient colour is the colour of an object where it is in shadow. This colour is what the object reflects when illuminated by ambient light rather than direct light.

To change ambient light color:

- Press on the coloured square button next to the **Ambient light color** label in the **Display Styles** preferences page. The **Color** dialogue window appears on the screen:
  - Press **Define Custom Colors** button.
  - Define a new intensity of the colour.
- Confirm your selection by pressing **OK** button.
- Back to the preferences page. Press **Apply** or **OK** buttons to apply the selected colour (see Figure 4).



a) b)

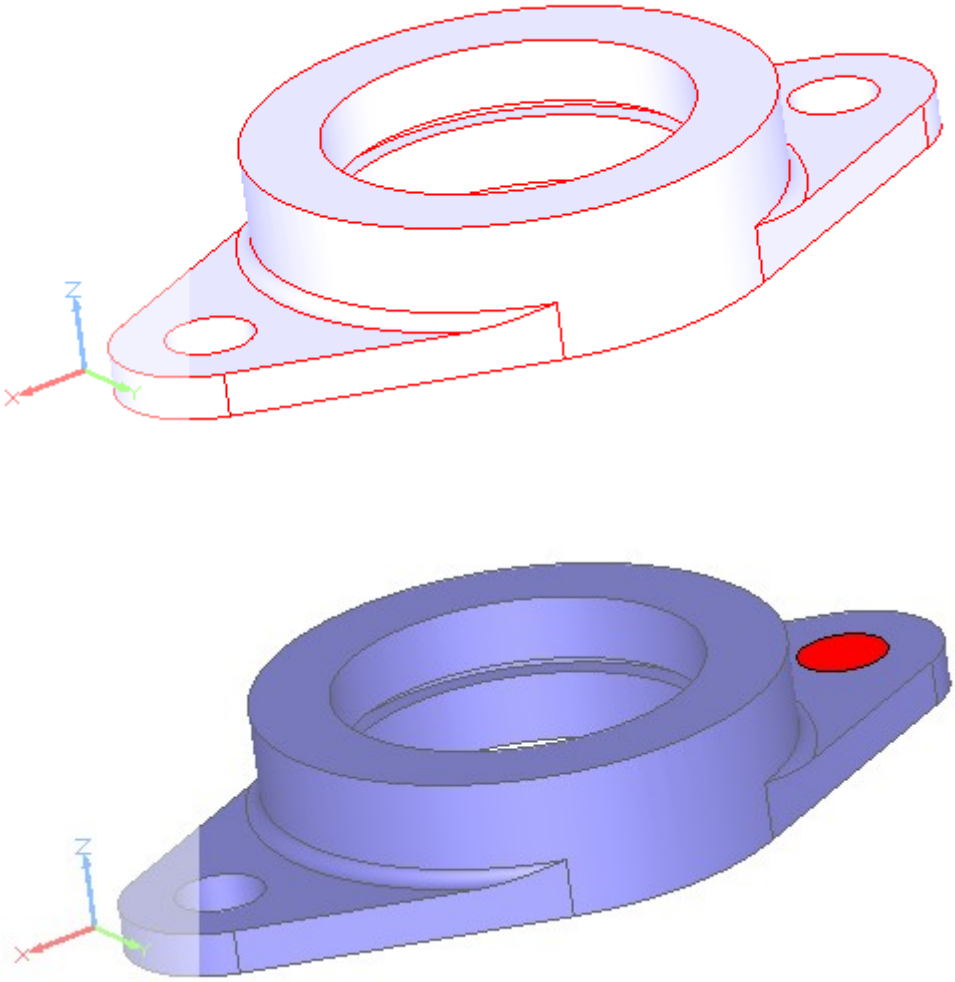
Figure 4. Applied ambient light colour 'lilac': a) normal, b) dark-lighted

Diffuse light color

Diffuse colour is the most instinctive meaning of the colour of an object. It is that essential colour that the object reveals under pure white light. It is perceived as the colour of the object itself rather than a reflection of the light.

To change diffuse light color:

- Press on the coloured square button next to the **Diffuse light color** label in the **Display Styles** preferences page and select an applicable colour (i.e. lilac) in the appeared **Color** window. Confirm your selection by pressing **OK** button.
- Back to the preferences page. Press **Apply** or **OK** buttons to apply the selected colour (see Figure 5).



a) b)

Figure 5. Diffuse light colour changed to 'lilac': a) normal, b) dark-lighted

Specular light color

Specular light - the light comes from a point source with specular lighting, it is reflected in the manner of a mirror where most of the light bounces off in a particular direction defined by the surface shape.

To change specular light color:

- Press on the coloured square button next to the **Specular light color** label in the **Display Styles** preferences page and select an applicable colour (i.e. lilac) in the appeared **Color** window. Confirm your selection by pressing **OK** button.
- Back to the preferences page. Press **Apply** or **OK** buttons to apply the selected colour (see Figure 6).

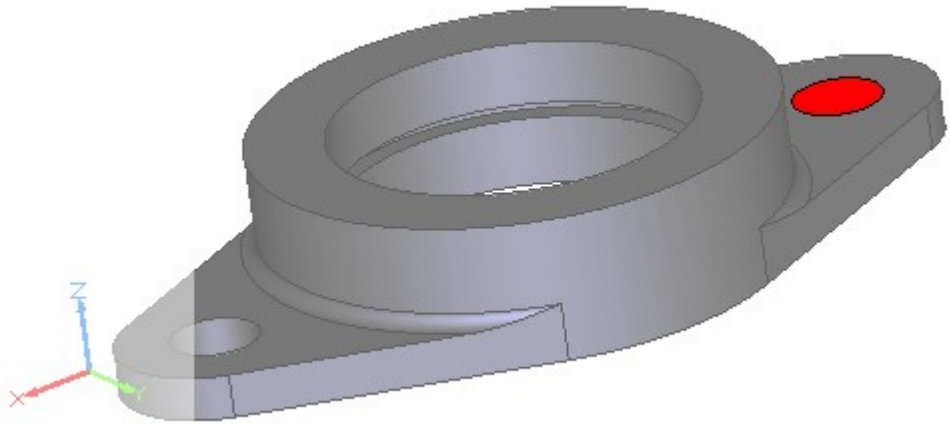


Figure 6. Specular light colour changed to 'lilac' (dark-lighted element)

Conflicting annotations color

Conflicting annotations color - the case, when the annotation colour in data is the same or very similar (differs ~5%) to the background colour. In this case the colour of annotations is changed to a different colour (default or specified by a user). A user may uncheck the **Conflicting annotations color** check mark, to apply the initial colour for annotations. **Note.** By default, this option is set *On* (see Figure 1) and a user is informed about the conflict in the **Log** view.

To change the default conflicting annotations colour:

- Press on the coloured square button next to the **Conflicting annotations color** label in the **Display Styles** preferences page and select an applicable colour in the appeared **Color** window. Confirm your selection by pressing **OK** button.
- Back to the preferences page. Press **Apply** or **OK** buttons to apply the selected colour.

To top

Element Colors tabbed panel

The **Element Colors** tabbed panel controls the default display preferences for a element, which is not an assembly component, and is presented as a single element in the **3D View** (see Figure 7).

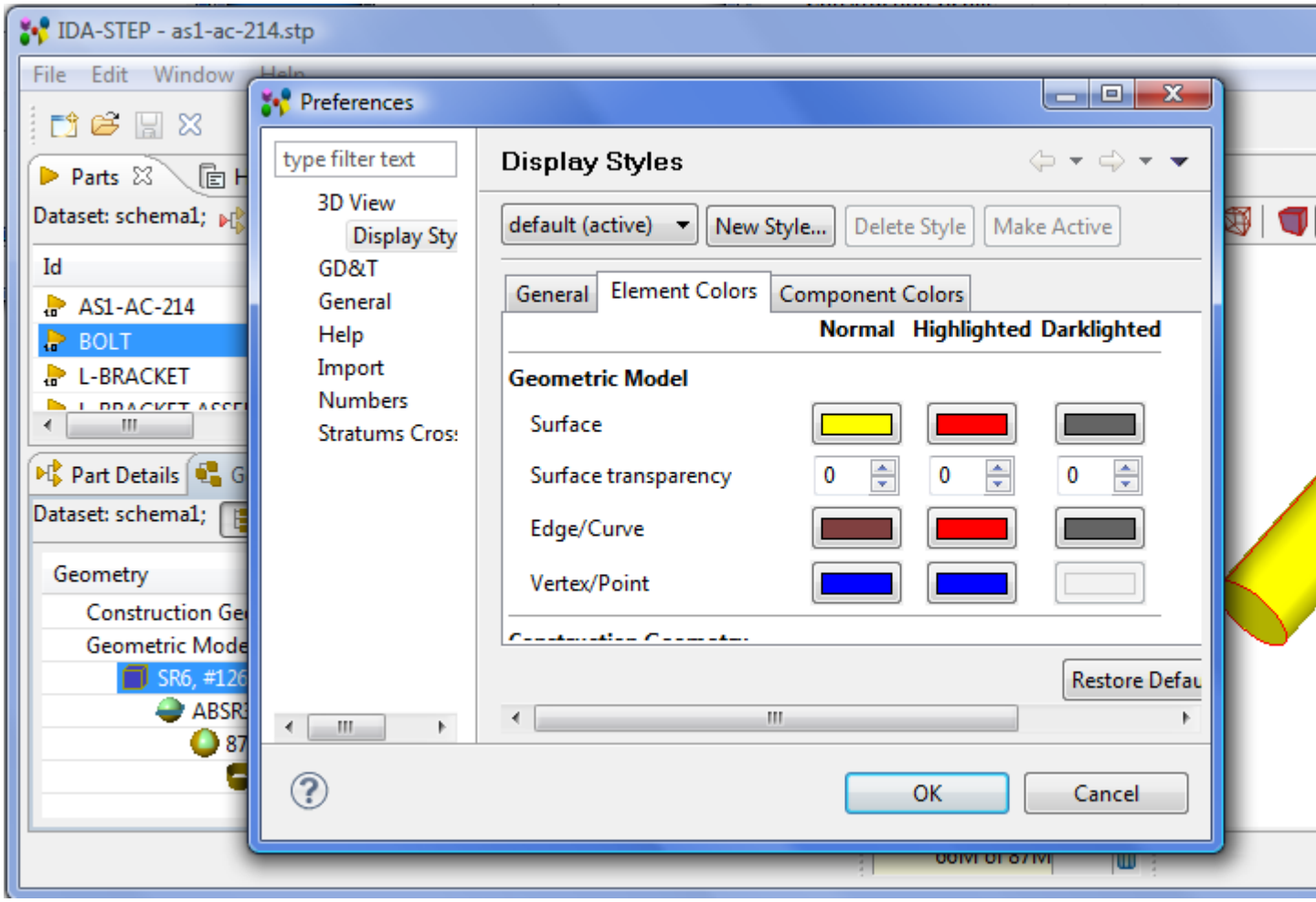


Figure 7. Element Colors tab

Find below the list of configurable default properties:

- Geometric Model:
  - **Surface**
  - **Surface transparency**
  - **Edge/Curve**
  - **Vertex/Point**
- Construction Geometry:
  - **Surface**
  - **Surface transparency**
  - **Edge/Curve**
  - **Vertex/Point**
- Annotation:
  - **Plane**
  - **Plane transparency**
  - **Text**
  - **Curve**
  - **Dimension curve**
  - **Leader curve**
  - **Projection curve**
  - **Fill area**
  - **Point**
  - **Symbol**
  - **Terminator symbol**
  - **Dimension curve terminator**
  - **Leader terminator**
  - **Subfigure**

- **Table**

Columns of the table:

- **Normal** - properties for an element in a normal state - not selected, not darkligthed.
- **Highlighted** - an element properties after it is selected/highlighted.
- **Darklighted** - a colour of an element, when other component in an assembly is selected/highlighted.

To change colour of an element:

- Press on a coloured square and select a colour in the appeared **Color** dialogue window.
- Press **OK** button to confirm the selection.

To change transparency level for a surface:

- Enter a number value into the field.
  Or
- Use up or down arrow next to the text field to increase or decrease number value.

To top

Component Colors tabbed panel

The **Component Colors** tabbed panel presents possibility to change display (colour, transparency) preferences for an assembly components (see Figure 8).

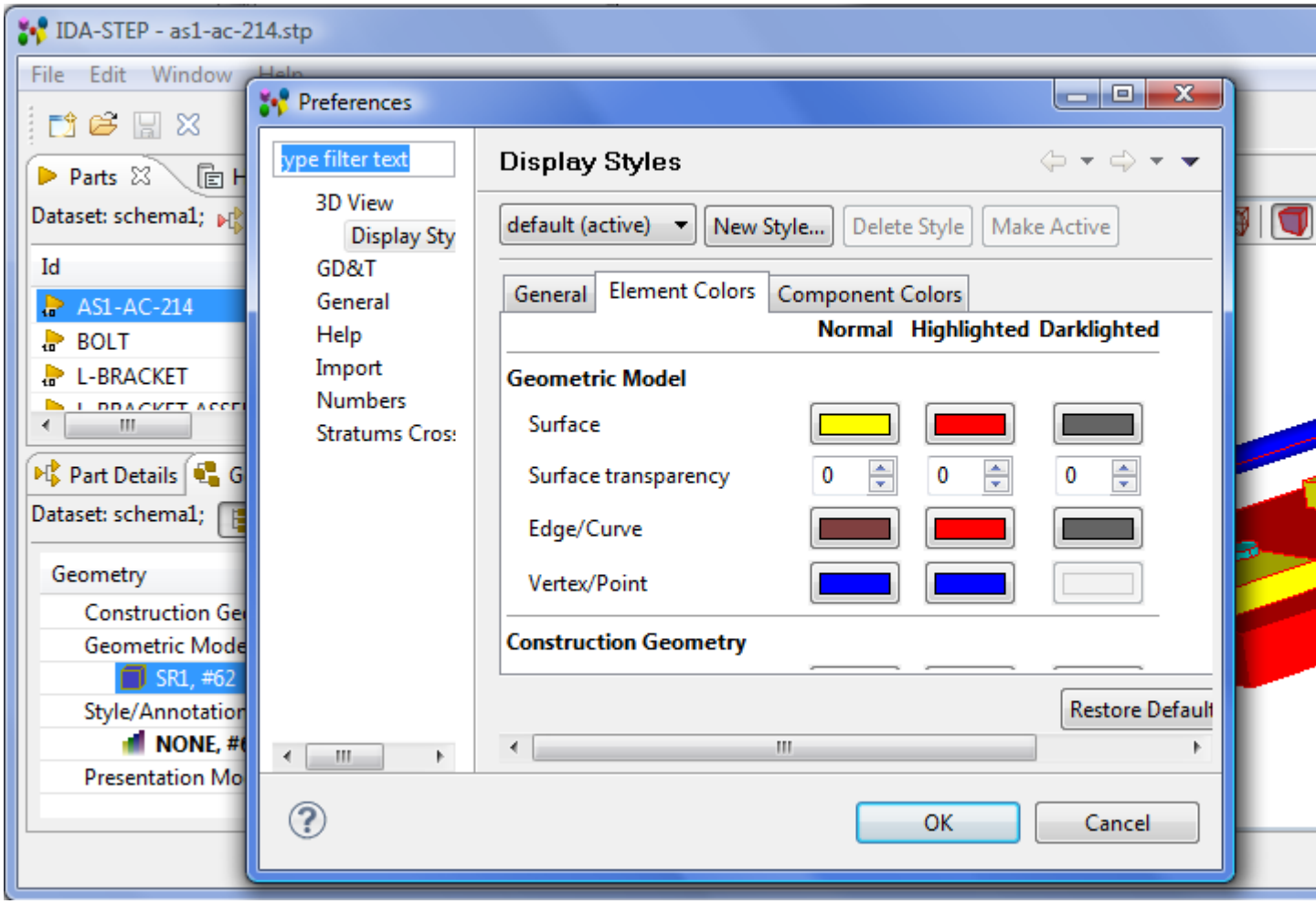


Figure 8. Component Colors tabbed panel

Component Color List

The **Component Color List** displays a list of default colours applied to the components in the **3D View**. The first colour in the list is set for the first component in STEP file. A colour in a square presents a colour of a surface and colour of a label 'Edge/Curve' inside the coloured square indicates the colour of the edge/curve of the surface. After a coloured square is selected in the list, it is marked with an asterisk. The table on the right side of the **Component Colors** panel displays the list of components with configurable properties.
Columns of the table:

- **Normal** - properties for a component in a normal state - not selected, not darkligthed.
- **Highlighted** - a component properties after it is selected/highlighted.
- **Darklighted** - a colour of a component, when other component in an assembly is selected/highlighted.

Rows of the table:

- **Surface** - default colour for a surface.
- **Surface transparency** - default transparency level for a surface.
- **Edge/Curve** - default colour for an edge/curve.
- **Vertex/Point** - default colour for a vertex/point.

To change color of a component:

- Press on a coloured square and select a colour in the appeared **Color** dialogue window.
- Press **OK** button to confirm the selection.

To change transparency level for a surface:

- Enter a number value into the field.
  Or
- Use up or down arrow next to the text field to increase or decrease number value.

If a needed colour is not found in the **Component Color** list, use **Add** button to add an applicable colour. An redundant colour can be removed by using **Remove** button. The button **Up** and **Down** can be used to change the order of colours.

Note. Preferences for the assembly components are not applied straight after they were changed in the **Component Colors** panel. You have to reload **3D View** to see changed preferences.

To top

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4.2. GD&T

The **GD&T** page specifies the default values used when creating elements in the **GD&T** view and the corresponding annotations in the **3D View** (see Figure 1):

- **Default dimensioning standard** - ISO and ASME have similar but different rules on how to apply GD&T and what is the exact meaning. Therefore it is essential to select one of them:
  - ASME, or
  - ISO.
- **Default length unit dimensions and tolerances** - all GD&T specifications have to be done in the same unit. The radio group allows to identify whether the default unit is
  - mm, or
  - inch.
- **Default decimal places for GD&T** - number of digits after the decimal separator for all length and area values. The numbers are rounded to this value.
- **Show trailing zeroes for GD&T** - if set indicates to show trailing zeroes after the decimal separator for length and area values; this is required by ASME. For ISO trailing zeroes are typically not shown and so this flag might be off.
- **Default decimal places for geometry -** number of places after the decimal separator for length and area values of geometric elements such as coordinates. All queried values from the 3D model and construction geometry are rounded to this value.
- **Shape aspect name generation strategy** - when creating GD&T element new features (shape_aspect) have to be created that corresponds to geometric_representation_items. Some CAD-Systems provide meaningful names for geometric_representation_items that might be used instead of creating new names, consisting of a prefix and a number:
  - **Use representation item name, if possible** - use the name from the corresponding geometric_representation_item if available.
  - **Generate new** - always generate a new name.

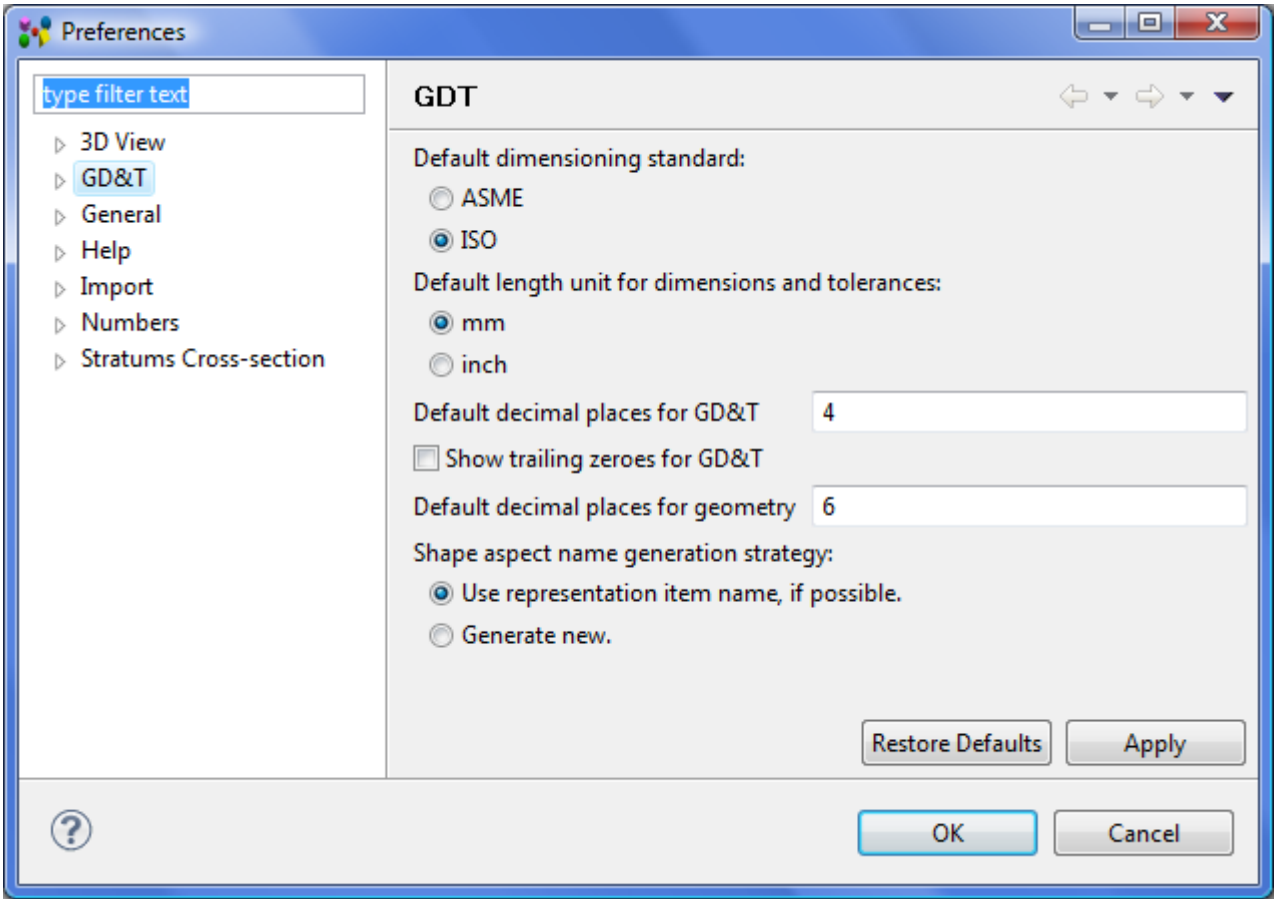

Figure 1. GDT preferences page

To top

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4.2.1. Annotation Styles

The **Annotation Styles** page specifies styling parameters to be used when creating annotations (see Figure 1):

- **Unit** (**mm**, **inch** or **mil**) - specifies the measure units to be used for the style parameters listed below;
- **Default way to display dimensions** - there are several ways how dimension, especially diameter dimensions, can be shown:
  - **always with projection curves** - a diameter dimension is presented with two projection lines. This case is used when (a) the annotation plane is perpendicular to the centre axis of a circle/cylinder or (b) when the annotation plane is in-line with a centre axis. This option is tuned *On* by default.
  - **without projection curves when possible** - a diameter dimension is presented with a dimension line directly ending on the circle/cylinder or its extensions.
- **Text height** - height of all texts in an annotation;
- **Curve width** - width of dimension, extension, reference and leader curves;
- **Terminator symbol size** - size of terminator symbol;
- **Minimum reference line length** - the minimum length for a reference line segment (continuous narrow line connecting with the leader line horizontally or vertically and on or at which the additional instructions are indicated).
- **Initial length of projection curve** - initial length of projection curves;
- **Length of projection curve extension beyond dimension line** - length of part of a projection curve that extends beyond the dimension line;
- **Ideal window size** - for 3D models up to this size and not smaller than a factor 2 or 2.5 an initial scaling value of 1 is chosen. Otherwise IDA-STEP determines a proper enlargement or reduction scale.

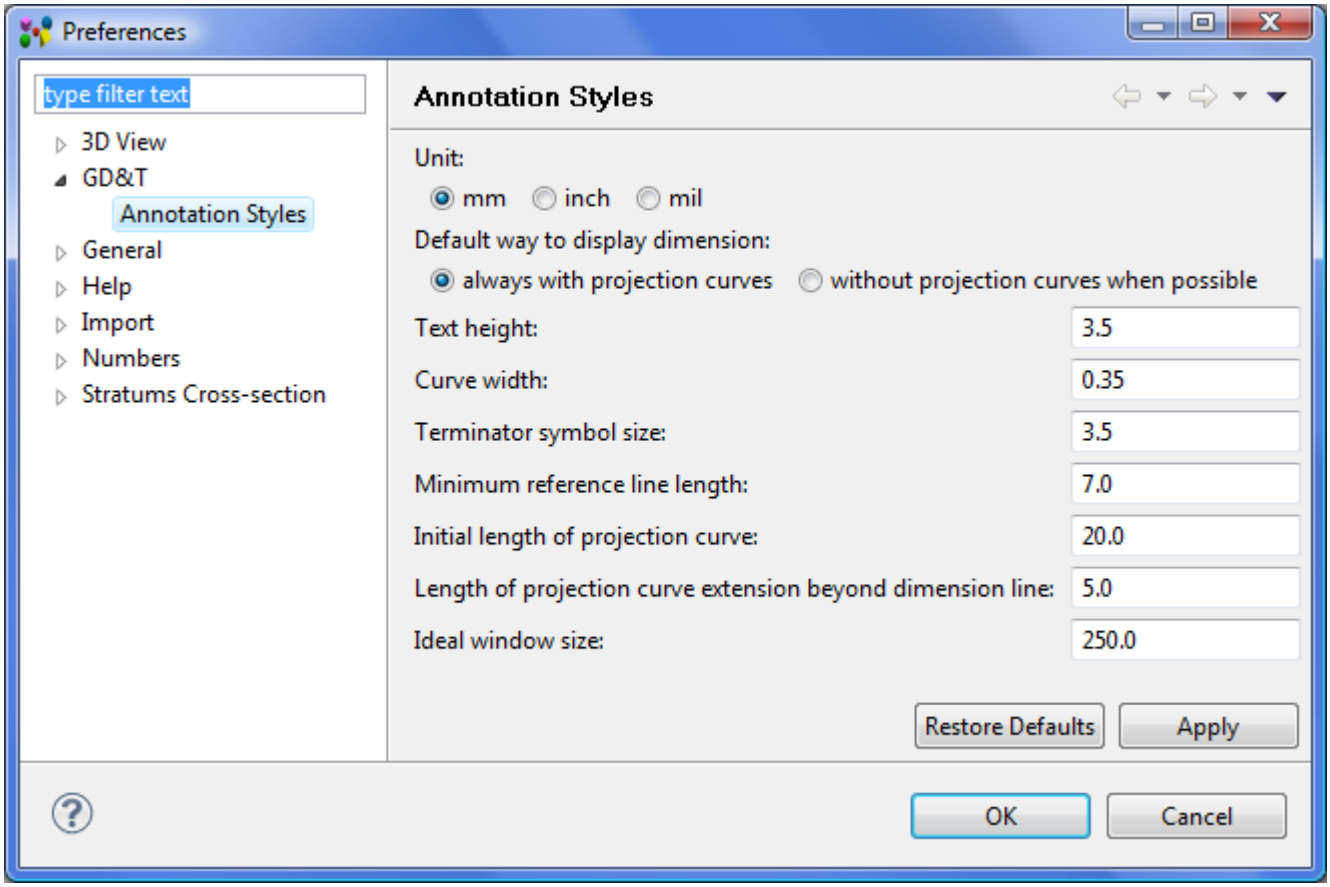

Figure 1. Annotation Styles preferences page

To top

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4.3. General

The **General** page displays setting for the workbench (see Figure 1). Each workbench window contains one or more perspectives. Perspectives contain views and control what appears in certain menus and tool bars. More than one workbench window can exist on the desktop at any given time.

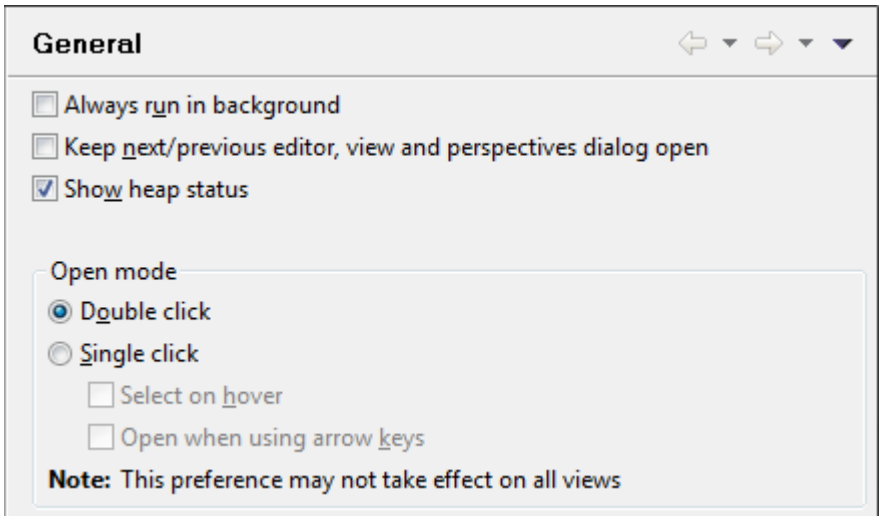

Figure 1. General preferences page

- Select **Always run in background** check box to perform long running operations in the background without blocking you from doing other work.
- Select **Keep next/previous editor, view and perspectives dialog open** check box to remain open view cycle dialogs when their activation key is let go. Normally the dialog closes as soon as the key combination is release.
- Select **Show heap status** check box to display an indicator showing information about current Java heap usage. See also Virtual Machine Memory.
- You can select one of the following methods for opening resources in the **Open mode** layout:
    - **Double click** - Single clicking on a resource will select it and double clicking on it will open it in an editor.
    - **Single click** (Select on hover) - Hovering the mouse cursor over the resource will select it and clicking on it once will open it in an editor.
    - **Single click** (Open when using arrow keys) - Selecting a resource with the arrow keys will open it in an editor.

      Note: Depending on which view has focus, selecting and opening a resource may have different behavior.

To top

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4.3.1. Keys

On the **Keys** page, you can customize the function of the keyboard in IDA-STEP (see Figure 1). Within Eclipse, key strokes and key sequences are assigned to invoke particular commands.

A *key stroke* is the pressing of a key on the keyboard, while optionally holding down one or more of these modifier keys: `Ctrl`, `Alt` (`Option` on the Macintosh), `Shift`, or `Command` (only on the Macintosh.) For example, holding down `Ctrl` then pressing `A` produces the key stroke `Ctrl+A`. The pressing of the modifier keys themselves do not constitute key strokes.
A *key sequence* is one or more key strokes. Traditionally, Emacs assigned two or three key stroke key sequences to particular commands. For example, the normal key sequence assigned to `Close All` in emacs is `Ctrl+X Ctrl+C`. To enter this key sequence, one presses the key stroke `Ctrl+X` followed by the key stroke `Ctrl+C`.
A *key binding* is the assignment of a key sequence to a command.

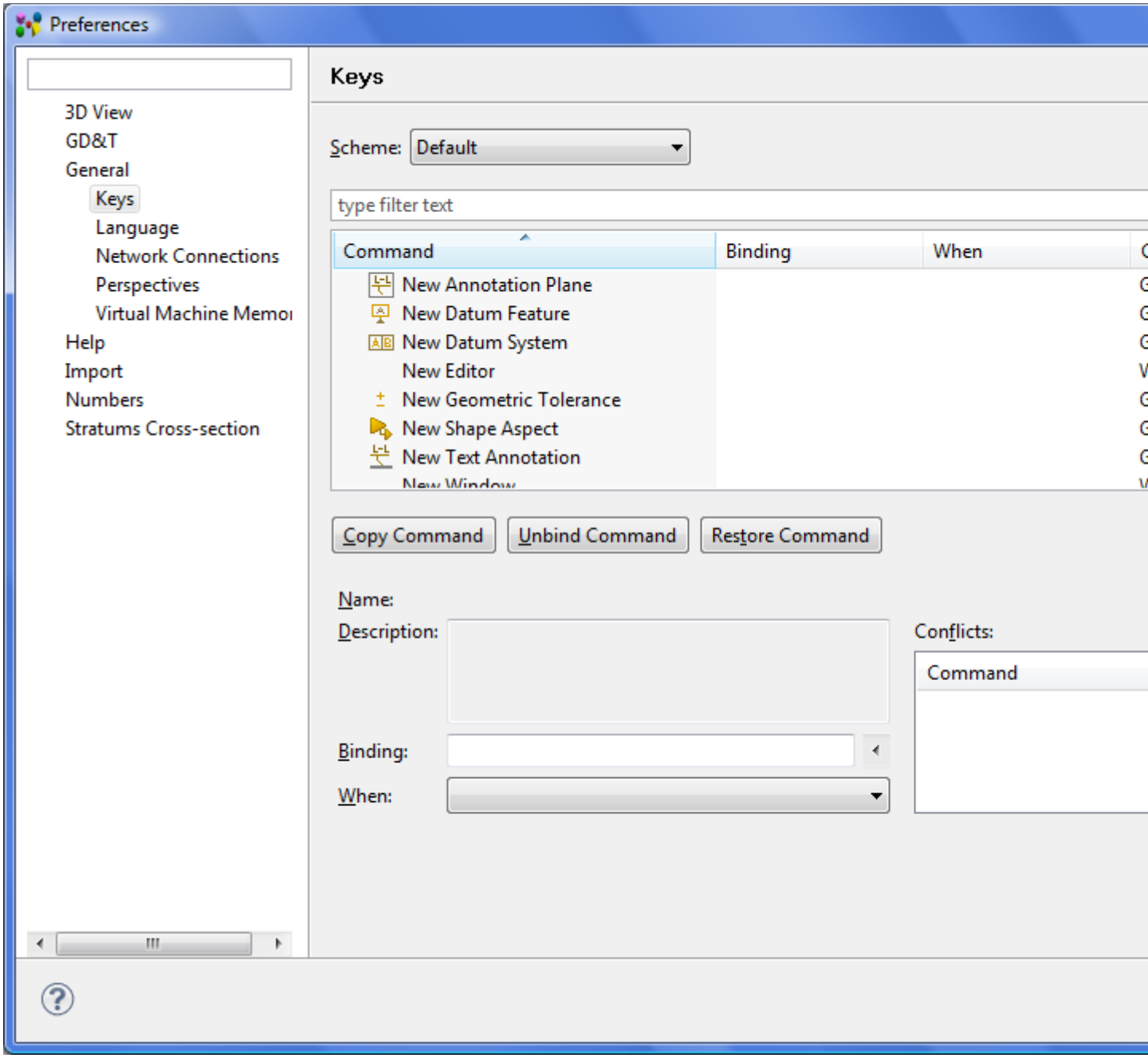

Figure 1. Keys preferences page

- **Scheme** combo box contains a set of bindings. Choose the scheme you are most comfortable with by changing the **Scheme** setting on the keys preference page.
    - *Default* scheme contains a general set of bindings, in many cases recognizable as traditional key sequences for well known commands. For instance, `Ctrl+A` is assigned to `Select All`, and `Ctrl+S` is assigned to `Save`.
    - *Emacs* scheme contains a set of key bindings familiar to users of Emacs. For instance,

`Ctrl+X H` is assigned to `Select All`, and `Ctrl+X S` is assigned to `Save`.
The *Emacs* scheme is not a complete set of bindings like the *Default* scheme. Rather, it borrows from the *Default* scheme where possible, only defining explicit Emacs-style bindings where they vary from the *Default* scheme. Generally, only well known commands like `Select All`, `Save`, etc. have specific Emacs key sequences associated with them. If you choose the *Default* scheme, all *Emacs* bindings are ignored. If you choose the *Emacs* scheme, explicit Emacs-style key sequence assignments take precedence over any conflicting assignments in the *Default* scheme.

- Filter field above the table with keys parameters. Enter a filter text to filter out commands relevant to the entered text.
- The table contains read only information:
  - **Command** - a name of the command, for which the key sequence is assigned.
  - **Binding** - indicates a key sequence, which is assigned to a particular command.
  - **When** - the context of IDA-STEP. This context is usually determined by the active part, but it can be influenced by the active window or dialogue as well. If the active part does not choose a particular context, the workbench will set the active context to *In Dialogs and Windows*. Key bindings can vary based on the current context of IDA-STEP:
    - Editing Text,
    - In Dialogs,
    - In Dialogs and Windows,
    - In Windows.
  - **Category** - identifies the category of IDA-STEP context (Editor (for example 'GD&T'), Window, File and etc.).
  - **User** - indicates that this is a user binding by displaying a **U**. If there was a conflict with another key, this column would also display a **C**.
    Note. In case of the conflict, the conflicted items are listed it the **Conflicts** table (see below).
- **Copy Command** button - press this button to create a second command entry for you to bind another key to.
- **Unbind Command** button - press this button to remove a key sequence (i.e. **Binding** value) for the selected command.
- **Restore Command** button - press this button to delete the binding. **Note.** All copies of the command will be removed (if any is previously created).
- **Name** field (read only) - this filed displays a name of the command, selected in the table.
- **Descriptions** multi line field (read) - presents additional information about the command, which name is defined in the **Name** field.
- **Binding** field - bind the command by putting focus in the **Binding** text box and pressing key combination (for example 'CTRL+5') like you would if you were executing the command.
- **When** combo box - select a context value in the drop down list.
- **Conflicts** table - if the user sets a key binding and creates a conflict, the conflicting bindings will be displayed in the conflicts list. This can be used to navigate between conflicting keybindings so that they can be changed. These types of conflicts can be resolved by explicitly assigning the key sequence to one of the commands, or remove it from the other.
  See Figure 2: There are listed two commands in the **Conflicts** list. The *New Editor* command has a user assigned (you can see *U* letter in the **Users** column) key stroke. The **Undo** command has a predefinde binding. The *C* letter in the **Users** columns informs about the conflict. Click on an applicable command in the **Conflicts** list and resolve the conflict by removing the binding or adding a new sequence of keys into the **Binding** field.

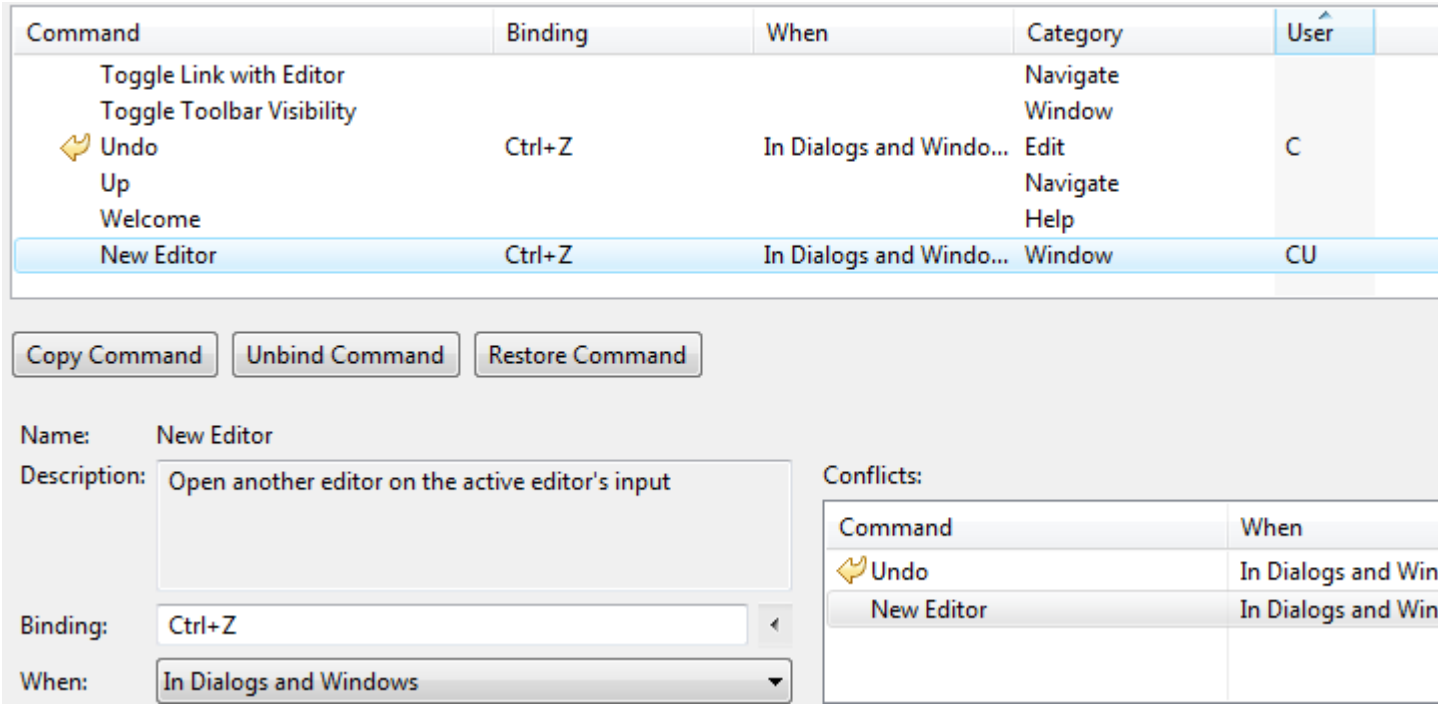

Figure 2. Conflicts list with defined conflicted bindings.

- **Filters...** button - filter out a list of commands in the table according to the category of IDA-STEP contexts. The **When Context Filters** window appears after the filter button is pressed (see Figure 3):
  - **Filter action set contexts** - select this option not to show commands with action set contexts,
  - **Filter internal contexts** - select this option not to show commands with internal contexts,
  - **Filter uncategorized commands** - select this option not to show commands with an unrecognized context.
    Note. By default, all options are selected.
  - Use **OK** button to confirm selections.

Figure 3. When Context Filters

- **Export CSV...** button - the list of binded command can be saved in external file of CSV format. Press this button, specify a filen name and a location in the file system for, and press **Save** button in the appeared **Save as** dialog window.

For more detailed information, refer the **Eclipse** documentation: Workbench User Guide > Reference > Preferences -> Keys.

To top

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4.3.2. Language

On the **Language** page you can set the default language for data by selecting an applicable language in the combo box.

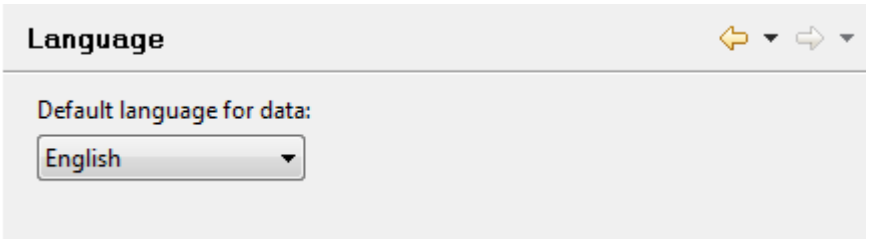

Figure 1. Language page

To top

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4.3.3. Network Connections

By default IDA-STEP is performing the license activation and software installation and update processes via direct Internet access to the central IDA-STEP license server. This is called **Online registration, installation and update**, see Register, Update and Install.

Depending on the kind of operating system you are using you may need to grant Internet Access to IDA-STEP to be able to do so. In most cases the default **Direct connection to the Internet** option is then sufficient for IDA-STEP to connect to the license server. But in some organisations special proxy servers are set up that have to be taken care. This is done in the **Network Connections** preference page described here:

- The System proxy configuration (if available) - this option is very useful, when the proxy is already configured in the user system. No additional configuration is needed.
- By default the **Direct connection to the Internet** option is selected. Enable this option to have all connections to remote systems happen directly without involving a proxy server.
- To allow connection to the remote server from behind the firewall, select the **Manual proxy configuration** option. The below fields are enabled. Specify needed options for the proxy server:

    - **HTTP proxy** - Use this option to specify the server and port that is to be used when making HTTP connections.
      Enter URL into **HTTP proxy** field. Also you can specify a port in the **Port** field. If this field is empty, the default port of 80 is used. You can set *On* the option **Use this proxy server for SSL**. If this option is checked, the HTTP proxy server is used for SSL connections as well.
    - Specify SSL proxy server location in the **SSL proxy** field. Use this option to specify the server and port that is to be used when making SSL connections. Optionally you can enter a port in the **Port** field. If this field is empty, the default port of 443 is used.
    - Specify SOCK proxy server location in the **SOCKS proxy** field (please refer Wikipedia for more information about SOCKS). Optionally you can enter a port in the **Port** field. If this field is empty, the default port of 1080 is used.

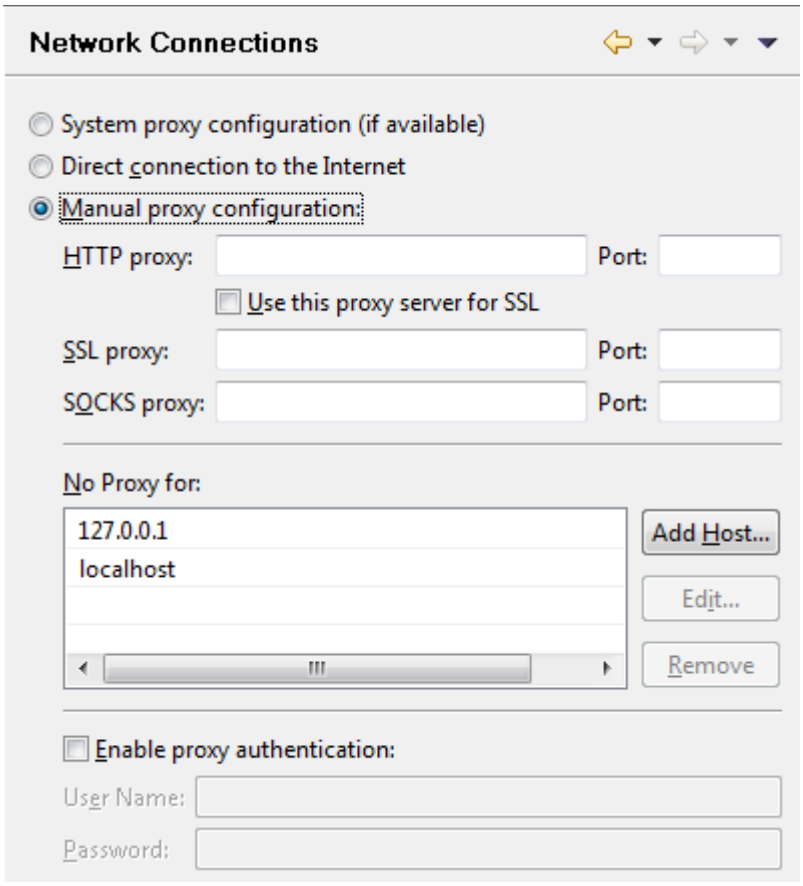

Figure 1. Network Connections page

- **No Proxy for** - Use this table to specify, either by name or pattern, which hosts should not use any proxy but instead should always be connected to directly:
  - Add a new host by clicking on **Add Host** button. The **No proxy host names** dialog window appears (see Figure 2).
  - Specify a host name in the text field and click **OK** button.

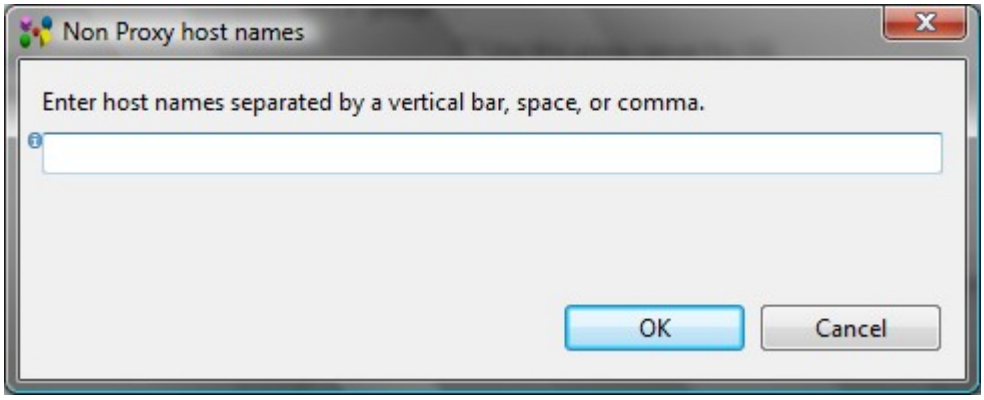

Figure 2. Nor Proxy host names dialog window

- **Enable proxy authentication -** Enable this option to specify a **User name** and **Password** that is to be used when connecting to the proxy server.

  - After all needed entries were filled, click **OK** or **Apply** buttons on the bottom section of the page to apply settings.

To top

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4.3.4. Perspectives

On the **Perspective** page you can select the following options for the launching of perspectives:

- **Enable perspective switching** - controls if user wants perspectives to be proposed. *On* by default.
- **Automatically switch single perspective** - if *On*, IDA-STEP will auto switches perspective, if there is only one perspective proposed to the user. This option is enabled for selecting only after the previous option is set *On*.
- **GDT perspective** - controls the switching of **GDT** perspective, when some GD&T data are created (available only with GD&T Editor feature):
    - **Always open** - automatically switch perspective, if some geometric tolerance, dimension, or datum is created and the current perspective is not a **GDT** perspective.
    - **Never open** - do not switch to **GDT** perspective automatically on the creation of GD&T elements.
    - **Prompt** - prompt a dialogue window to select an applicable action.

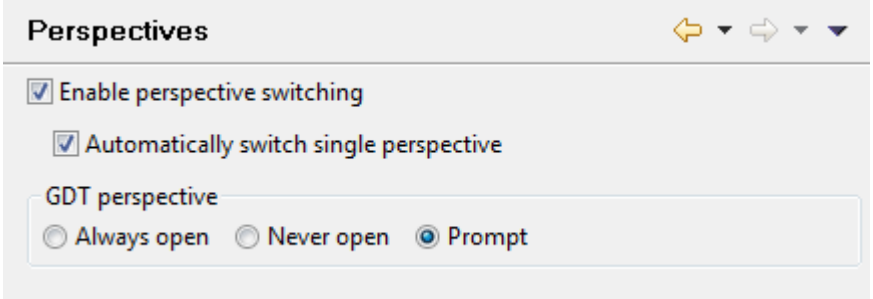

Figure 1. Perspectives page

See also Pre-defined Perspectives.

To top

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4.3.5. Virtual Machine Memory

The **Virtual Machine Memory** controls the maximum amount of memory available to the Java Virtual Machine (JVM) that is used by IDA-STEP. The setting depends of the kind of operating system (Windows or Linux), the inner architecture (32 or 64 bit), the installed physical memory (e.g. 2 GByte) and other installed software packages. No fixed rule can be given about the maximum memory settings.

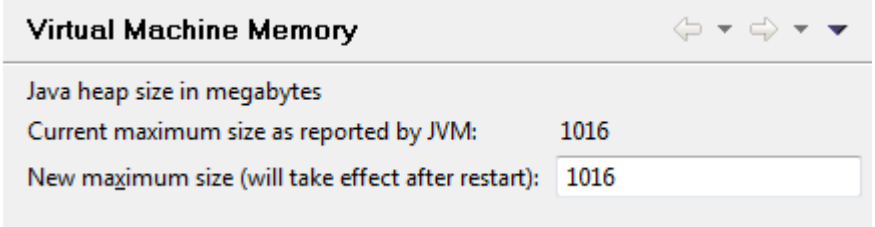

Figure 1. Virtual Machine Memory page

We have to distinguish between the *current maximum size* and the *possible maximum size*. The current maximum size is displayed in this dialogue and it is less or equal to the possible maximum size. But as explained above the possible maximum size is not know and thus can't be displayed.

The actual memory consumed and allocated by the JVM is displayed in the small *Heap Status* control in

the status bar (see Figure 2). The lower number is the amount of memory actual in use by the JVM and the upper number the amount of memory currently allocated from the operating system. The upper number is always smaller or equal to the current maximum size. The actual number of memory in use is dynamically growing and shrinking. It can be reduced when pressing the so called **Garbage Collection** button (waste bin symbol).
See General preferences on how to switch the **Heap Status** control on and off.

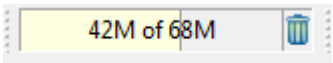

Figure 2. Heap status with actual used and allocated memory

Example: Under Windows 32bit the maximum physical memory that can be managed by the operating system is about 3 GB. If you have installed more memory Windows can't take advantage of it. If you now try to set the maximum memory setting to e.g. 2.5 GB the JVM will not allow you to do so because a significant portion of the memory is reserved for the operating system. A typical maximum value under 32 Windows is 1.6 GB (or 1600 megabytes). But if IDA-STEP would allocate this whole amount of memory then other applications running in parallel would decrease in performance because windows has to swap in and out the applications between the hard disk and physical memory. If you need more memory than is available under 32 bit we recommend to switch to a 64 bit operating system with at least 6 GB.

Any change of the maximum size will only become effective after re-starting of IDA-STEP. The fields are (see Figure 1):

- **Current maximum size as reported by JVM** - the text field displays the actual max. memory available to the currently running JVM.
- **New maximum size (will take effect after restart)** - the new maximum value to be used the next time IDA-STEP is started anew.

Note that the initial values of *current maximum size* and *new maximum size* are often not identical, even after re-start. This is because the value the JVM is asking for and what it get by the system may differ. When entering new values for *new maximum size* IDA-STEP is dynamically checking with the operating system and if the value is not accepted by the system it will indicate this by the message "Invalid maximum memory value".
The "new maximum size" value will be stored in the file ida-step.ini that you can find in the directory where IDA-STEP is installed. The line of interest looks like this

```
-Xmx1500m
```

for a new maximum size of 1500 megabytes. Dependent of your kind of installation this directory may be write protected. In this case when you try to change the new maximum size you will see an error message like this:

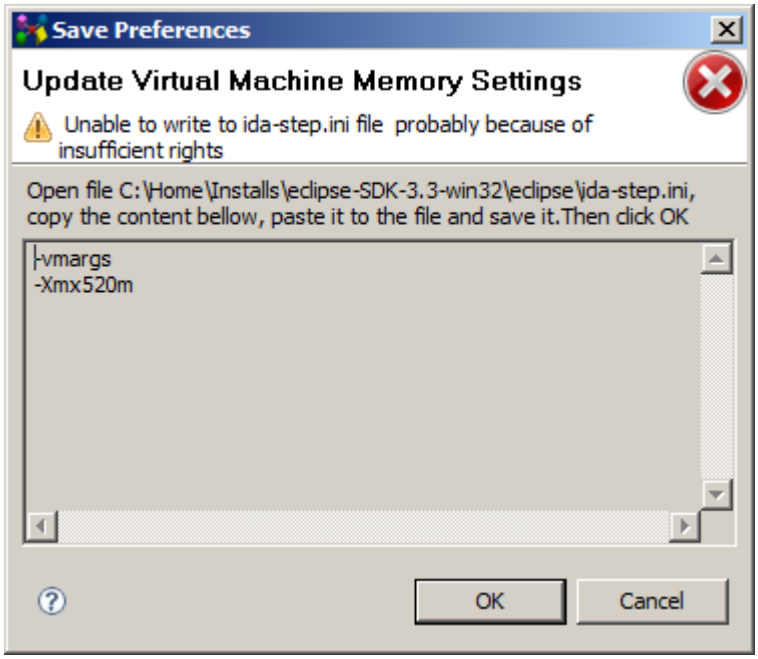

Figure 3. Save Preferences dialogue window in failure case

If you see this message you have to change this entry manually with some editor in protected mode.

To top

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4.4. Help

On the **Help** preferences page you can indicate, how to display the help information (see Figure 1).

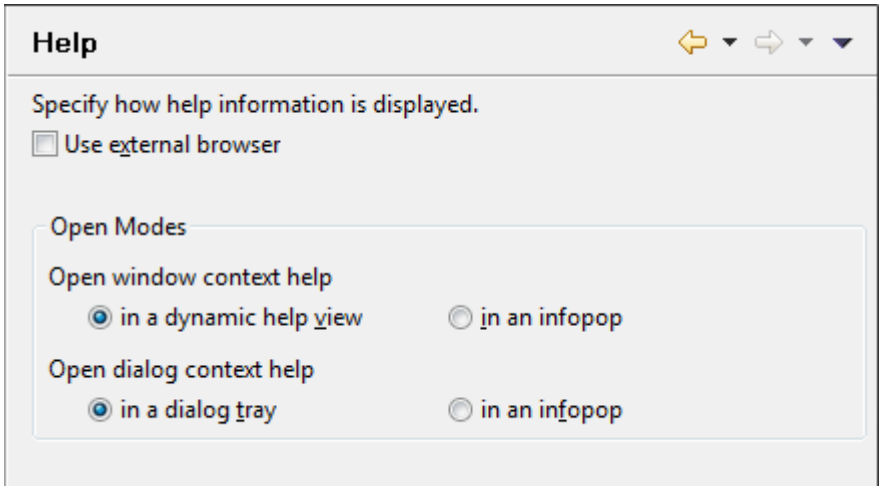

Figure 1. Help preferences page

- Select the **Use external browser** check box to force help to use external browsers.
- Select the open modes in the **Open Modes** layout:
  - **Open window context help in a dynamic help view/in an infopop** - this option allows you to determine whether the window context help will be opened in a dynamic help view or in an infopop.
  - **Open dialog context help in a dialog try/in an infopop** - this option allows you to determine whether the dialog context help will be opened in a dynamic help section of help view or in an infopop.
- Click **OK** or **Apply** buttons on the bottom section of the page to apply settings.

To top

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4.5. Numbers

On the **Numbers** page you can indicate, how to display the numbers, which are not quantities (see Figure 1)

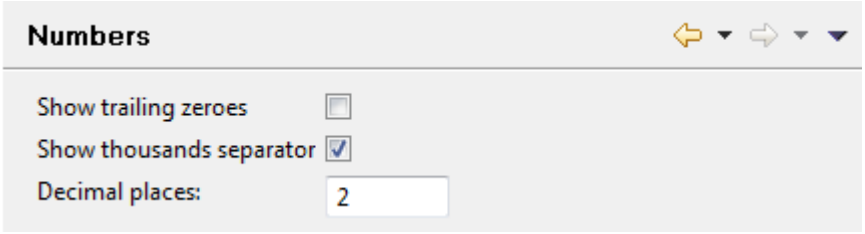

Figure 1. Numbers preferences page

- **Show trailing zeroes** - select the check box to set an option *On*.
- **Show thousands separator** - select the check box to set an option *On*.
- **Decimal places** - enter the number of decimal places.
- Click **OK** or **Apply** buttons on the bottom section of the page to apply settings.

To top

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4.5.1. Quantities and Units

On the **Quantities and Units** page you can define needed quantities and units settings (see Figure 1).

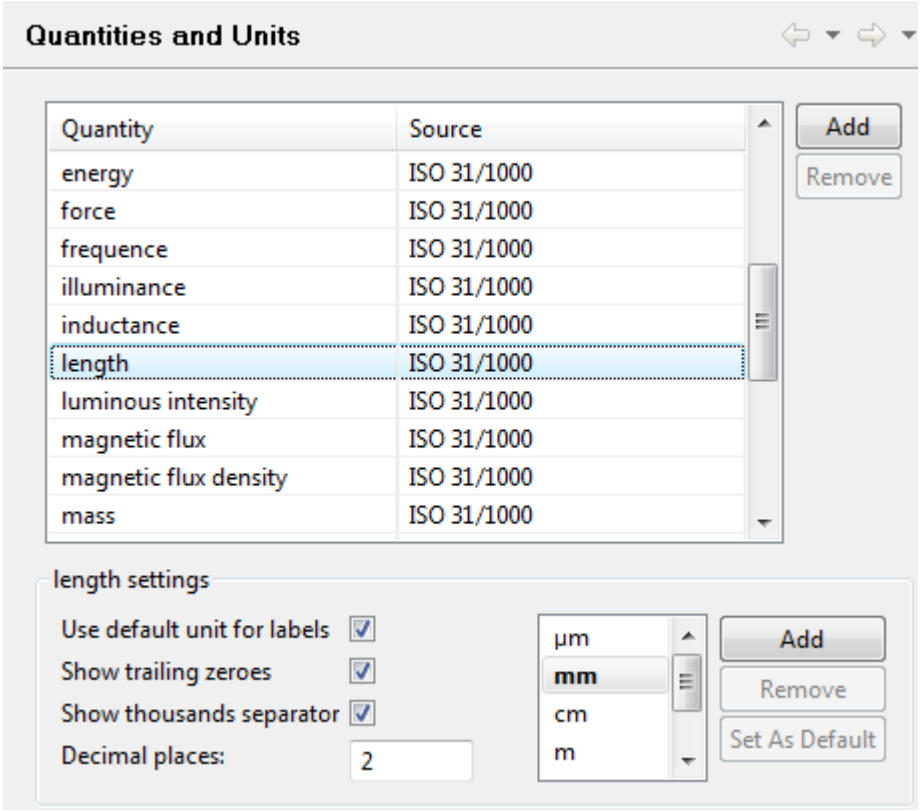

Figure 1. Quantities and Units page

- Select an applicable quantity in the table on the top section of the page, where:
  - **Quantity** - the quantity name;
  - **Source** - the source of the selected quantity (ISO or user defined).

Or

- Add a new quantity by clicking on **Add** button next to the table. The New quantity wizard dialog window appears. Created quantity appears in the table.

Or

- Remove the redundant or incorrect quantity by using **Remove** button next to the table.

In the **<quantity> settings** layout the settings layout the setting for the selected quantity are displayed. The name of the layout is dynamic, it shows the currently selected quantity name (see Figure 1, the 'length' quantity is selected and displayed in the label of the layout).

- **Use default unit for labels** - if this option is *On*, the default unit label is displayed in the table cell (see Figure 2).

Figure 2. The table cell label

- Available units for the selected quantity are displayed in the list on the right side of the settings layout (see Figure 1):
  - Use **Add** button to add a new unit into the list. The Unit wizard dialog window appears on the screen.
  - Use **Remove** button to remove the incorrect or redundant value from the list. The default value can not be removed.
  - Use **Set As Default** button to set the currently selected unit in the list is set as default unit.

- **Show trailing zeroes** - select the check box to set an option *On*.
- **Show thousands separator** - select the check box to set an option *On*.
- **Decimal places** - enter the number of decimal places.

- Click **OK** or **Apply** buttons on the bottom section of the page to apply settings.

To top

New quantity wizard

The **New quantity** wizard helps the user to create a new quantity:

- Enter a name of the quantity to be created into the **Name** field.
- The table in the middle of the window displays the default unit settings for the quantity to be created.
    - Select an applicable value from the **Prefix** drop down list.
    - Select a value from the **Unit** drop down list.
    - Enter an exponent value into the **Exponent** cell (available values -1, 1, 2 and 3).
    - Click **Add** button to add more components if needed.
    - The **Label** above the table displays the quantity representation.

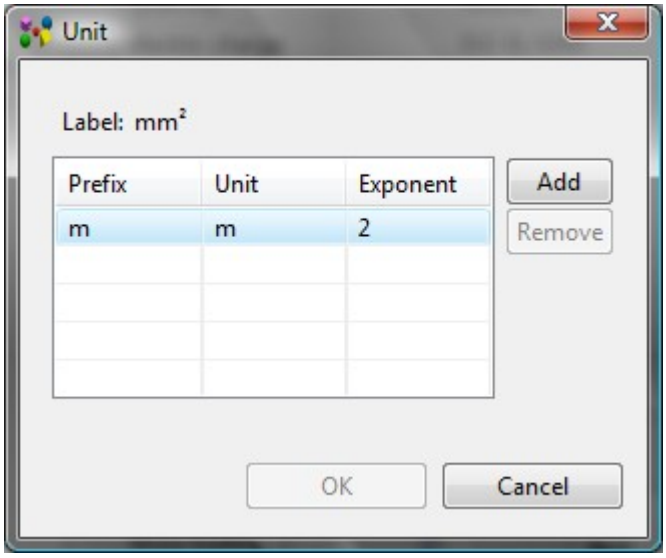

Figure 3. New quantity wizard dialog window

- After all parameters are set, click **OK** button. The created quantity appears in the table (see Figure 4). The source of the created quantity is set to 'User defined'.

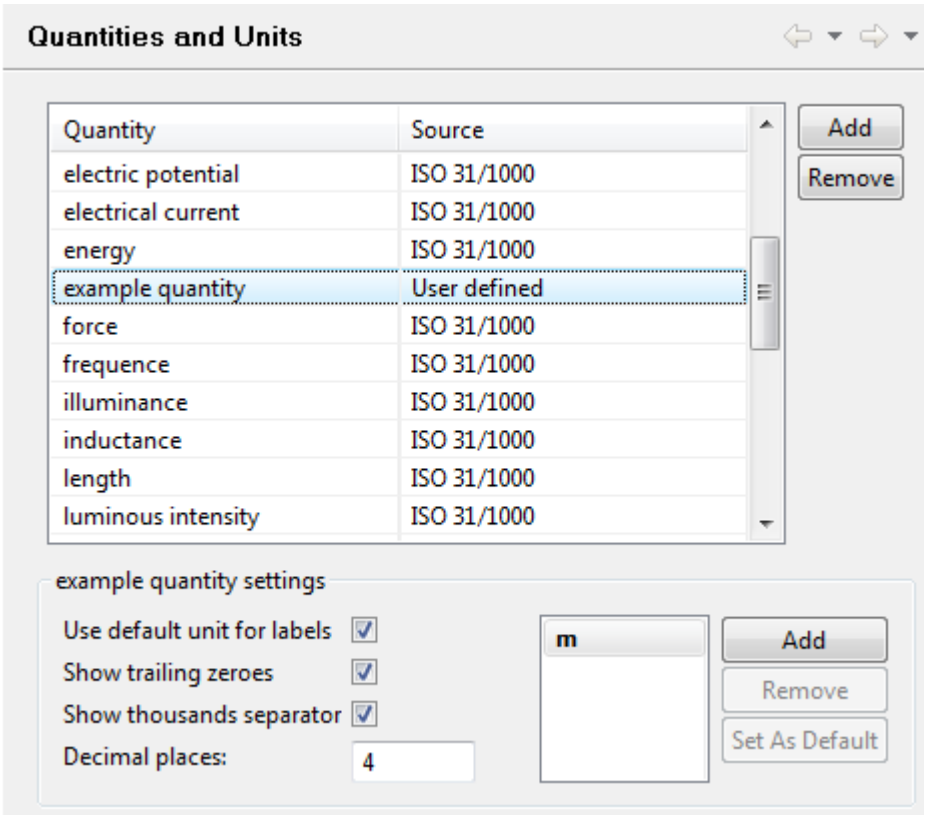

Figure 4. Created new quantity ('example quantity') in the table

To top

Unit wizard

The **Unit** wizard helps the user to create a new unit for the selected quantity (see Figure 6).

- Select an applicable quantity in the **Quantities and Units** page table (see Figure 5) and click **Add** button next to the list of units in the **force settings** layout.

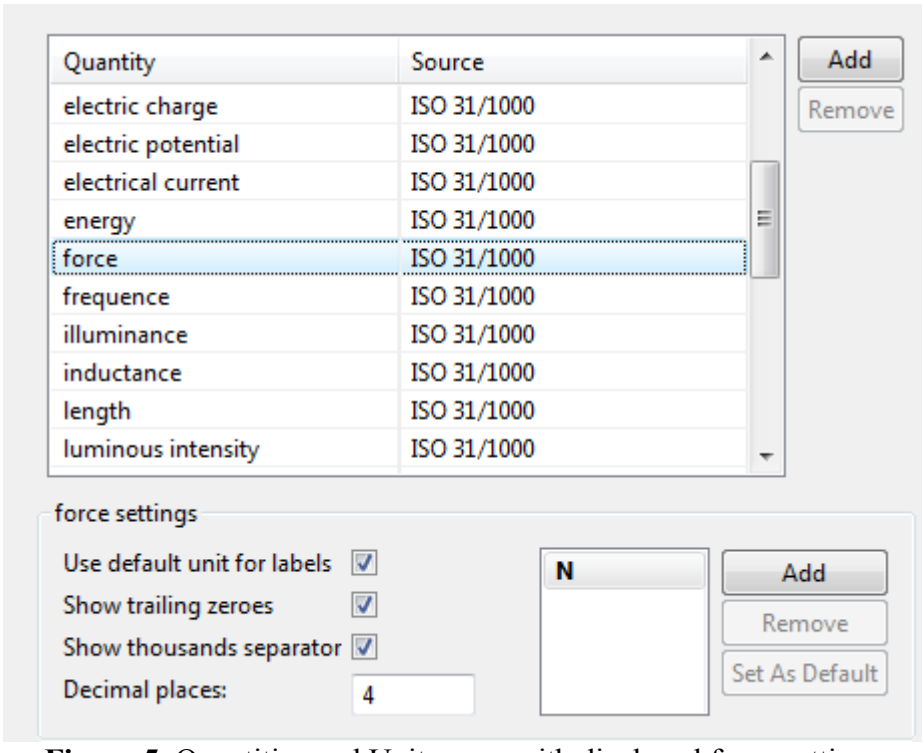

Figure 5. Quantities and Units page with displayed force settings

- The **Unit** wizard dialog window appears on the screen (see Figure 6):

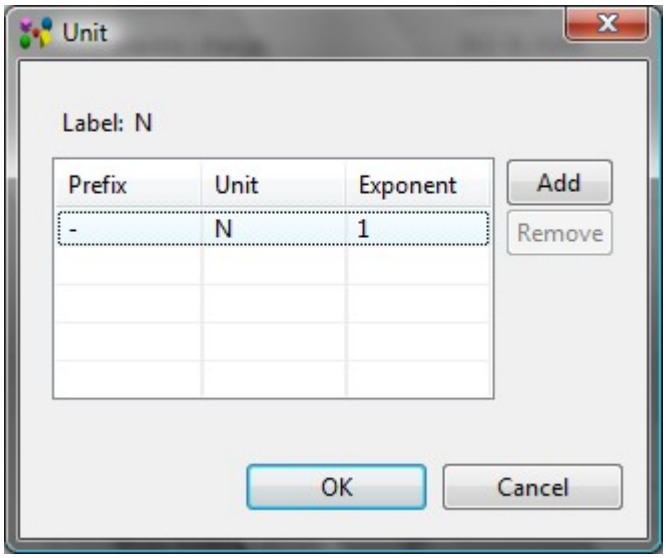

Figure 6. Unit wizard dialog window

- Select an applicable value from the **Prefix** drop down list.
- Select a value from the **Unit** drop down list.
- Enter an exponent value into the **Exponent** cell (available values -1, 1, 2 and 3).
- Click **Add** button to add more components if needed.
- The **Label** above the table displays the quantity representation (see Figure 7).

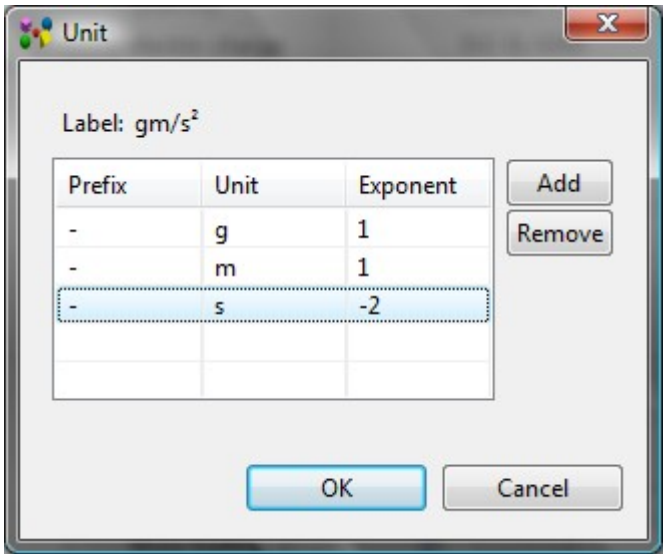

Figure 7. Unit wizard dialog window with unit parameters

- Click **OK** button to create a new unit. The unit appears in the list in the settings layout (see Figure 8).

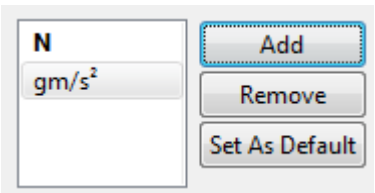

Figure 8. Created unit in the list

To top

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4.6. PCB Stratums Cross-section

The **Stratums Cross-section** page displays the current settings for the **Stratums Corss-section** view (see Figure 1):

- **Spacing between stratums** - the vertical spacing in pixels between stratums in expanded view.
- **Number of feature spacings** - a number of the feature spacings on most design stratums. This number denotes how many gaps are drawn on each design stratum with *Min. Feature Size* and *Min. Feature Spacing* information available.
- **Number of feature spacings for power or ground** - a number of feature spacings on the power or ground design stratums. This number denotes how many gaps are drawn on each design stratum with *Min. Feature Size* and *Min. Feature Spacing* information available and *Purpose* set to 'Power or ground'.
- **Spacing between passages** - a horizontal spacing in pixels between passages.
- **Passage maximum plating multiplier** - a multiplier denoting what width one passage plating can occupy in a comparison to the whole passage width.
- **Minimum stratum width** - a minimum width of a stratum in pixels. This is usually relevant only when a design stack model has no features and no passages.
- **Laminate indentation** - a horizontal distance between different levels of the laminate. This is also a distance between two consecutive laminates on the same level.
- **Spacing between labels** - a vertical spacing in pixels between labels in a collapsed view.
- **Feature trapezoid cut** - a horizontal distance in pixels between corners of the feature trapezoids. The distance is horizontal only and between corners on one side (left or right) of the trapezoid.
- **Padding between labels and stratums** - a horizontal distance in pixels between labels and stratums.
- **Vertical padding** - a vertical distance in pixels between top of the picture area and a picture itself.
- **Horizontal padding** - a horizontal distance in pixels between the left side of the picture area and a picture itself.
- **Default plating width in pixels** - a horizontal size in pixels of the passage plating when no plating width is defined.

.

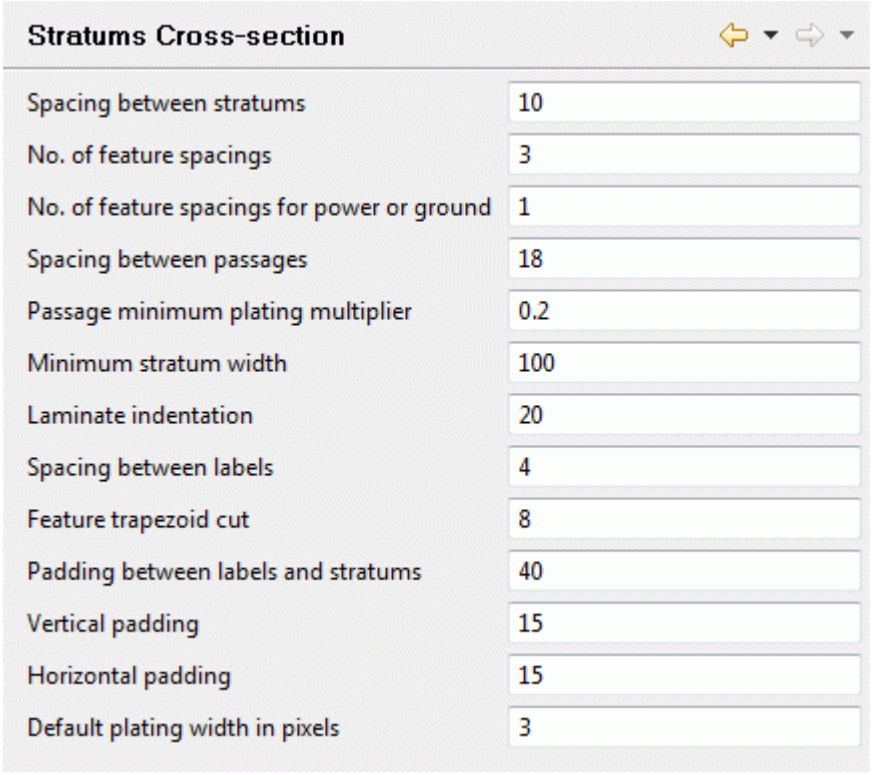

Figure 1. Stratums Cross-section preferences page

An applicable setting can be changed by entering a new value into the text field next to the setting label. To apply newly entered settings, press **Apply** or **OK** button on the bottom section of the page.

To top

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4.6.1. Color Schemas

On the **Color Schemas** page (see Figure 1) you can edit default component, text font and background colors for the color schemas, create a new color schema and define your own colors for the component in one layer, label text and background.

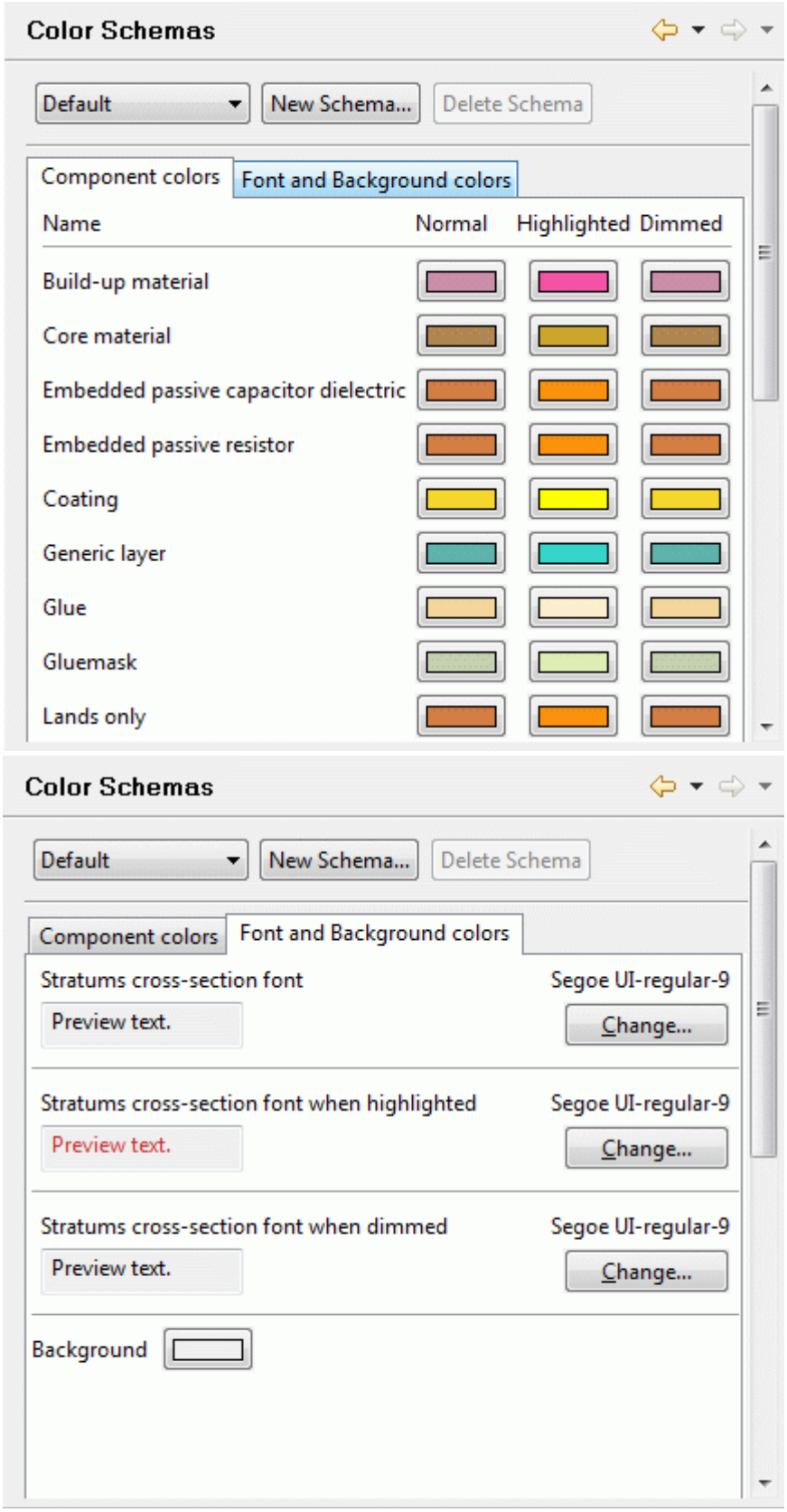

Figure 1. Color Schemas preferences page, Component colors and Font and Background colors tabbed panels

You can select the schema to be applied for the current data by selecting an applicable value in the combo box on the top left corner of the page or create a new schema by clicking on the **New Schema** button next to the previously mentioned combo box.

To top

New schema

In order to create a new color schema, click on the **New schema** button on the top section of the **Color Schemas** page. The **Create new Color Schema** dialog window appears (see Figure 2):

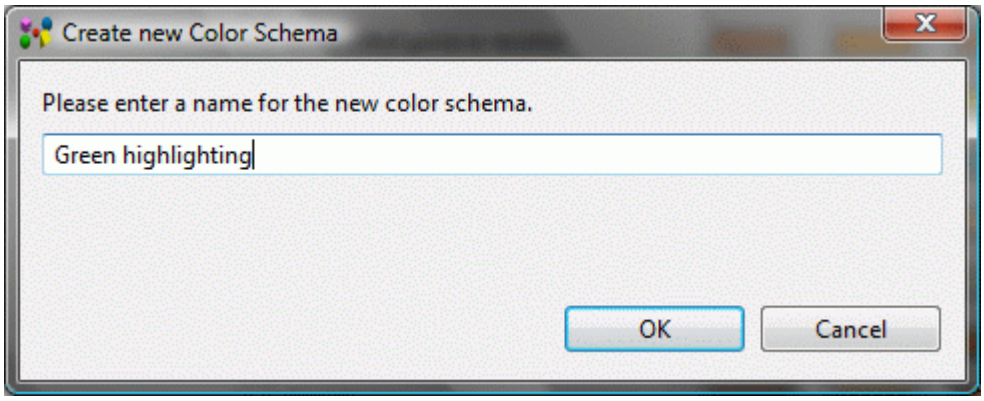

Figure 2. Create new Color Schema wizard dialog window

- Enter a name for the new color schema into the text field.
- Press **OK** button. A new schema is created and marked as selected for the current data - the name of the newly created schema is presented in the combo box in front of the **New Schema** button (see Figure 3).

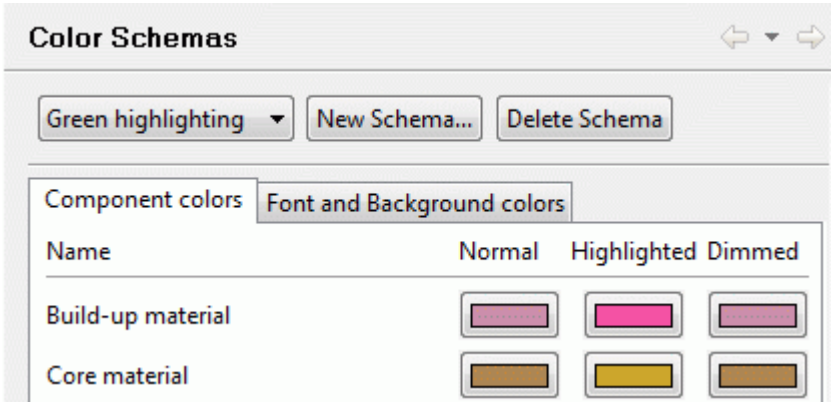

Figure 3. New schema name in the combo box in front of New Schema button

The colors for a newly created schema are the same as for the previously selected schema. You can change the color of the components in one layer in the **Component color** tabbed panel. To change text font and background color, go into the **Font and Background** colors tabbed panel.

To top

Deleting Color Schema

There is a possibility to remove a redundant or an incorrect color schema by using **Delete Schema** button on the bottom section of the page.

Note: Only user created color schema can be removed. You can not remove default color schemas.

To top

Component colors

The **Component colors** tabbed panel presents the colors for a PCB library components in one layer (see Figure 4).

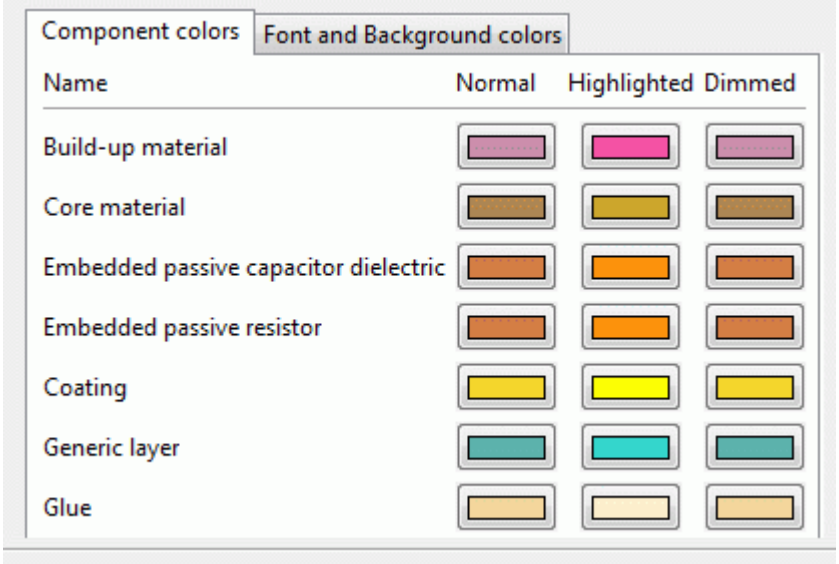

Figure 4. Component colors tabbed panel

You can change the color for the following:

- Stratum technology stratum purpose components:
  - Build-up material,
  - Core material,
  - Embedded passive capacitor dielectric,
  - Embedded passive resistor,
  - Coating,
  - Generic layer,
  - Glue,
  - Gluemask,
  - Lands only,
  - Signal,
  - Pastemask,
  - Power or ground,
  - Silkscreen,
  - Soldermask,
  - Solderpaste.
- Passage technology type components:
  - Component termination,
  - Border,
  - Unsupported,
  - Via.
- Via filling color.

You can change the following colors for the components:

- **Normal** - the regular color for the components.
- **Highlighted** - the color of the highlighted component.
- **Dimmed** - the color of the components, after one of other components/layers is highlighted.

To top

Font and background colors

In the **Font and Background colors** tabbed panel you can define text font of layer labels and background

color for the particular color schema (see Figure 5).

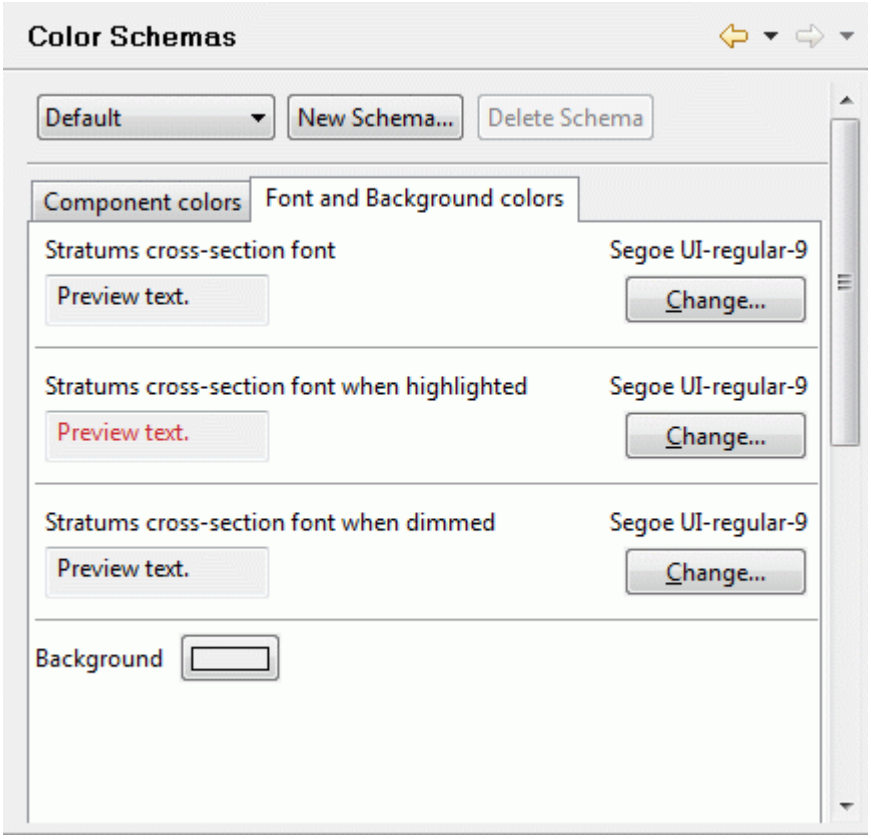

Figure 5. Font and Background colors tabbed panel

- **Stratums cross-section font** - format font for the labels of a regular layers (when layer is neither highlighted, nor dimmed).
- **Stratums cross-section font when highlighted** - format font for the label of the highlighted layer.
- **Stratums cross-section font when dimmed** - format font for the label of the dimmed layer.

To change font formating, press **Change** button in a particular layout and select an applicable options in the appeared dialog window. You can preview the formated text in the text fields in front of **Change** button.

To top

Changing Colors

Follow the routine to change a color in the **Color Schemas** preferences page:

- Activate the Component colors tabbed panel.
- Click on the colored button ( ) in a table of colors. The **Color** dialog window appears on the screen (see Figure 6).

Figure 6. Color dialog window

- Click on an applicable color in the **Basic color** or **Custom colors** layout.

Or

- Define your custom color by clicking on the **Define Custom Colors** button. The additional part of the dialog window appears (see Figure 7):

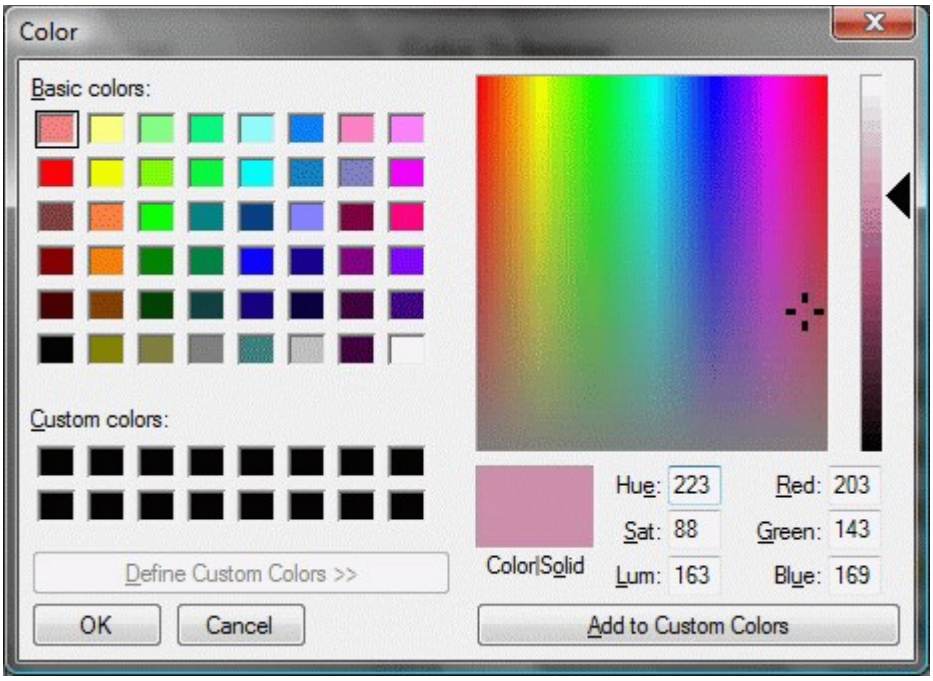

Figure 7. Color dialog window to add custom colour

  - Select a color and click **Add to Custom Colors** button.
  - A square with a chosen color appears in the **Custom colors** layout (on the left).
  - Click **OK** button.
- The color of the selected component is changed.

To top

Restoring Default Colors

If any default schema was modified, you can restore a default set of colors:

- Select an applicable default color schema in the list on the top section of the page.
- Press **Restore Defaults** button on the bottom section of the page.

To top

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5. General User Interface Elements

IDA-STEP is opening one workbench window for each XIM or STEP file. Like most other window applications IDA-STEP has a global menu, toolbar and a statusbar. Inside the main working area one or several views can show up. Each view may have its own individual toolbar. There is a wide range of possibilities on how the views can be arranged. A fixed collection of views at dedicated positions and sizes is called a perspective.

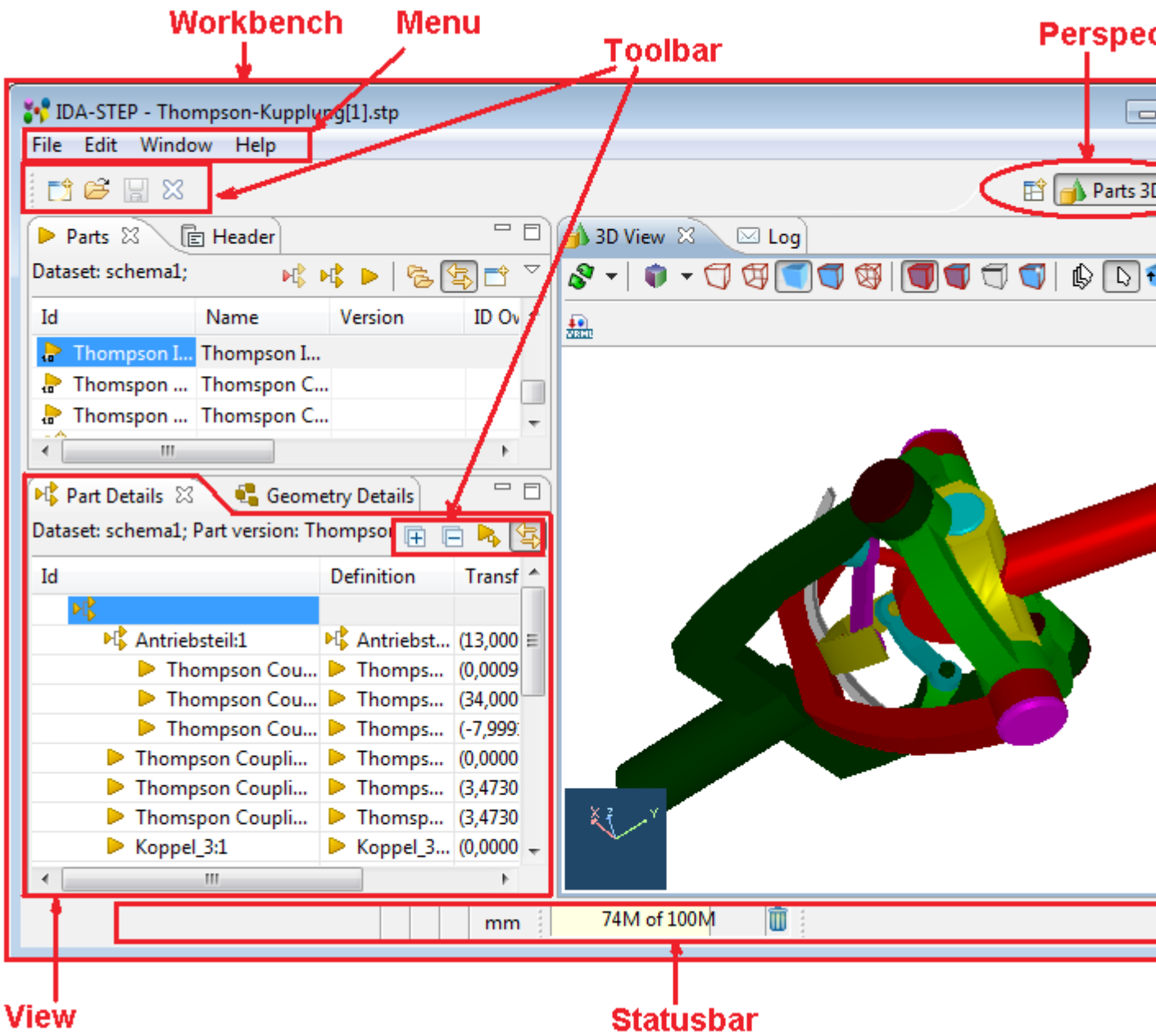

Figure 1. IDA-STEP Workbench

See also
- Workbench Window Layout
- Views
- Perspectives
- Toolbars
- Workbench Menus

To top

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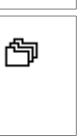
5.1. Workbench Window Layout

You can rearrange the layout of workbench windows as follows:

- Drag views to different positions within the workbench window.
- Drag views such that they are simultaneously visible beside, above, or below another editor.
- Resize views by dragging the sashes which separate them.

Drop Cursors

Drop cursors indicate where a view will dock when you release your mouse button. This indication is relative to the view or editor area underneath the cursor.

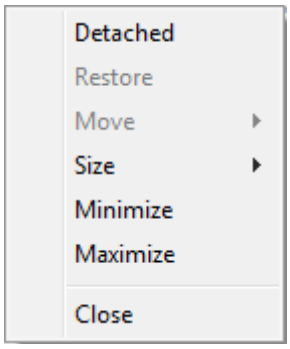
Cursor	Name	Description
	Dock above	The view will appear above the view underneath the cursor.
	Dock below	The view will appear below the view underneath the cursor.
	Dock to the right	The view will appear to the right of the view underneath the cursor.
	Dock to the left	The view will appear to the left of the view underneath the cursor.
	Stack	The view will appear as a tab in the same pane as the view underneath the cursor.

Double Click

Double-clicking a view title bar maximizes the part in the workbench window.

Title Bar Context Menu

From the context menu of a view title bar, you can select how you want the view to appear within the workbench window.



Context menu option	Description
Detached	The part becomes a detached view (hosted in its own shell).
Restore	Restores the view to its originating (non-maximized/non-minimized) size and position within the workbench.
ove	Move the part or part group.
Size	Change the size of the part in the direction specified.
Minimize	Minimizes the part in the workbench window.
Maximize	Maximizes the part in the workbench window.
Close	Closes the part.

To top

5.2. Views

Views provide alternative presentations or navigations of the information in the workbench. A view might appear by itself or stacked with other views in a tabbed notebook or a view may be closed and you can open it when needed.

To activate a view that is a part of a tabbed notebook simply click it's tab. The workbench provides a number of quick and easy ways to configure an environment, including whether the tabs are at the bottom or top of the notebooks.

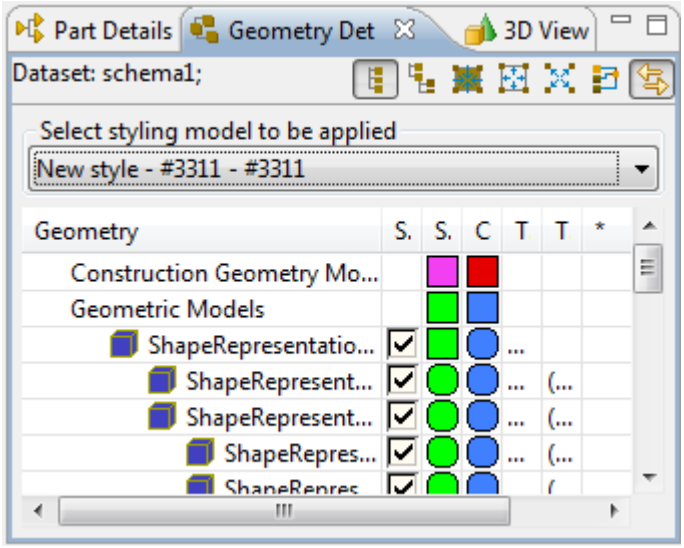


Figure 1. Tabbed views, Geometry Detaisl view is active view

Views may have a context menu menu, which is accessed by right clicking on the view's tab, allows the view to be manipulated.

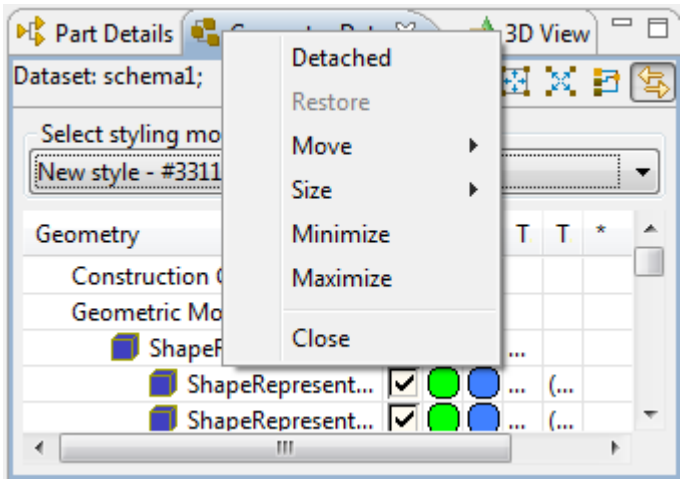


Figure 2. Title bar context sensitive menu

A view can be displayed by selecting it from the **Window > Show View** menu. A perspective determines which views may be required and displays these on the **Show View** sub-menu. Additional views are available by choosing **Other...** at the bottom of the **Show View** sub-menu. This is just one of the many features that provide for the creation of a custom work environment.

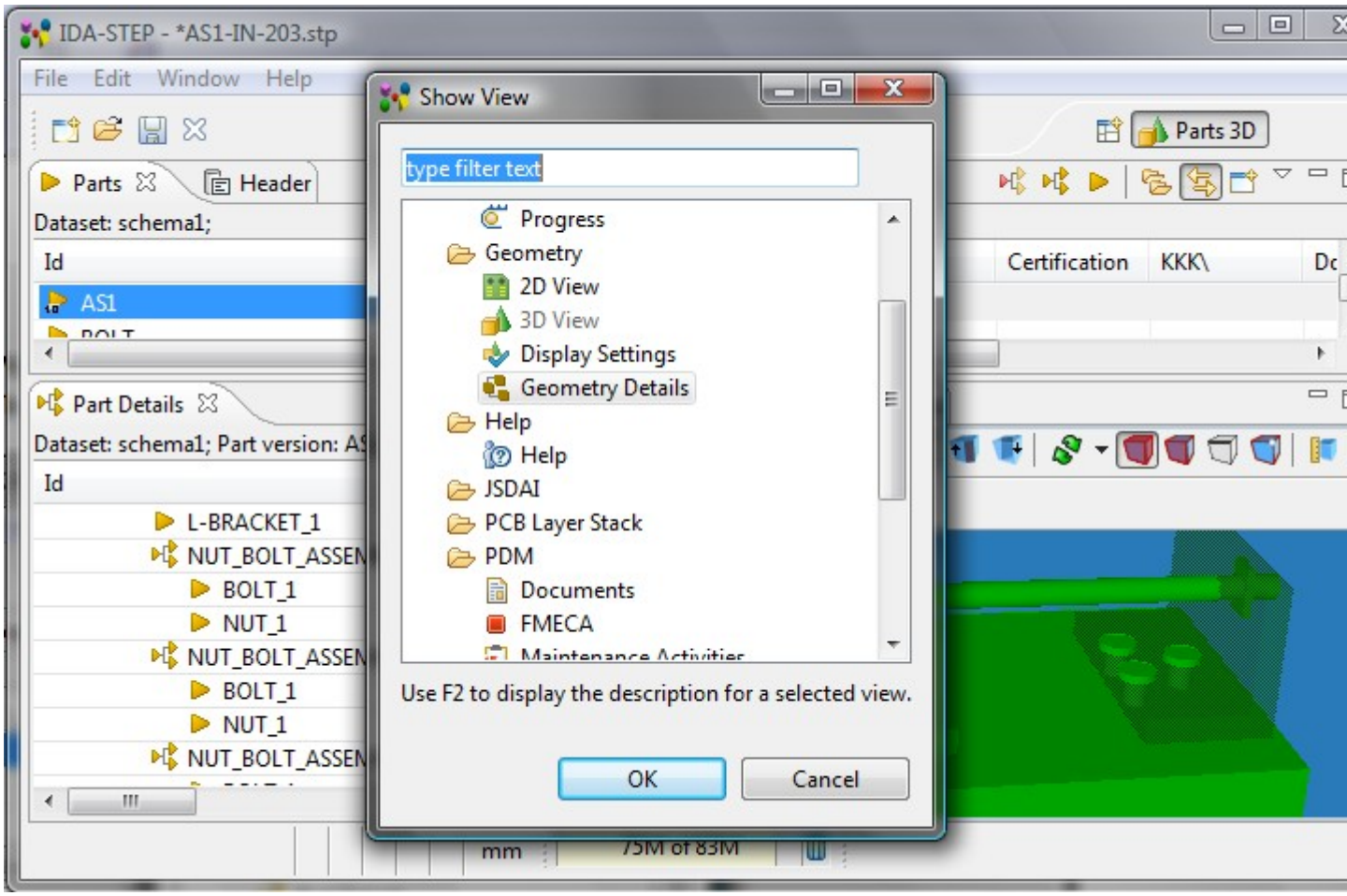


Figure 3. Dialog window invoked by **Show view --> Other** menu

To top

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5.2.1. Opening View

Perspectives offer pre-defined combinations of views. To open a view that is not included in the current perspective, select **Window > Show View** from the main menu bar. If a view is not listed in the appeared context menu, select **Other...** menu item. The **Show views** dialogue appears on the screen (see Figure 1).

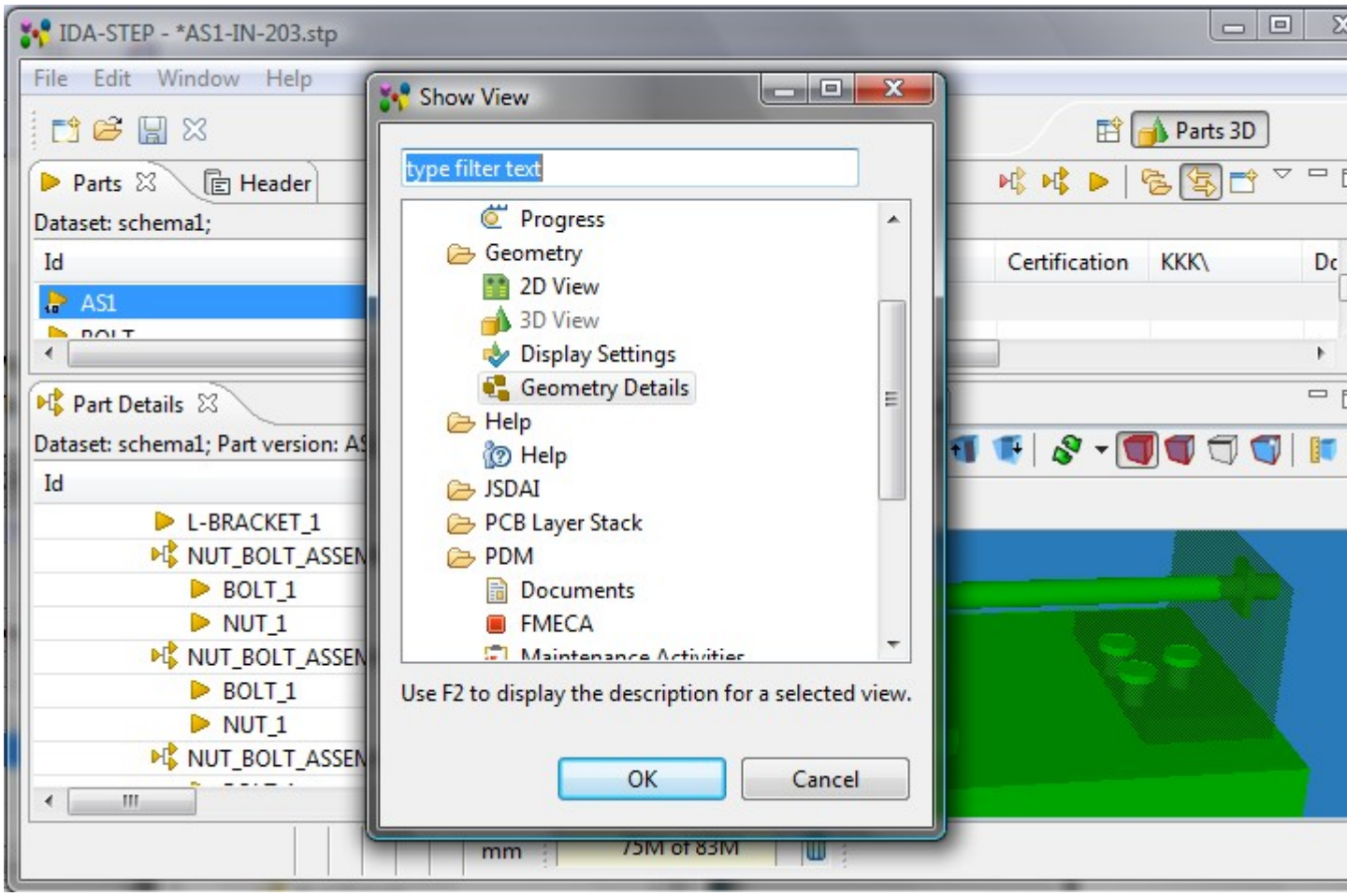

Figure 1. Show view dialogue

The **Show View** dialogue presents available views, classified under specific folders. Already active views are displayed with the grayed name labels. Use **F2** key to display the description for a selected view. The **OK** button closes the **Show View** dialogue and the selected view appears in the workbench.

After adding a view to the current perspective, you may wish to save your new layout (see New Perspective).

To top

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5.2.2. Moving and Docking View

The location of a view in the workbench is defined in a perspective. To change the location of a view:

- Press on a view tab and drag. **Note.** A group of stacked views can be dragged using the empty space to the right of the view tabs.
  The mouse pointer changes to one of the *drop cursors* shown in the table below. The drop cursor indicates where the view will be docked if you release the left mouse button. To see the drop cursor change, drag the view over the left, right, top, or bottom border of another view or editor. You may also drag the view *outside* of the workbench area to turn it into a "Detached" view.

Table 1. Drop cursors

Drop Cursor	View location

	Dock above	The view is docked above the view underneath the cursor.
	Dock below	The view is docked below the view underneath the cursor.
	Dock to the right	The view is docked to the right of the view underneath the cursor.
	Dock to the left	The view is docked to the left of the view underneath the cursor.
	Stack	The view is docked as a Tab in the same pane as the view underneath the cursor.
	Detached	The view is detached from the workbench window and is shown in its own separate windows.

- When the view is in the needed location, release the left mouse button.

The view or a stack of views may be moved by using context sensitive menu:

- Right click on a view title bar. The context sensitive menu appears on the screen:
  - Select **Move** menu item, then **View** - to move a view or **Tab Group** - to move stack of views.

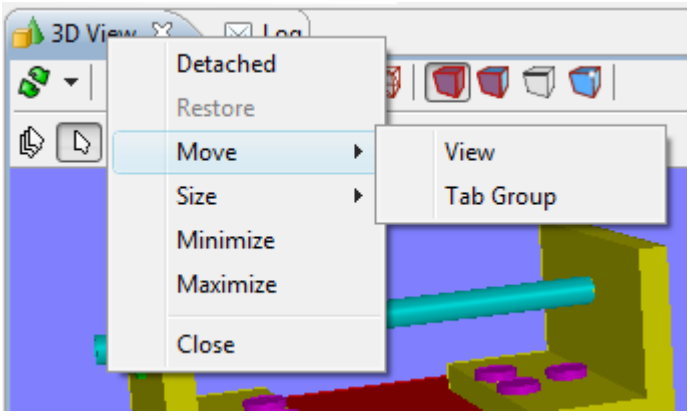

Figure 1. Context sensitive menu, Move menu item

- The *drop cursor* (see Table 1) and a dynamically moving square appears on the screen (see Figure 2).

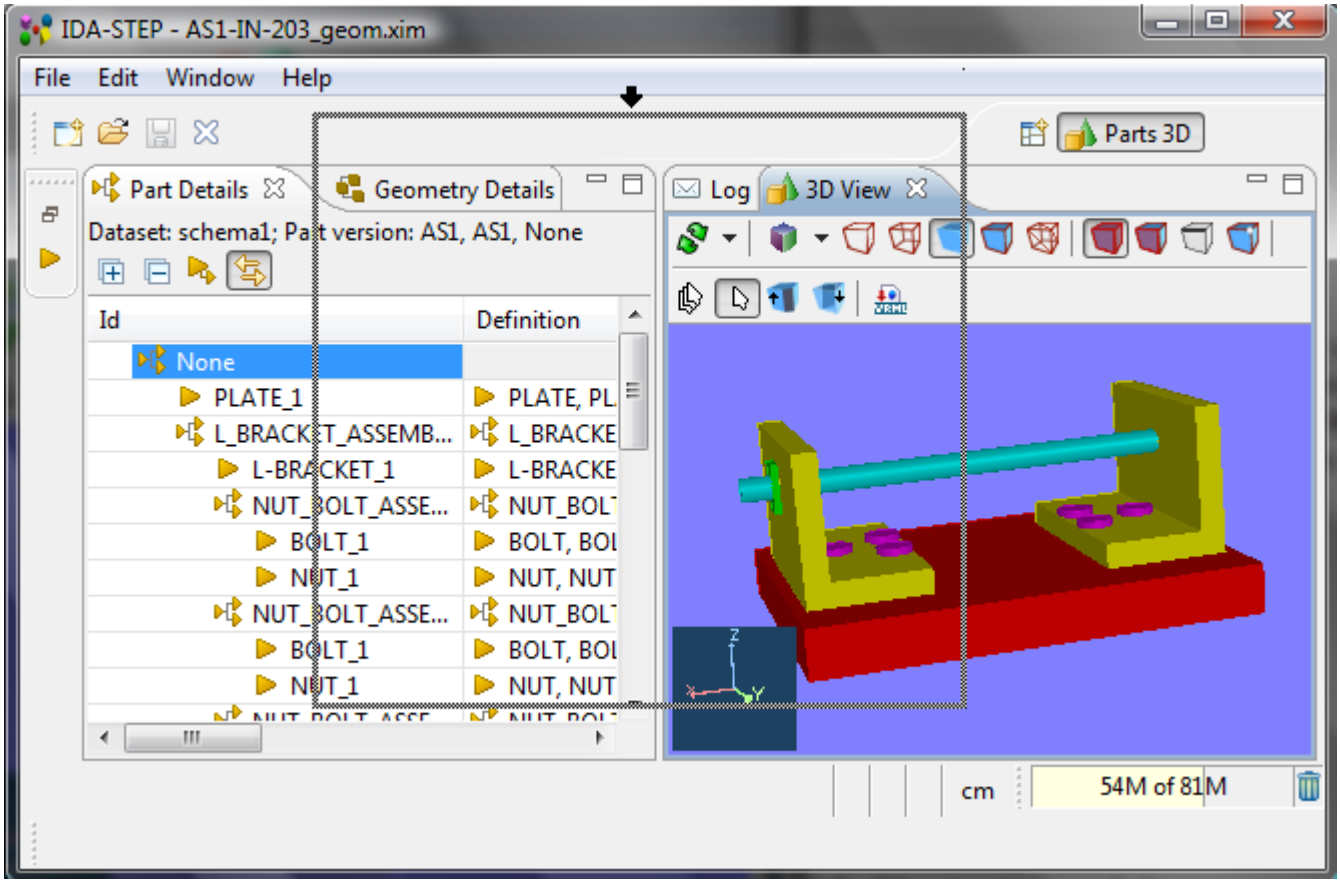

Figure 2. Drop cursor and dynamically moving square

- Move with a mouse to locate the selected view or stack of views and click a mouse button.

Note. You can save changes in a new perspective (see New Perspective for more details).

To top

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5.2.3. Detaching View

Detached views are views that are shown in a separate window with a smaller trim. They work like other views except they they are always shown in front of the workbench window (see Figure 1).

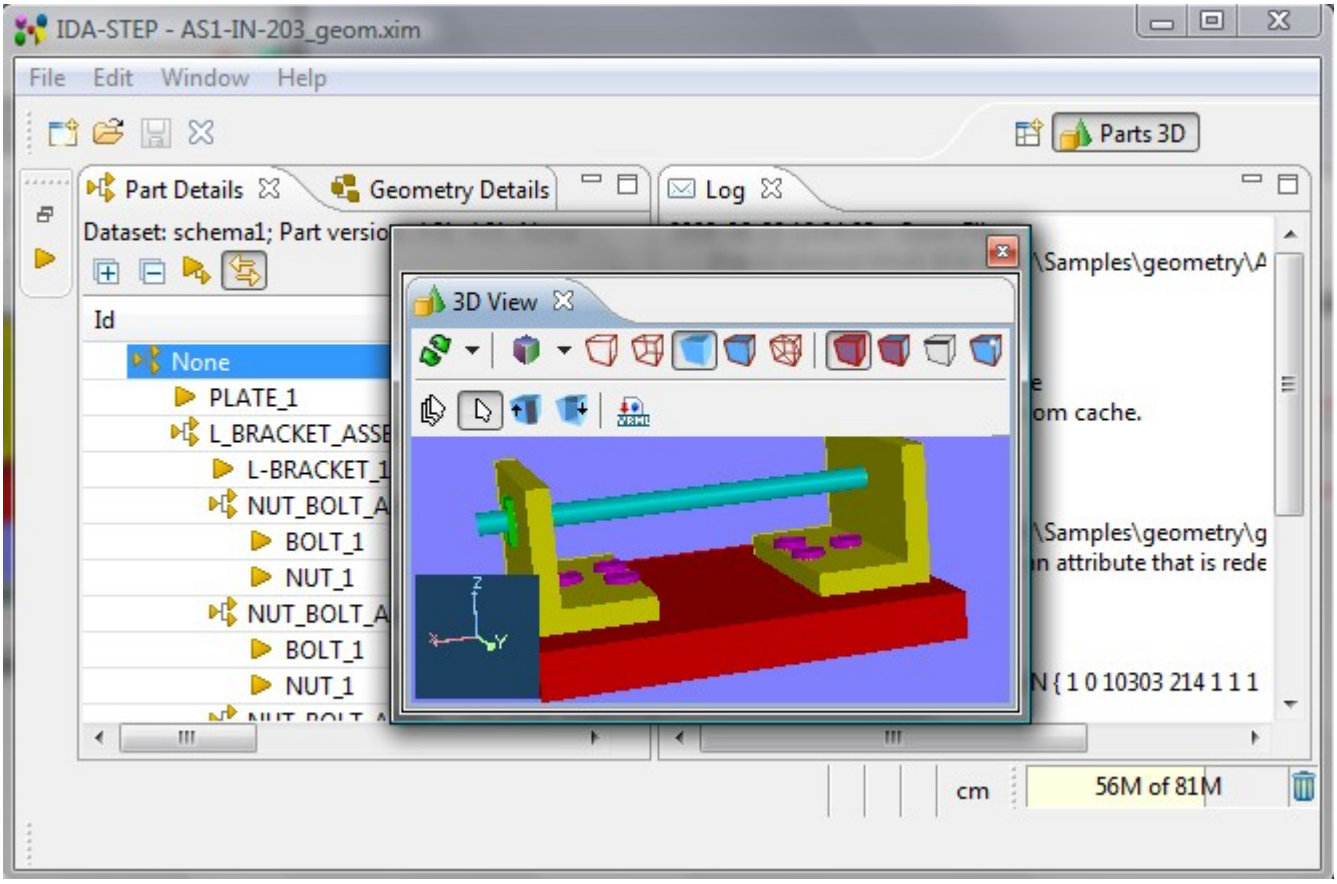

Figure 1. IDA-STEP workbench with detached 3D View

Follow the routine to detach a view:

- Right click on a view tab.
- Select **Detached** menu item in the appeared context sensitive menu (see Figure 2). The selected view is detached.

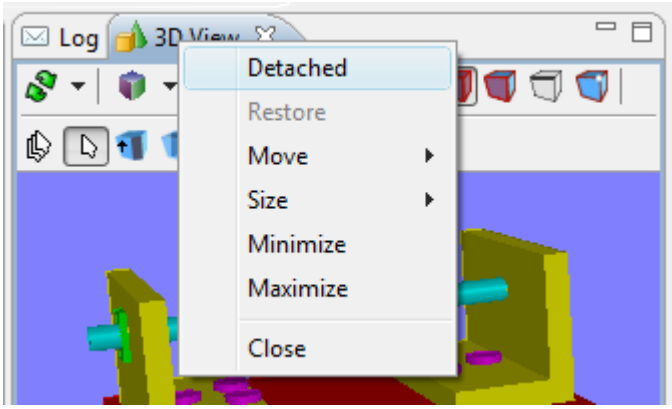

Figure 2. Context sensitive menu, Detached menu item

To attach the detached view back to the workbench:

- Right click on the detached view tab.
- Select **Detached** menu item in the appeared context sensitive menu. The view is returned to the initial position in the workbench.

The view may be detached by moving it outside the workbench window (see Moving and Docking View for more details). To restore the view to be shown inside of the workbench window, drag it into the workbench window.

To top

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5.2.4. Minimizing and Maximizing View

A view in a workbench occupies an area, defined in a perspective. Some time it is convenient to have a *maximized* view, which takes as much area in the workbench as it can. Or some times it is needed to *minimize* a view temporary, the view takes an area as small as it can.
There are some ways to manage minimizing and maximizing of a view area:

1. Using the minimize and maximize buttons provided on a stack's border;
2. Selecting the 'Minimize' or 'Maximize' item on the context sensitive menu for a stack;
3. Double-clicking on a stack;
4. Using 'Ctrl + M' keys: this is a key binding for a command that will toggle the currently active view between its 'maximized' and its 'restored' (i.e. normal) states.

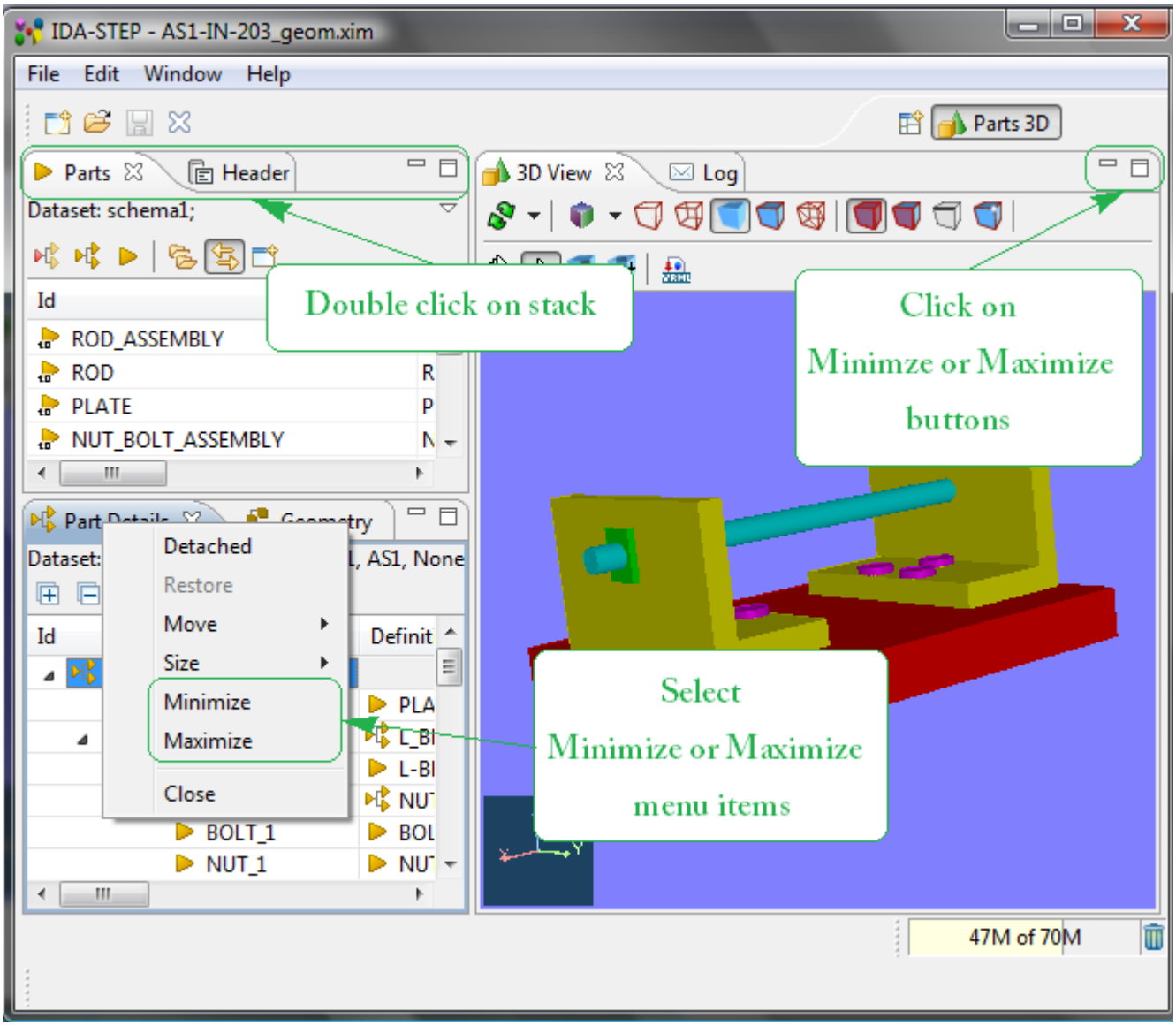

Figure 1. Minimizing/Maximizing ways

Maximize

It is desirable at times to focus your attention on one particular view to the exclusion of the others. The most popular candidate for this is maximizing the view area in order to make as much of the display

available for editing/viewing as possible (but there are workflows where it would make sense to focus on a view as well).
IDA-STEP implements the maximize behavior by minimizing all stacks *except* the one being maximized. This allows the maximized stack to completely occupy the main presentation while still allowing access any open views in your perspective by using the icons in their *Trim Stack* (the area around the edges of the window is called the 'trim').

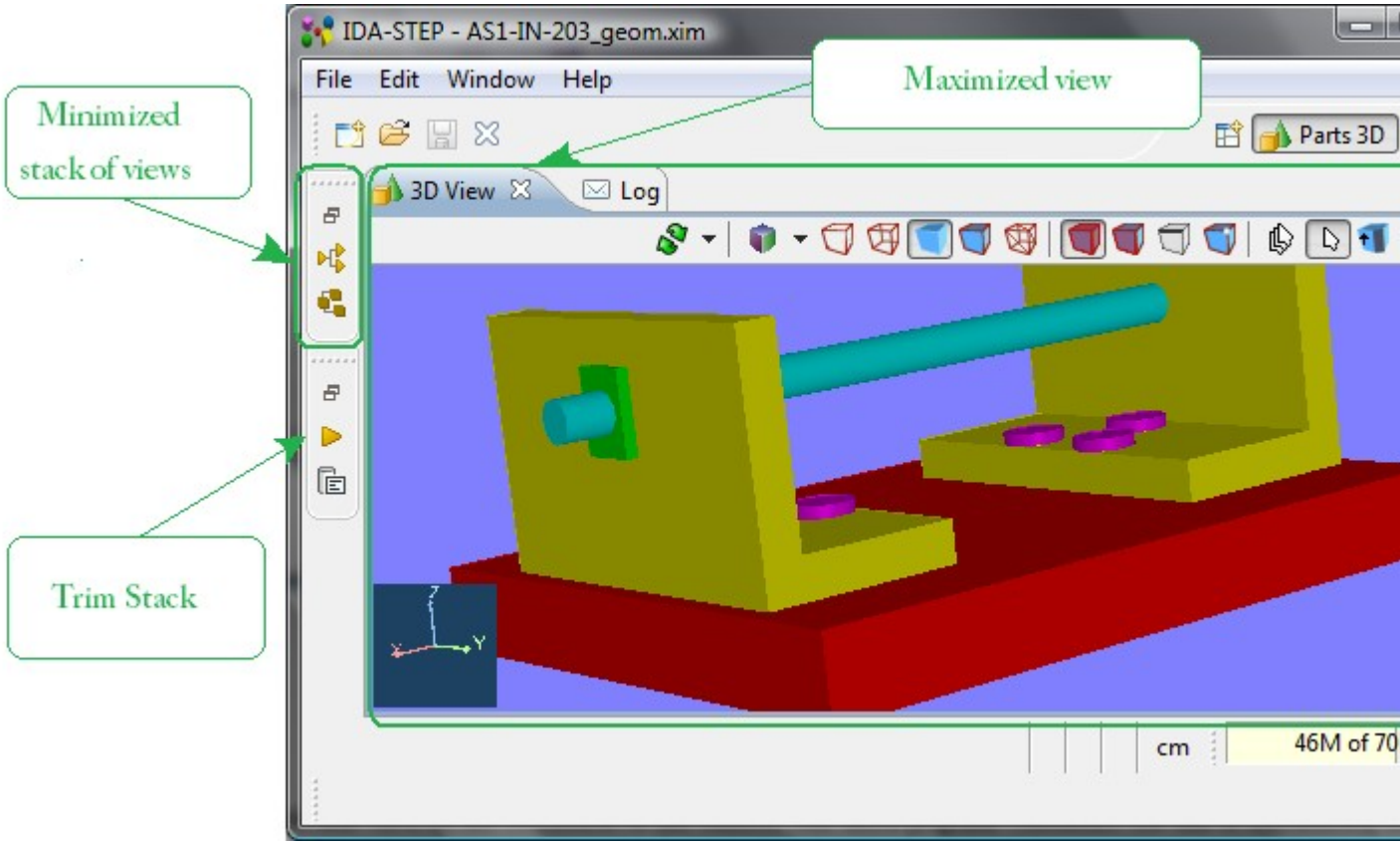

Figure 2. Maximized view in workbench

To top

Minimize

Another way to optimize the use of the screen area is to directly minimize stacks that are of no current interest. While minimizing a view or view stacks is moved into the trim area at the edges of the workbench window, creating a *Trim Stack*. View Stacks get minimized into a trim representation that contains the icons for each view in the stack (see Figure 3).

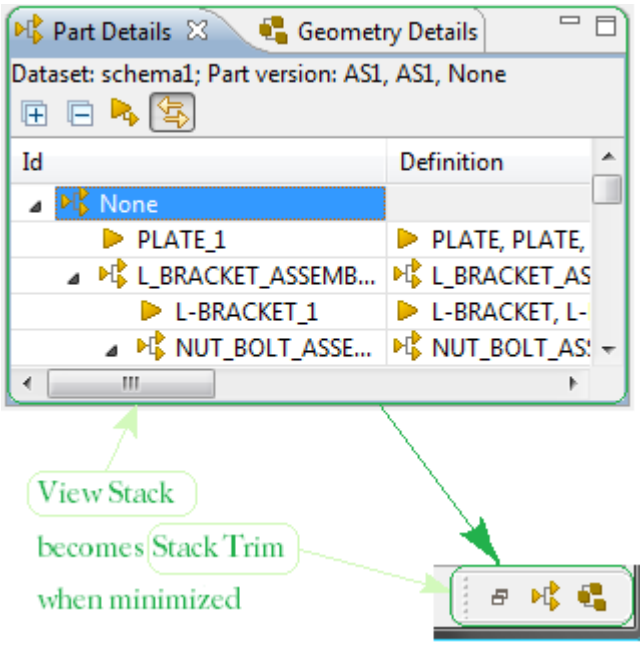

Figure 3. Stack View minimized to Stack Trim

If your particular workflow is such that you need to have more than one element (i.e. having the view area *and* a View Stack in the presentation at the same time) you can still gain additional screen space by minimizing the stacks that aren't of current interest. This will remove them from the main presentation and place them on the outer edge of the workbench window as *Trim Stacks*, allowing more space for the remaining stacks in the presentation (see Figure 2).

Note: There are two ways to end up with a stack in the trim:

- Directly minimizing the stack
- As the result of another stack being maximized

Depending on how the Trim Stack was created its behavior is different; when un-maximizing only those trim stacks that were created during the initial maximize will be restored to the main presentation while stacks that were independently minimized stay that way.

This difference is important in that it allows you fine grained control over the presentation. While using maximize is a one-click operation it's an 'all or nothing' paradigm (i.e. no other stack is allowed to share the presentation with a maximized stack). While adequate for most tasks you may find yourself wanting to have the presentation show more than stack. In these scenarios don't maximize; minimize all the other stacks *except* the ones you want in the presentation. Once you have it set up you can still subsequently maximize the editor area but the un-maximize will only restore the particular stack(s) that were sharing the presentation, not the ones you've explicitly minimized.

To top

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5.2.5. Fast Viewing

When views are minimized in to stack trims, they can be quickly opened and closed. Views are represented as a toolbar buttons. You can read a name of a particular view in the tool tips, which appear after the mouse is moved over a view icon button (see Figure 1).

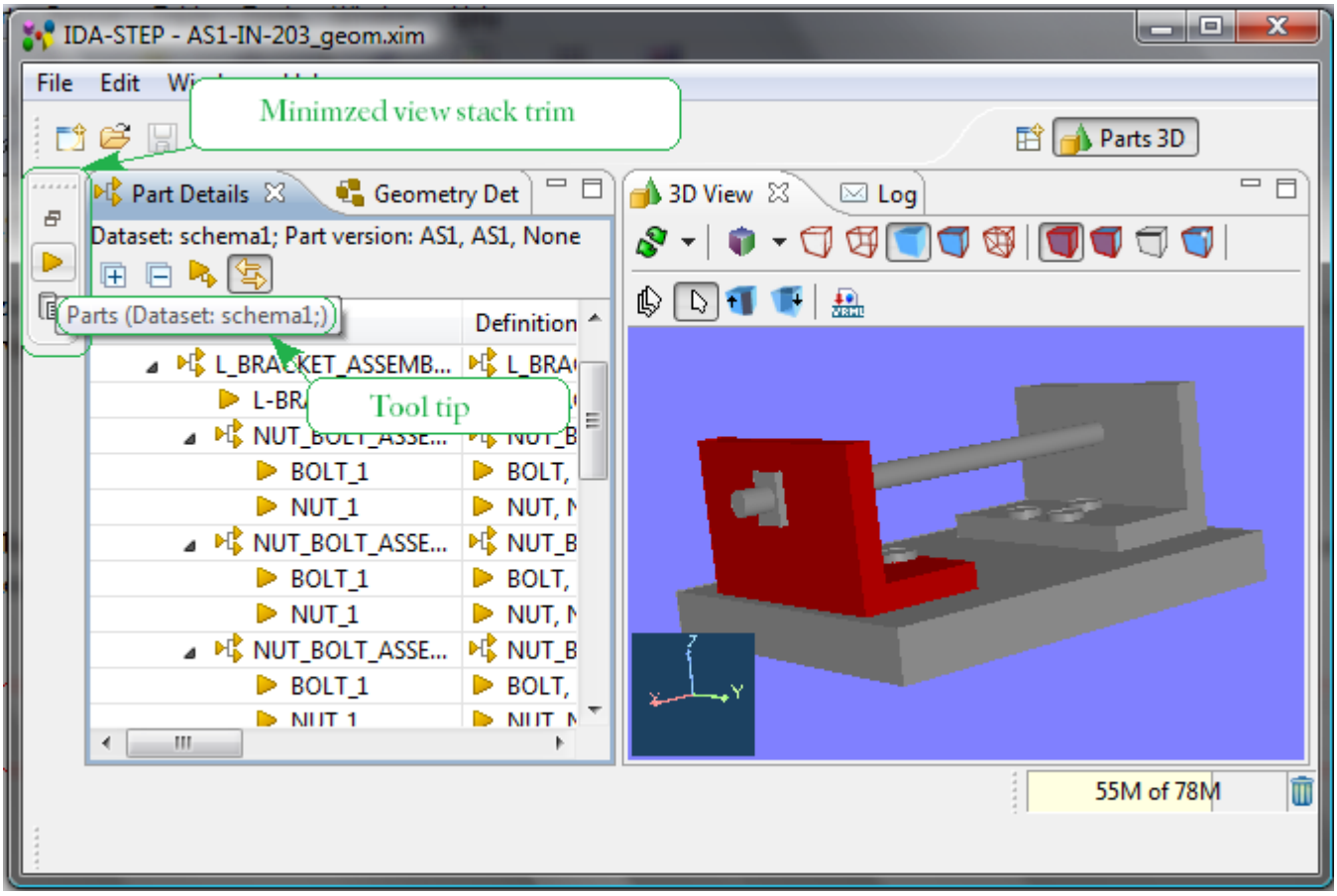

Figure 1. IDA-STEP workbench view with stack trim and view button tool tip

When you click the toolbar button for a particular view, that view opens temporarily in the current perspective, overlaying the area it takes in a normal workbench presentation (see Figure 2). As soon as you click outside that view it is hidden again. The stack trim can also be docked on the other sides of the workbench window.

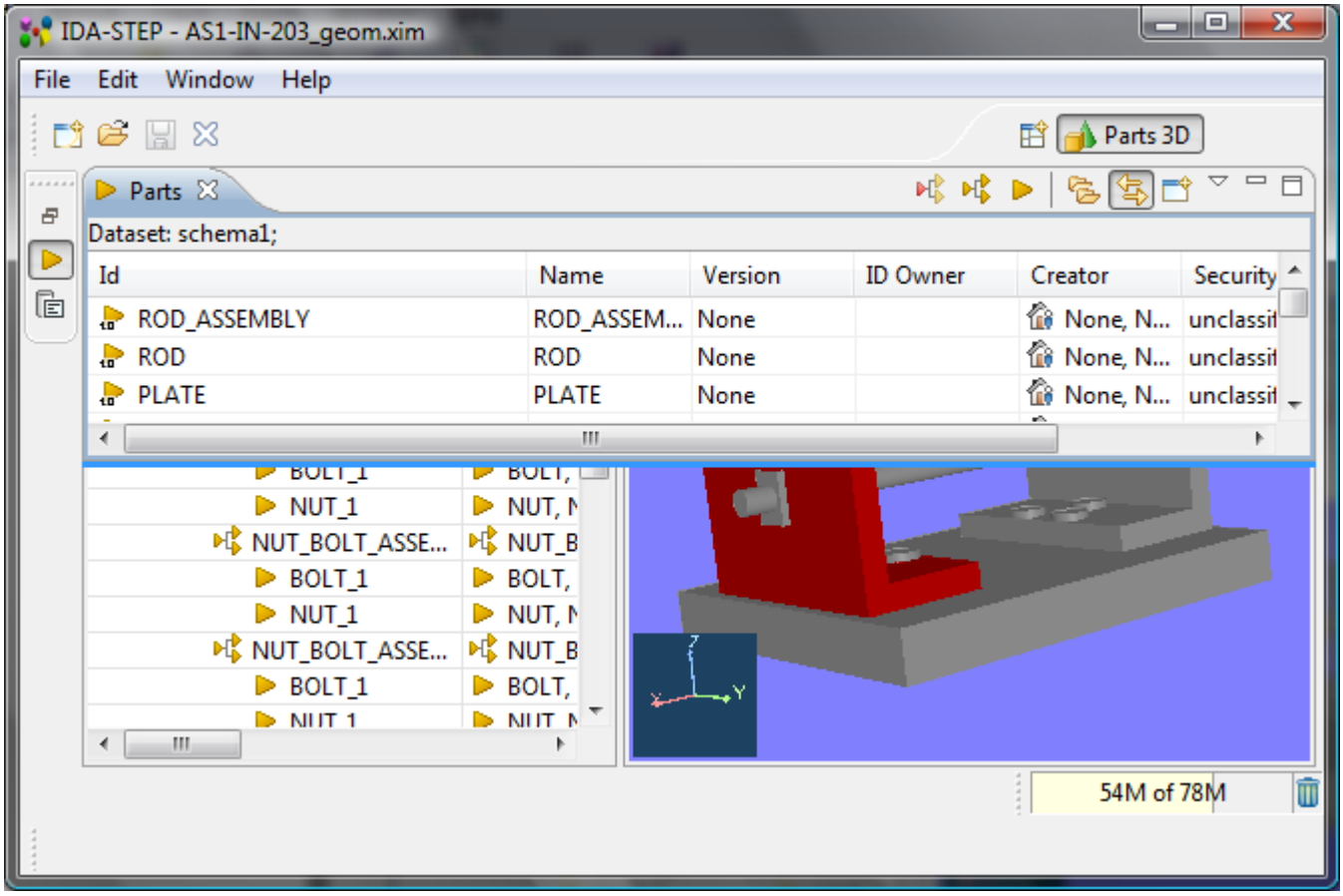

Figure 2. Temporary opened Parts view

See also Minimizing and Maximizing View.

To top

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5.2.6. Resizing View

A size of a view in the IDA-STEP workbench may be changed in the following ways:

- Move a mouse pointer over the edge of a view. The *resize* mouse pointer appears.
- Press the mouse button, hold and drag to available direction to resize the view area (see Figure 1).
- Release the mouse button after the needed size of the view area is reached.

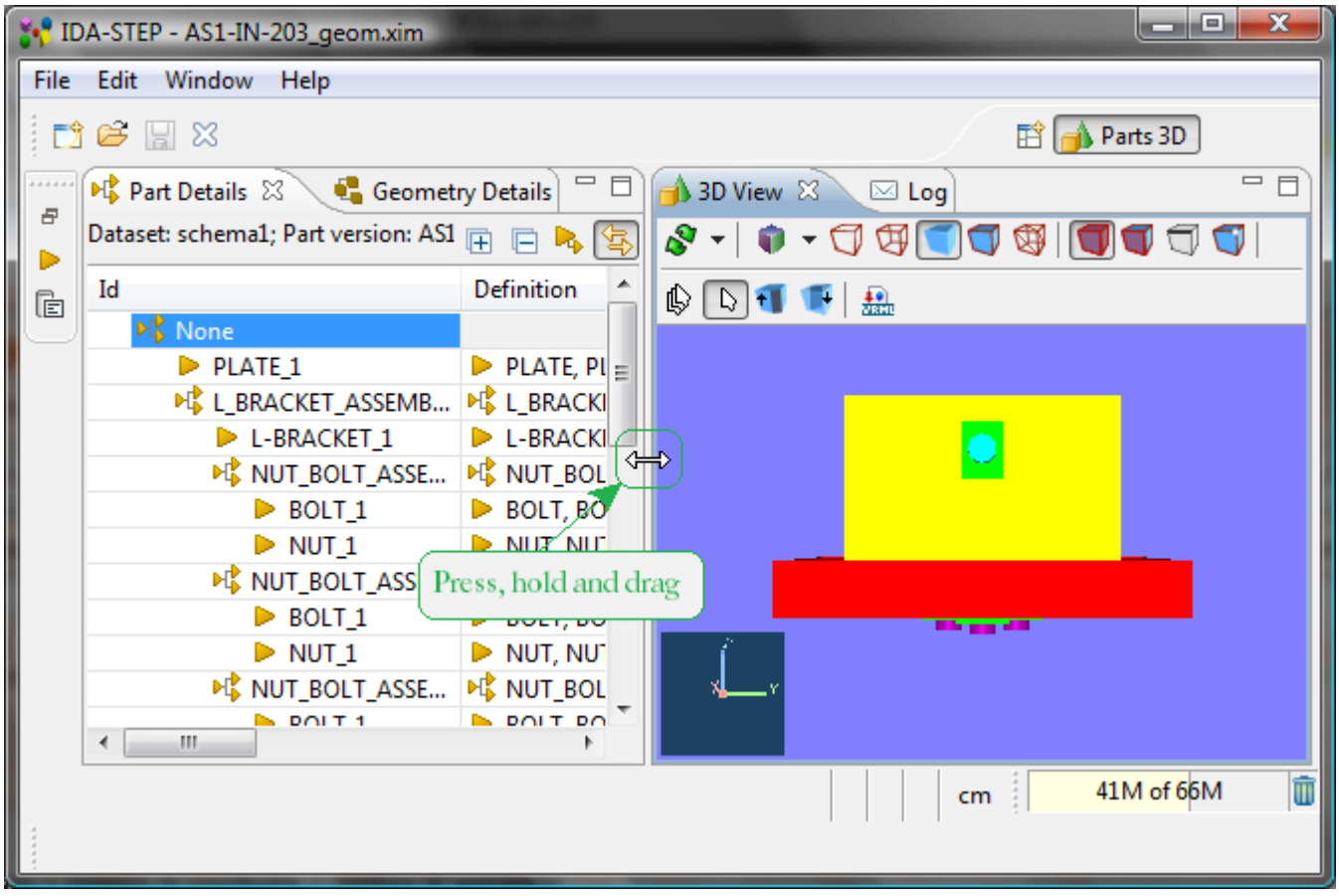


Figure 1. IDA-STEP workbench with resize mouse pointer

Another way to change size of a view is by using context sensitive menu:

- Right click on a view stack bar. The context sensitive menu appears on the screen.
- Select **Size -> <direction>** menu item. Note. Only currently available resizing direction is enabled (see Figure 2).

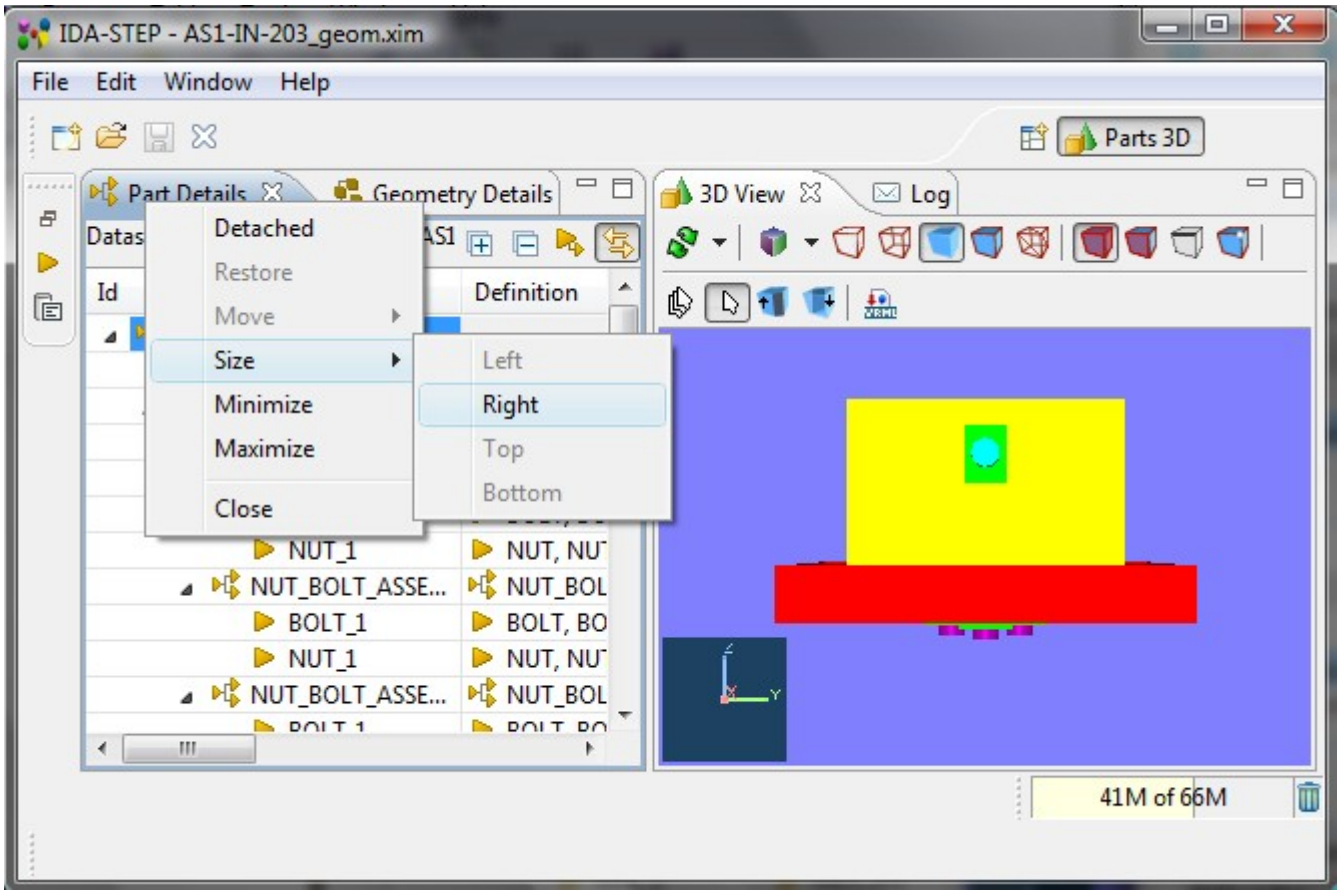
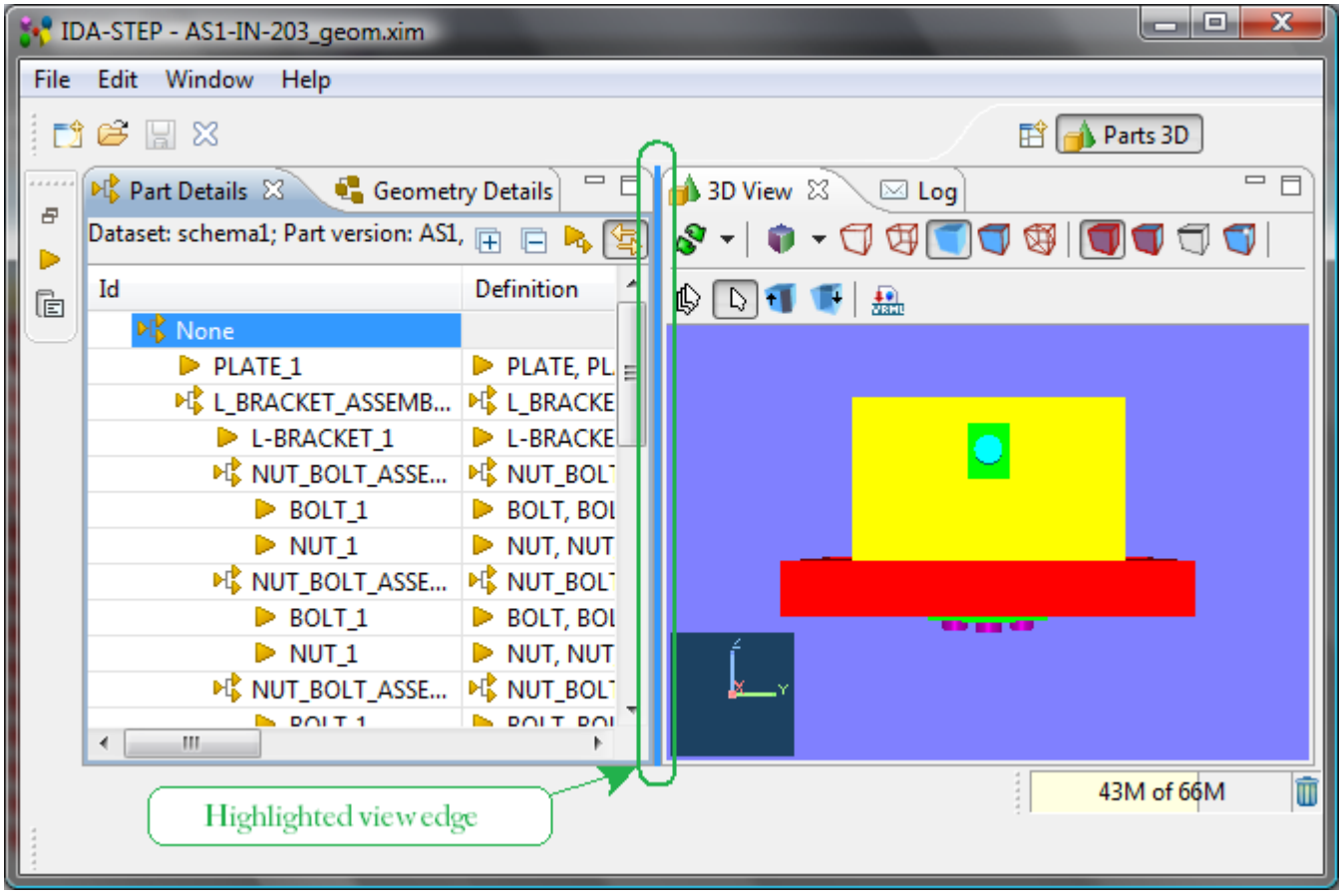


Figure 2. Context sensitive menu to resize view

- After an applicable resizing direction is selected, the particular edge of the view are is highlighted on the workbench (see Figure 3).



- Point with a mouse on the highlighted line. The *resize* mouse pointer appears. Press, hold and drag

to the needed direction.
- Release the mouse button after the needed size of the view area is reached.

To top

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5.2.7. Find Text

The **Find** functionality allows you to quickly find and highlight text in a currently active view.

To find a particular text in a view:

- Select in the main menu bar **Edit -> Find...** menu item.
  Or
- Use key combination **Ctrl+F**.

The **Find** dialog appears on the screen (see Figure 1).

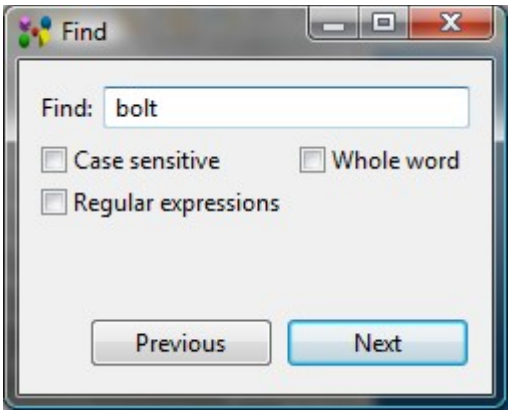

Figure 1. Find dialog window

Enter the text to find into the **Find** field and set an applicable options for the search. Use buttons **Previous** or **Next** to searh for the entered text.
Note. After any button is pressed, the dialog is closed and the found text is highlighted in a view. To find the next occurrence of the search string, you have to open the **Find** window again. The previously eneterd string should be present in the **Find** field.

Options

- **Case sensitive** - check this option to match the capitalization you have entered.
- **Whole word** - check this option to match the entire word entered as the search term.
- **Regular expression** - check this option to match the regular expression entered as the search item.

Directions

To select the search direction - use one of the buttons:

- **Previous** - search through the records following the current record.

- **Next** - search through the records preceding the current record.

To top

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5.2.8. Sorting

IDA-STEP presents a possibility to arrange data in an ascending or descending order. Data can be sorted in an alpha-numerical order. The elements are sorted from left to right, i.e. a1, a2, a10, a11 and etc. This functionality is implemented only for those tree tables, where sorting is meaningful - the sorting does not damage the data structure. If the order of elements is strongly defined, the sorting of data is not available (i.e., **Passages view**, because stratums displayed under passage node have always to be sorted according to their order in design; **Tasks view**, because the order of elements in that view is important).

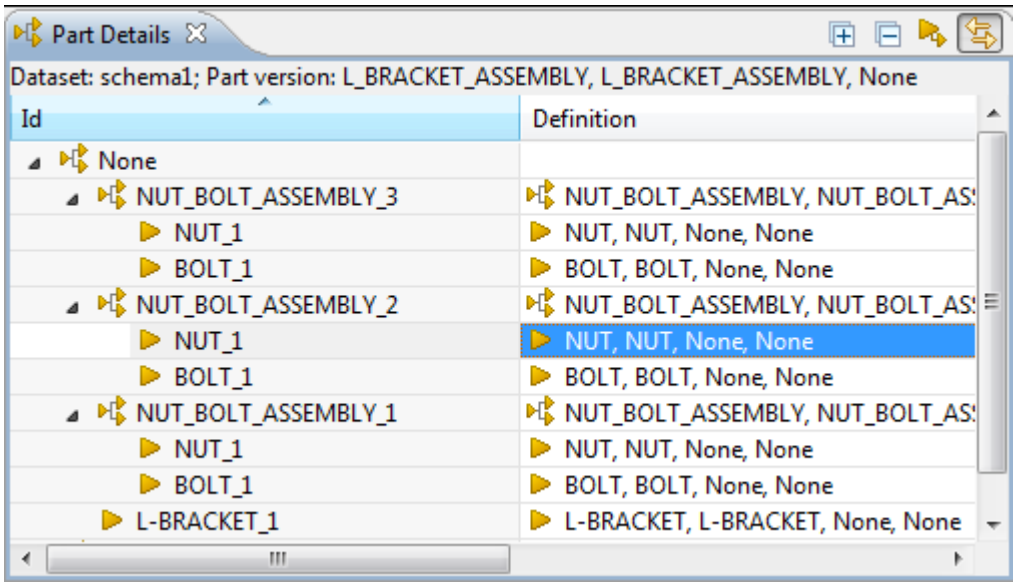

Figure 1. Part Details view, tree table with sorted assemblies

To sort elements in a view:

- Activate a view with a tree table.
- Press with a mouse on a column to be sorted title. Data in the selected column is sorted (see Figure 1).

The sorting direction is indicated in the column title:

- Ascending order (triangle arrow up),
- Descending order (triangle arrow down).

To top

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5.3. Perspectives

A perspective defines the initial set and layout of views in the workbench window. One or more perspectives can exist in a single workbench window.

Perspectives define visible action sets, which can be changed to customize a perspective. A perspective that is built in this manner can be saved, creating a custom perspective that can be opened again later.

The workbench window displays one or more perspectives. Each product determines initially what default perspective is displayed, in this example it is the **Parts 3D** perspective (see Figure 1). The predefined perspectives are:

1. Parts 3D,
2. GD&T,
3. PCB/PCA,
4. PCB Layer Library,
5. PCB Layer Stack,
6. Tasks.

More than one workbench window can be open at any given time.

IDA-STEP presents functionality to choose perspective upon loading/importing of a file according to data in the file. The particular perspective is proposed in the dialog window after a STEP file is loaded/imported. After the user selects a perspective, it is opened in the workbench and in some cases the "key" view is activated as well. If there is only one perspective to choose for the selected file, it can be launched automatically in the future. This can be managed by selecting particular preferences in the **Perspectives** page (accessed by selecting **Window-> Preferences-> General-> Perspectives**).

The right click on the perspective button in the perspective bar invokes the context sensitive menu, which presents the following perspective options:

- **Customize** - customize the active perspective (reference Customize Perspective);
- **Save as** - create a new perspective or update the existing one (reference New Perspective);
- **Reset** - reset the active perspective to it's default layout (reference Reset Perspective);
- **Close** - close the currently selected perspective.

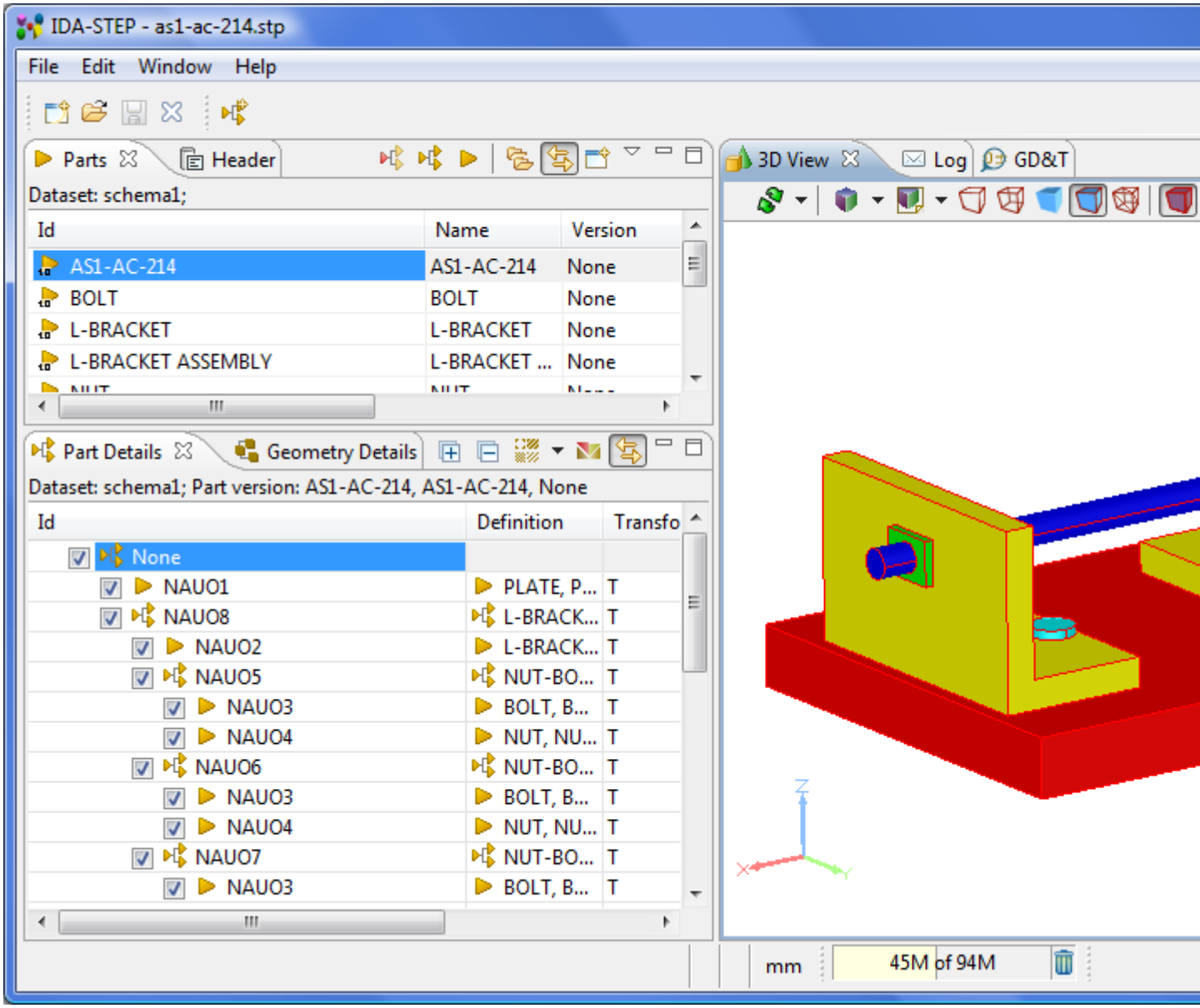

Figure 1. Parts 3D perspective

To top

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5.3.1. Open Perspective

There are several ways to open a new perspective within this workbench window:

- Using the Open Perspective button on the shortcut bar.
- Choosing a perspective from the **Window > Open Perspective** menu.

To open one by using the shortcut bar button:

- Click on the **Open Perspective** button .
- Choose **Other** from the menu. The **Open Perspective** dialog window appears (see Figure 1).

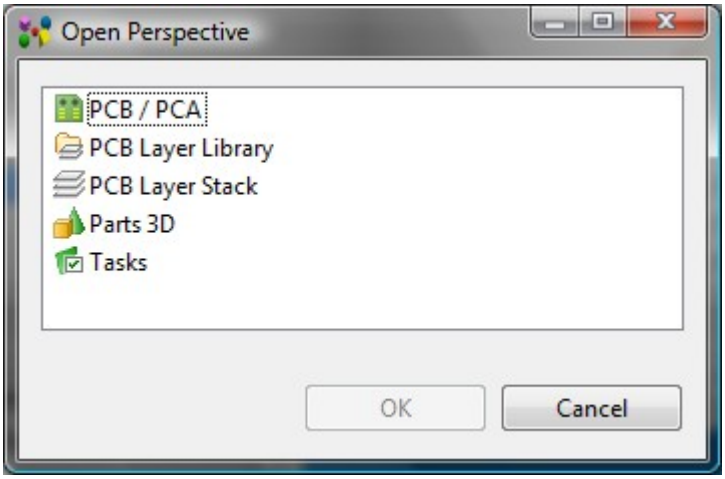

Figure 1. Open Perspective dialog window

- Select an applicable perspective from the list and click **OK** button. The selected perspective is displayed in the workbench.
  The title of the window now indicates which perspective is in use.The shortcut bar contains several perspectives. The selected perspective button is pressed in, indicating that it is the current perspective (see Figure 2).

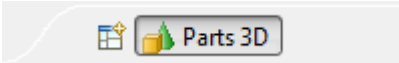

Figure 2. The current perspective button

- To display the full name of the perspective right click the perspective bar and check **Show Text**.

To top

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5.3.2. New Perspective

There is a possibility to add new views to the perspective and rearrange the views. The workbench also allows this layout to be saved for future use.

- Open an existing perspective in a workbench.
- Add new views on it.
- Right click on the active perspective button in the perspective bar. Select the **Save As..** menu item from the appeared context sensitive menu (see Figure 1).

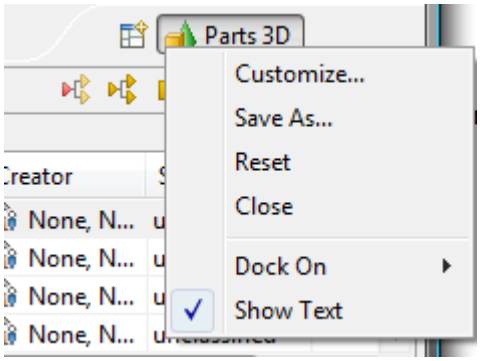

Figure 1. Context sensitive menu

- The **Save Perspective As...** dialog window appears (see Figure 2).

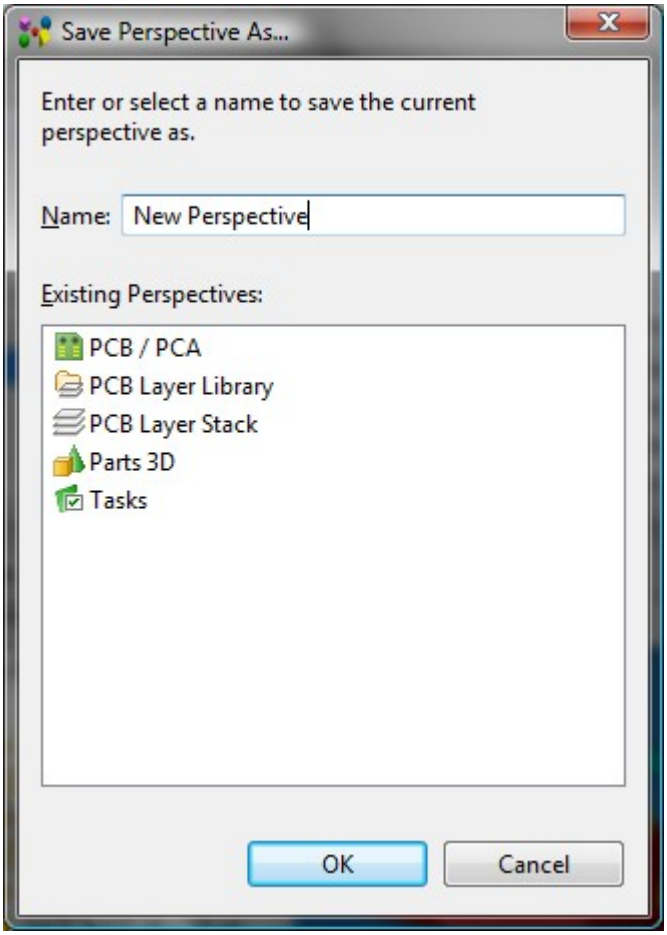

Figure 2. Save Perspective As... dialog window

- Enter a new name of the perspective to be created into the **Name** field or select one of the perspectives in the **Existing Perspectives** list. If an existing perspective is selected, the dialog appears informing, that the selected perspective already exist (see Figure 3). You have to confirm or cancel and enter a new name.

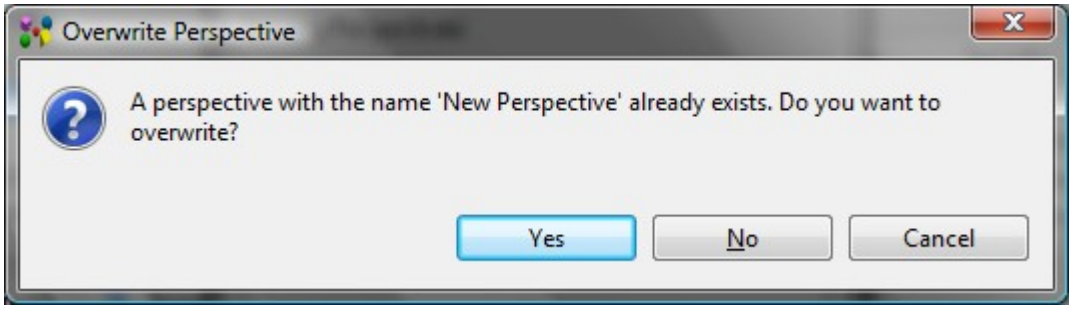

Figure 3. Overwrite Perspective dialog window

- Click **OK** button to confirm your selection and exit the dialog window.

- The new name of the perspective appears in the perspective bar (see Figure 4). The icon of the new perspective is the same as for the latest active perspective.

Figure 4. New perspective button in the perspective bar

To top

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5.3.3. Customize Perspective

In addition to configuring the layout of a perspective you can also control several other key aspects of a perspective. These include:

- The visibility of individual menu and tool bar items.
- The availability of command groups (groupings of menu items, tool bar items and key bindings).
- The items in the W**indow --> Open Perspective**, **Show View** menus.

To customize the current perspective:

- Right click on the perspective button in the perspective bar.
- Select the **Customize** menu item from the appeared context sensitive menu. The **Customize Perspective <current perspective name>** dialogue window appears (see Figure 1):

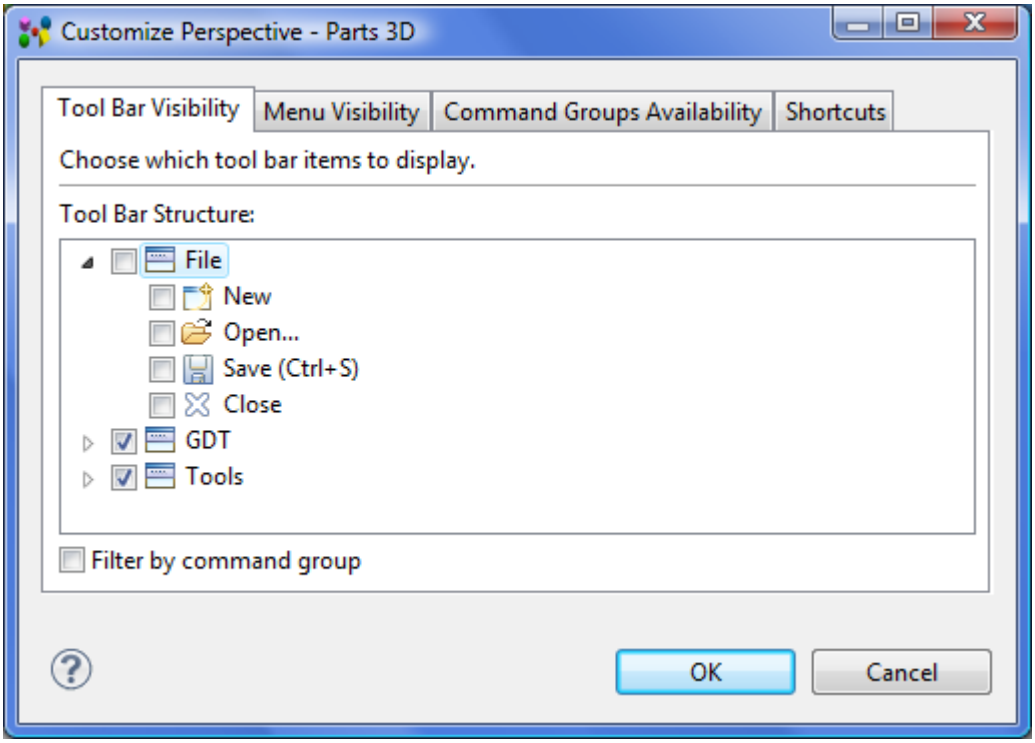

Figure 1. Customize Perspective Parts 3D dialogue window with active Tool Bar Visibility panel

Tool Bar Visibility

Choose which tool bar items to display. The checked box in front of a particular tool bar button icon means that this tool bar button is displayed in the tool bar. If the check box is empty - the particular button or group of buttons is not displayed in the tool bar.

- Select the **Tool Bar Visibility** tab.
- Uncheck the **File** item in the **Tool Bar Structure** tree.
- Press **OK** button.

The **File** group of buttons is no longer included in the main tool bar (see Figure 2).

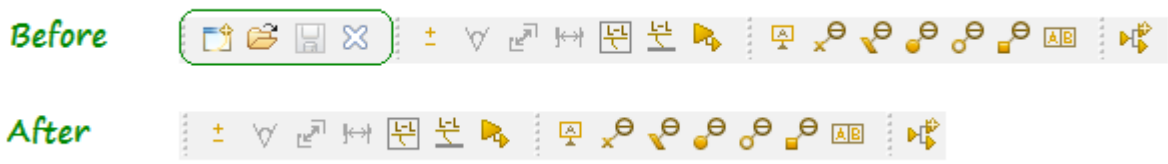

Figure 2. Main tool bar with File group buttons and tool bar after it was customized

Menu Visibility

Choose which menu items to display. The checked box in front of a particular menu item means that this menu item is displayed in the main menu bar. If the check box is empty - the particular menu item is not displayed in the main menu bar.

- Select the **Menu Visibility** tab
- Uncheck the **File-> Open** item in the **Menu Structure** tree.
- Press **OK** button.

The **GDT-> Create Constructive Geometry** item is no longer included in the main menu (see Figure 3).

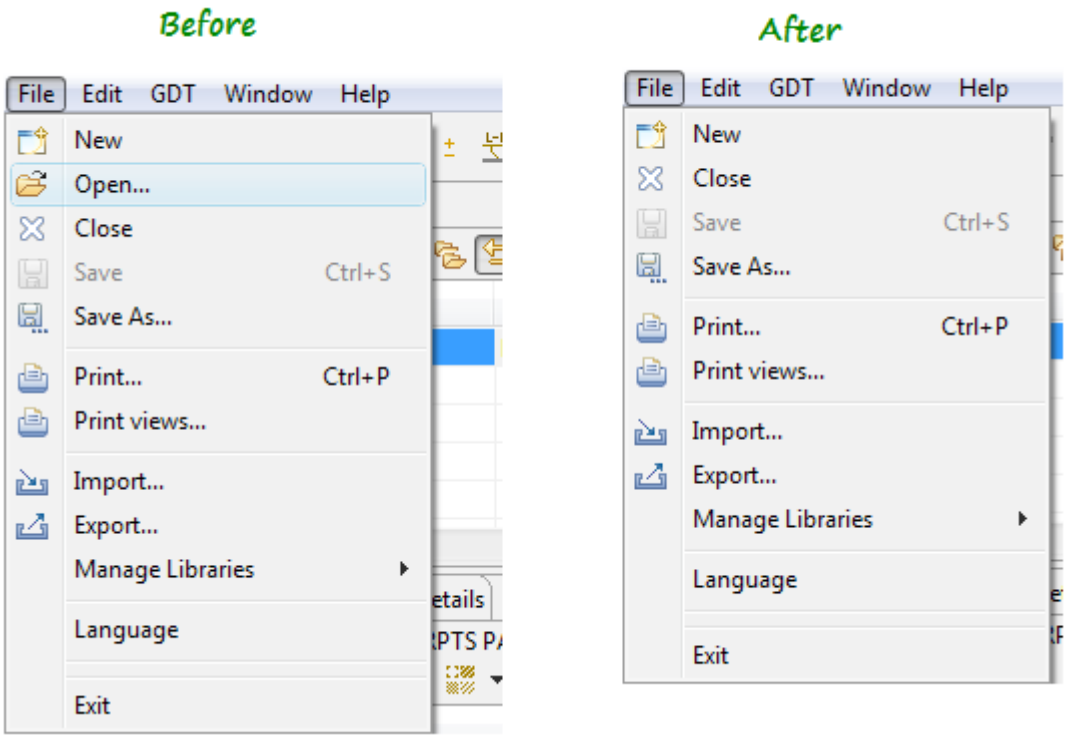

Figure 3. Menu with File->Open menu item and menu after it was customized

Command Groups Availability

Select the command groups that you want to see added to the current perspective. The details field identifies which menu items and/or tool bar items are added to the perspective by the selected command group.

Shortcuts

Select the shortcuts that you want to see added as cascade items to the selected sub menus. The selections made will only affect the current perspective.

- Click on the **Shortcuts** tab to activate the panel.
- Select an applicable sub menu in the **Sub menus** combo box (i.e. *Open Perspective*).
- Unselect shortcuts, which should not appear in the selected sub menu, by unselecting a particular

check box in front of the shortcut name (see Figure 4).

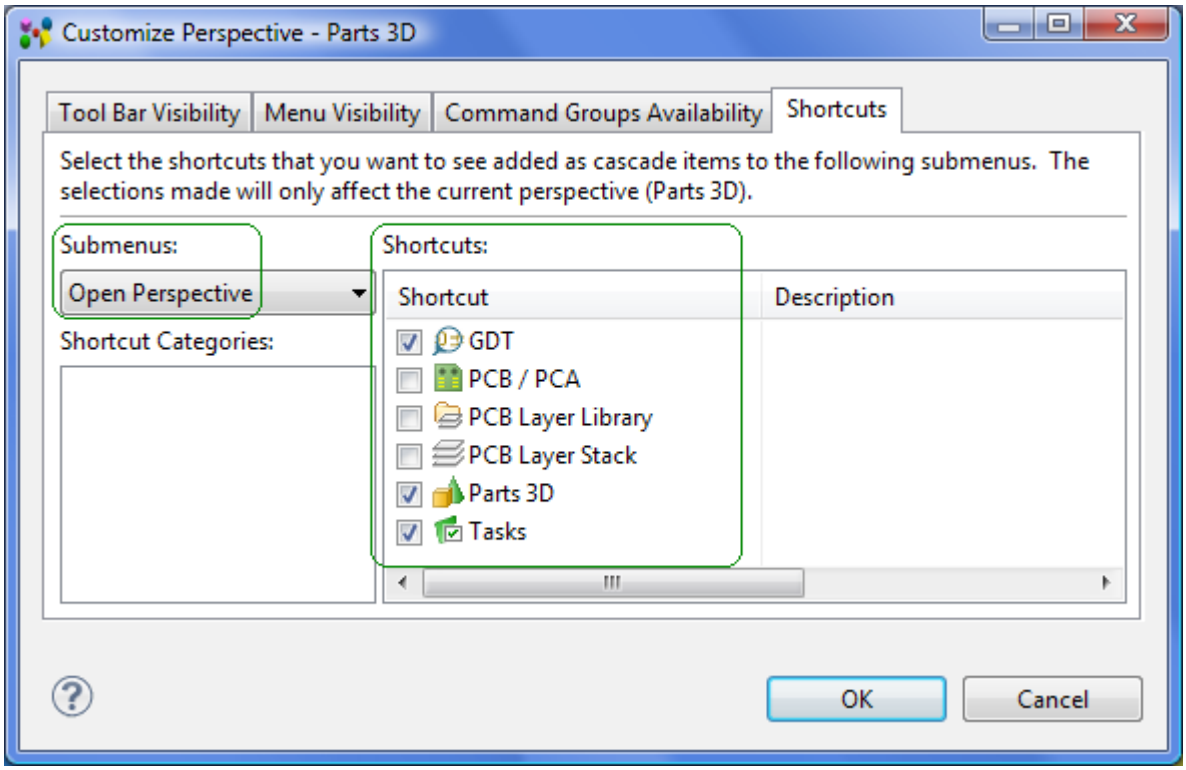

Figure 4. Customize Perspective Shortcuts wizard page with selected sub menu Open Perspective and list of available shortcuts

- Click **OK** button to confirm your selections and exit the dialogue window.

The unselected shortcuts are no longer displayed in the selected sub menu (see Figure 5).

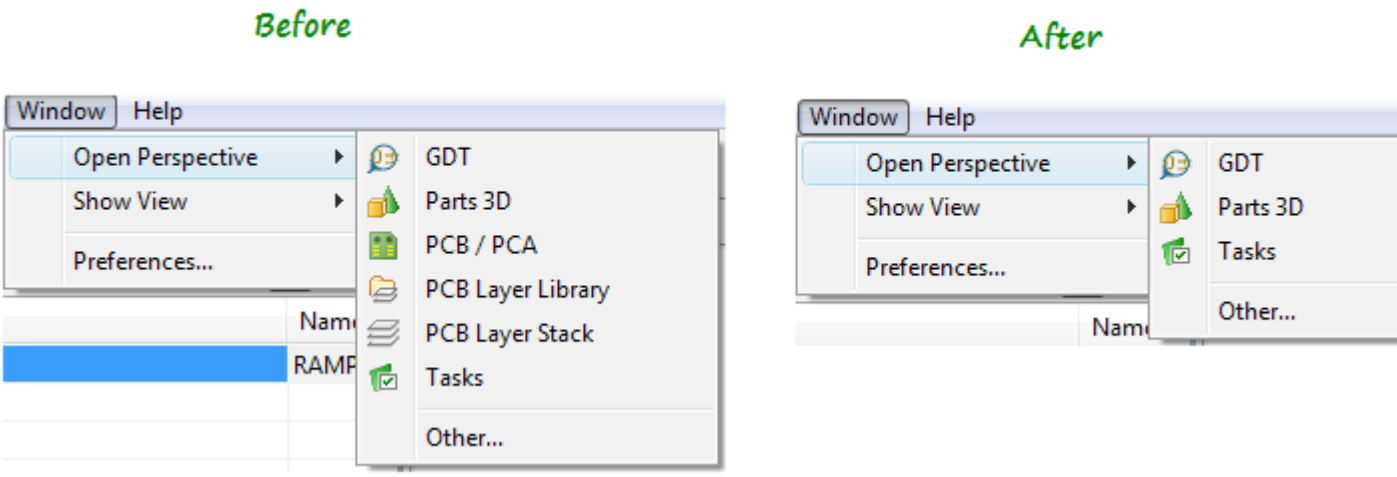

Figure 5. Open Perspective sub menu content before and after shortcuts were unselected

To top

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5.3.4. Reset Perspective

To restore a currently opened perspective to its original layout:

- Right click on the active perspective button in the perspective bar.
- Select **Reset** menu item in the appeared context sensitive menu (see Figure 1).

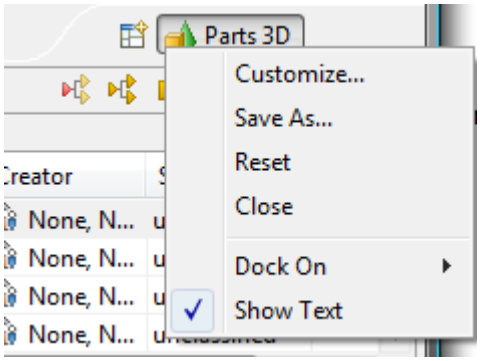

Figure 1. Context sensitive menu on the active perspective button

- The **Reset Perspective** dialog window appears asking whether you want to reset the current perspective to it's defaults or not (see Figure 2).

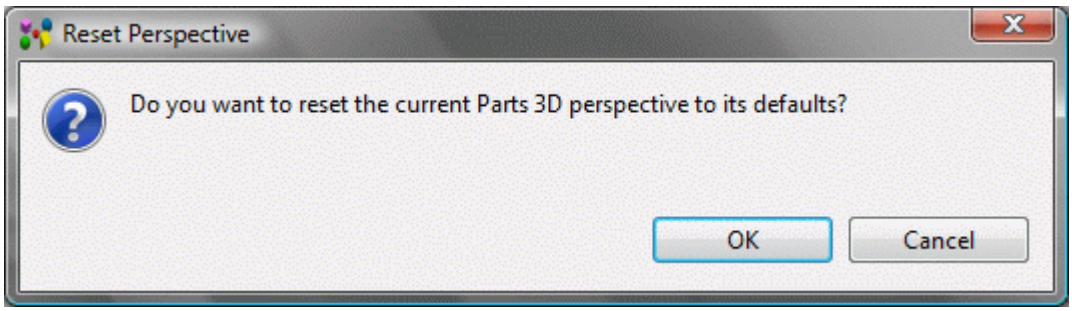

Figure 2. Reset perspective dialog window

- Press **OK** button to reset perspective or **Cancel** button to reject the resetting.

To top

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5.3.5. Switching Between Perspectives

Open perspectives are represented by icons on the perspective bar. When you have more than one perspective open, you can switch between them by clicking the icons on the shortcut bar.

To top

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5.4. Toolbars

There are three kinds of toolbars in **IDA-STEP**.

The *main toolbar*, sometimes called the workbench toolbar, is displayed at the top of the workbench window directly beneath the menu bar. The contents of this toolbar change based on the active perspective, installed features. Items in the toolbar might be enabled or disabled based on the state of either the active view or editor. Sections of the main toolbar can be rearranged using the mouse.

There are also individual *view toolbars*, which appear in the title bar of a view. Actions in a view's toolbar apply only to the view in which they appear. Some view toolbars include a **Menu** button, shown as an inverted triangle, that contain actions for that view.

A third type of toolbar is the perspective switcher (perspective bar). The perspective switcher allows quick access to perspectives that are currently open. It also has a button that can open new perspectives. The perspective switcher is normally located in the top-right, next to the main toolbar. However, it is also possible to position it below the main toolbar ("top-left"), or to position it vertically on the left-hand side of the workbench ("left"). The name of the perspectives is shown by default, but it is possible to hide the text and show only the icons. To reposition the perspective or hide the text, right-click on it and choose the appropriate item from the context menu.

In all cases, you can find out what toolbar buttons do by moving your mouse pointer over the button and reading the tooltip that opens.
You can customize toolbars for each perspective - turn tool bar items on and off (refer clause Customize Perspective).

The following commmon buttons may appear in the workbench toolbar, toolbars for views:

Button	Description
	Open a new perspective
	Hide view (Minimize)
	Restore view (Maximize)
	Open a help window
	Search in a help window
	View datasets of the selected view type
	Open a new window of the active view
	Go to top
	Go back
	Go in
	Link with views

IDA-STEP also has a number of toolbars, which group together associated functions (depends on the installed features).

Hide/Show toolbar on Mac OS

There's a button to hide/show the toolbar on Mac OS. In case the toolbar is hidden and the application closed, the toolbar and the button to show it disappears. The *Hide* toolbar action is available through toolbar context menu.

In case the toolbar is hidden, the the **Show toolbar** command becomes visible in the **Window** main menu.

To top

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5.4.1. File Buttons

File toolbar buttons:

Button	Description
	Open a new IDA-STEP window
	Open a new file
	Save the contents of an active file
	Close an active file

To top

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5.4.2. GD&T Buttons

GD&T tool bar buttons group functions for creating GD&T elements:

Button	Description
	Geometric Tolerance
	Angle
	Size
	Distance
	Annotation Plane
	Text Annotation

	Shape Aspect

To top

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5.4.3. Datum Buttons

The **Datum** tool bar buttons group functions for creating GD&T datum elements:

Button	Description
	Datum Feature
	Point Datum Target
	Line Datum Target
	Circle Datum Target
	Circular Curve Datum Target
	Rectangle Datum Target
	Datum System

To top

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5.4.4. Tool Buttons

Tool toolbar buttons:

Button	Description
	Create assembly from parts

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5.5. Statusbar

The *status bar* is a horizontal window at the bottom of the workbench window in which IDA-STEP displays various kinds of status information.
The status bar can contain the following information (see Figure 1):

- Dynamic selection message,
- Current point message,
  Note. For more information about selection in the **3D View**, see View Control -> Select/Highlight, Selection Modes.
- Distance message (how to measure distance, see Measuring),
- Unit message (how to set unit see Quantities and Units),
- Heap indicator (for more information about heap and memory stuff, see Virtual Machine Memory),
- Progress indicator.

Note. The content of the status bar depends on what features are currently installed and which view is active.

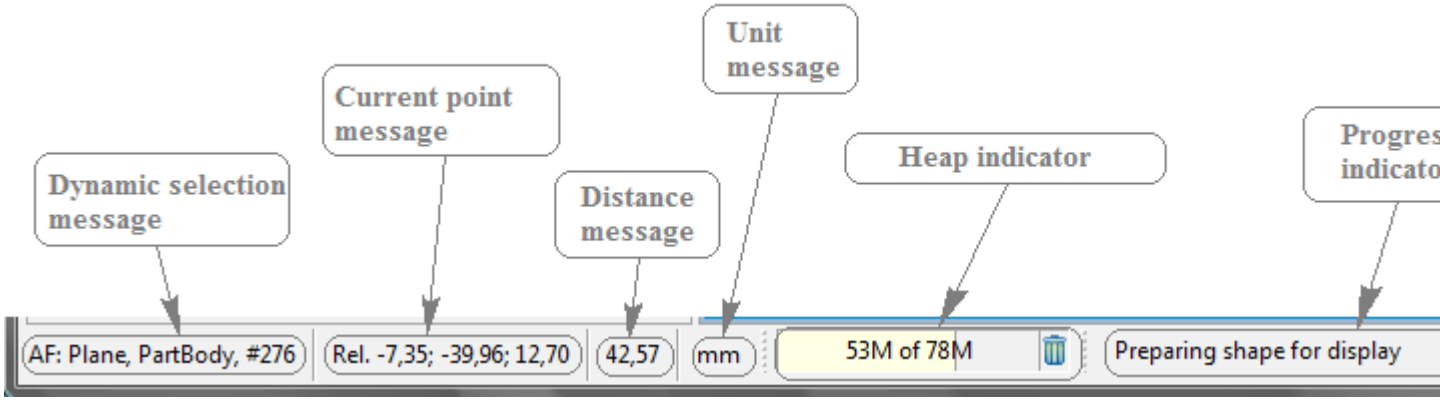

Figure 1. IDA-STEP workbench status bar

The *Heap indicator* and *Progress indicator* can be moved to other zones along the edges of the window. In addition a heap indicator can be closed.

To top

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5.6.1. File menu

This menu allows you to create, save files, import data and exit the **IDA-STEP** itself.

New

This command enables you to create a new empty STEP file (see New File).

Open

This command allows you to open files that do not reside in the workbench (see Open File).

Close

This command closes the currently opened file. If no file is opened, this menu item is disabled.

Save

This command allows you to save the contents of the active file (XIM). If a STEP file (p21) is opened, then this command allows you to save the contents as a XIM file.

Save As

This command allows you to save the contents of the active file under another file name or location. If a STEP file (p21) is opened, then this command allows you to save the contents as a XIM file.

Print

Prints the contents of the active view (see Print).

Print views...

Opens the selected view content in a preview window. This shows what the view content will look like when printed to a file (see Print Views).

Import

This option launches the import wizard, which allows you to add data to the **IDA-STEP** (see Import Wizard).

Export

This option launches the export wizard, which allows you to export data from the **IDA-STEP** (see Export Wizard).

Manage Libraries

Opens sub menu, which contains following items:

3D Part Libraries

This option launches wizard, which helps to maintain libraries of 3D parts.

E-CAD to M-CAD Mapping

This option launches wizard which helps to maintain package library.

Language

This command allows you to select a language for data.

Exit

This command closes and exits the **IDA-STEP**.

To top

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5.6.2. Edit menu

This menu helps you manipulate data in the **IDA-STEP** area.

Undo

This command reverses your most recent editing operation.

Redo

This command re-applies the editing action that has most recently been reversed by the Undo action.

Cut

This command removes the selection and places it on the clipboard

Copy

This command places a copy of the selection on the clipboard.

Paste

This command places the text on the clipboard at the current cursor location in the currently active view.

Delete

This command removes the selection.

Find...

This command opens the **Find** dialogue, which allows you to search for specific text within the view.

To top

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5.6.3. GD&T menu

This menu allows you to create new GD&T elements:

Datum Feature

This command launches the wizard, which helps to create a datum feature. **Note.** This menu item is disabled until some shape element is selected.

Point Datum Target

This command launches a wizard, which helps to create a *point* on a part that is used in establishing a datum.

Line Datum Target

This command launches a wizard, which helps to create a *line* on a part that is used in establishing a datum.

Circle Datum Target

This command launches a wizard, which helps to create a circular area on a part that is used in establishing a datum.

Circular Curve Datum Target

This command launches a wizard, which helps to create a circular curve on a part that is used in establishing a datum.

Rectangle Datum target

This command launches a wizard, which helps to create a rectangular area on a part that is used in establishing a datum.

Datum System

This command launches the wizard, which helps to create a datum system.

Geometric Tolerance

This command launches the wizard, which helps to create a geometry tolerance annotation.

Angle

This command launches a wizard, which helps to create a dimension that indicates the angle between two elements.

Size

This command launches a wizard, which helps to create a size (diameter, radius, length) dimensions for the selected element.

Distance

This command launches a wizard, which helps to create a distance dimension between two elements.

Annotation Plane

This command launches the wizard, which helps to create an annotation plane.

Text Annotation

This command launches the wizard, which helps to create a text annotation.

Shape Aspect

This command creates an element of the shape of the product. **Note.** This menu item is disabled, if no geometry element is selected.

To top

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5.6.4. Supplement menu

This menu allows you to create new geometric elements included in product definition data to communicate design requirements but not intended to represent a portion of the manufactured product:

Apex

This command allows you to create an apex for the selected shape aspect of type *cone*.

Symmetry Axis

This command allows you to create a line, which is a center line of the selected cylinder/cone/toroid/revolution.

Symmetry Plane

This command allows to create a supposed mirror for the selected planes.

Symmetry Point

This command allows to create a center *point* for the selected circle/sphere.

Common Line

This command line allows to create a *line*, which goes through the selected points

Common Plane

This command allows to create a *plane*, which goes through the selected two lines/line and the specified point.

Intersection

This command allows to create an intersection of two element:

- a Cartesian *point*, which is an intersection of two non parallel lines;
- a *line*, which is an intersection of two non parallel planes;
- a *circle/ellipse/line*, which is an intersection of a plane and a cylinder;
- a Cartesian *point*, which is an intersection of non parallel objects - line and plane;
- a Cartesian *point*, which is an intersection of non parallel objects - line and cylinder;
- a Cartesian *point*, which is an intersection of line and sphere;
- a *circle*/Cartesian *point*, which is an intersection of plane and sphere;

Offset

This command launches a wizard, which helps to create a *plane*/*line/ point*, whose shape parallels the shape of the original object.

Perpendicular Line

This command creates a *line,* which maintains a 90-degree angle to a particular object (plane, sphere, cylinder, cone) and goes through the specified point.

Perpendicular Plane

This command creates a *plane*, which maintains a 90-degree angle to a particular object and goes through

the specified point/line:

- a *plane*, which is perpendicular to the selected plane and goes through the specified line;
- a *plane*, which is perpendicular to the selected line and goes through the specified point;
- a *plane*, which is perpendicular to the center axis of the selected cylinder and goes through the specified point.

Extension

This command creates a *plane*, which is an extension of the selected plane.

Tangent Plane

This command creates a *plane* that touches the selected cylinder/cone/sphere at the specified point.

Point

This command launches a wizard, which helps to create a *point*.

Line

This command launches a wizard, which helps to create a *line*.

Plane

This command launches a wizard, which helps to create a *plane*.

To top

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5.6.5. Window menu

This menu allows you to display, hide, and otherwise manipulate the various views, perspectives, and actions in the workbench.

Open Perspective

This command opens a new perspective in this workbench window. All of the perspectives that are open within the workbench window are shown on the perspective bar. The perspectives you will likely want to open are listed first. This list is dependent on the current perspective. From the **Other...** sub menu you can open any perspective. For more information see Open Perspective.

Show View

This command displays the selected view in the current perspective. Views you are likely to want to open

are listed first. This list is dependent on the current perspective. From the **Other...** sub menu you can open any view. The views are sorted into categories in the Show View dialogue.For more information see Opening View.

Preferences

This command allows you to indicate your preferences for using the Workbench. For more information see Preferences.

To top

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5.6.6. Help menu

This menu provides help on using **IDA-STEP**.

Help Contents

This command displays the help contents in a help window: help book, topics, and information related to the **IDA-STEP**.

Search

This command displays the help view opened on the **Search** page.

Dynamic Help

This command displays the help view opened to **Related Topics** page.

Register, Install and Update

This command displays the **Register, Install and Update** page (see Register, Update and Install).

About IDA-STEP

This command displays information about the product, installed features, and available plug-ins (see About).

To top

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6.1. New File

Follow the routine to create a new STEP file:

- Select **File ->** **New** menu item in the main menu bar.

Or

- Press **New** button in the main tool bar.

A new empty document is opened in IDA-STEP with a default name *Untitled* in the frame header. The recently used perspective is the default perspective for a new document.

To top

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6.2. Opening File

After IDA-STEP is started, an empty STEP file is opened in the workbench with a default name *Untitled* in the frame header.
Follow the routine to open an applicable file (XIM, STP, STEP, P21) in the workbench:

- Select in the main menu **File ->** **Open** menu item or click on **Open** button in the toolbar.
- The **Open** window appears on the screen:
  - Select an applicable XIM (*.xim) or STEP (*.stp, *.step, *p21) file in your file system.
  - Click **Open** button.
- The selected file is opened in the workbench.

The name of opened file is displayed in the frame header (see Figure 1).

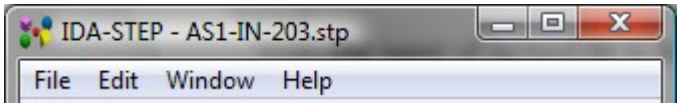

Figure 1. File name in the frame header

To top

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6.3.1. Import Wizard

The **Import** wizard helps to import data to IDA STEP (see Figure 1). To start the **Import** wizard select in **IDA-STEP** main menu bar **File -> Import**.

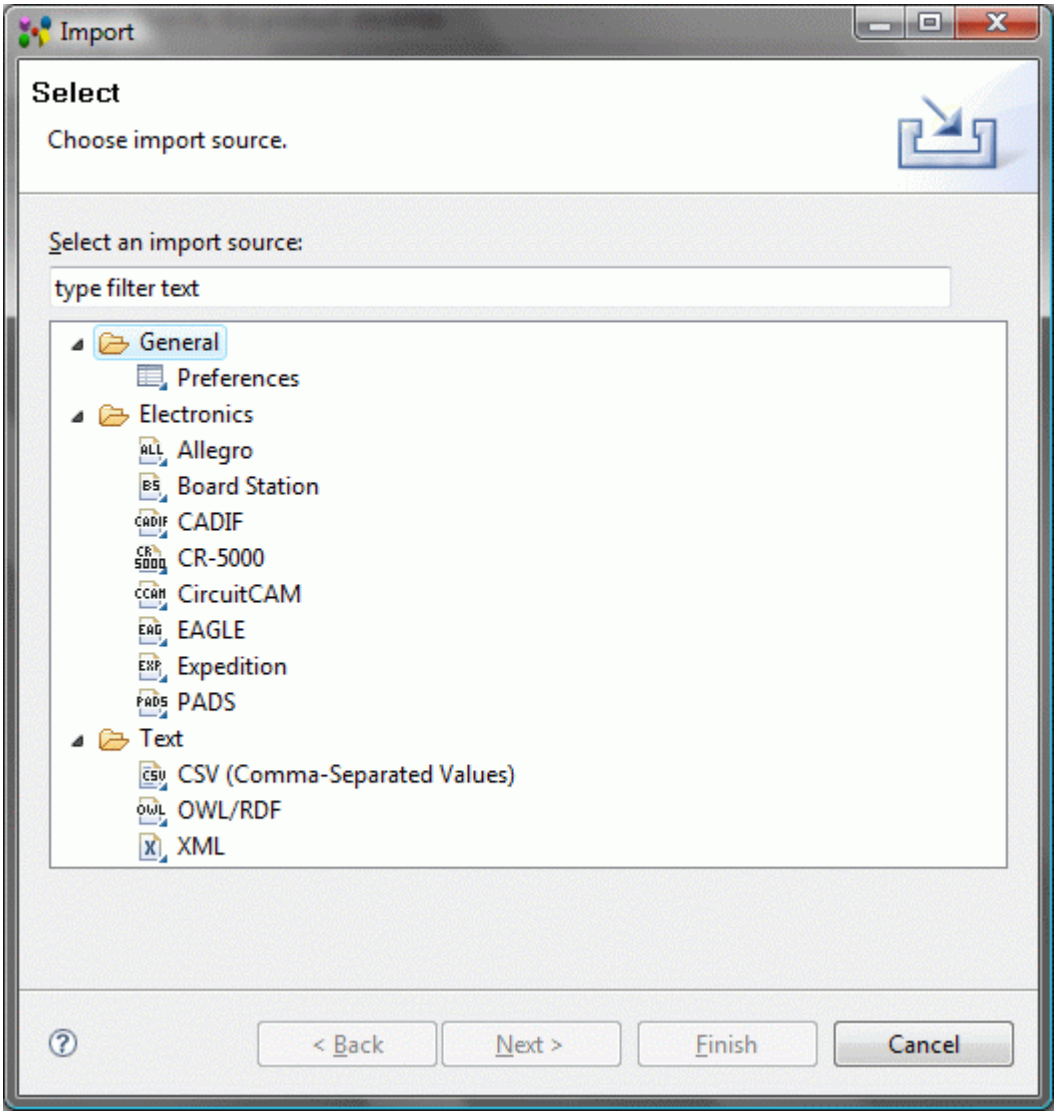

Figure 1. Import wizard page

When the **Import** wizard page appears on the screen you must select what type of import to do. To assist in locating a particular wizard, the text field can be used to show only the wizards that match the entered text.
After an import type is selected, click **Next** button to launch a particular wizard page.

Note. The available import capabilities depend on the kind of installed software components.

To top

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6.3.2. Importing and Exporting Preferences

Preference files (.epf) capture all the internal settings of IDA-STEP including the settings in the Preferences dialogue such as HTTP proxy server settings and GUI customizations. With Export and Import Preferences you can copy these settings from one IDA-STEP installation to another to ensure that the same settings are used there.

See also **Overview /** Preferences.

To top

Importing Preferences

With **Import / Preferences** all or some of the settings in a Preference File (*.epf) are loaded into IDA-STEP, replacing the settings that where there before.

See also Import Wizard.

Follow the routine to import preferences to IDA-STEP:

- Select in the main menu **File -> Import** menu item. The **Import** wizard page appears on the screen.
- Select **General -> Preferences** item (see Figure 1) and click **Next** button.

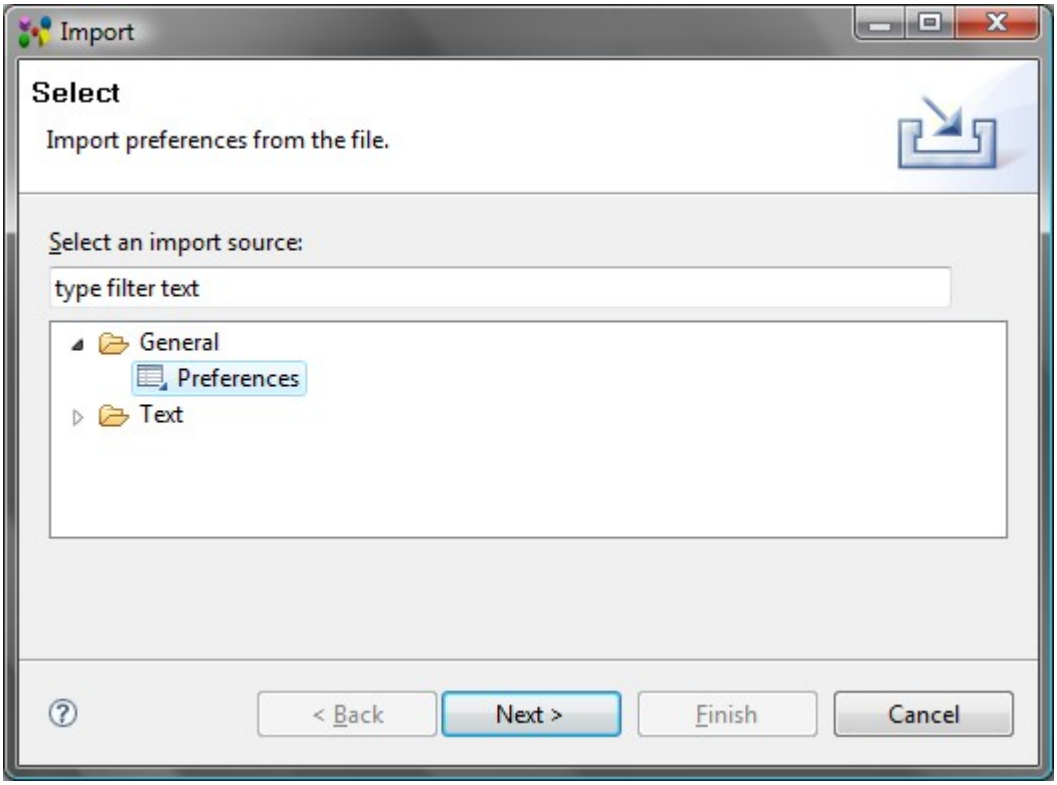

Figure 1. Import wizard page, selected General -> Preferences item

- The **Import Preferences** wizard page appears on the screen (see Figure 2):

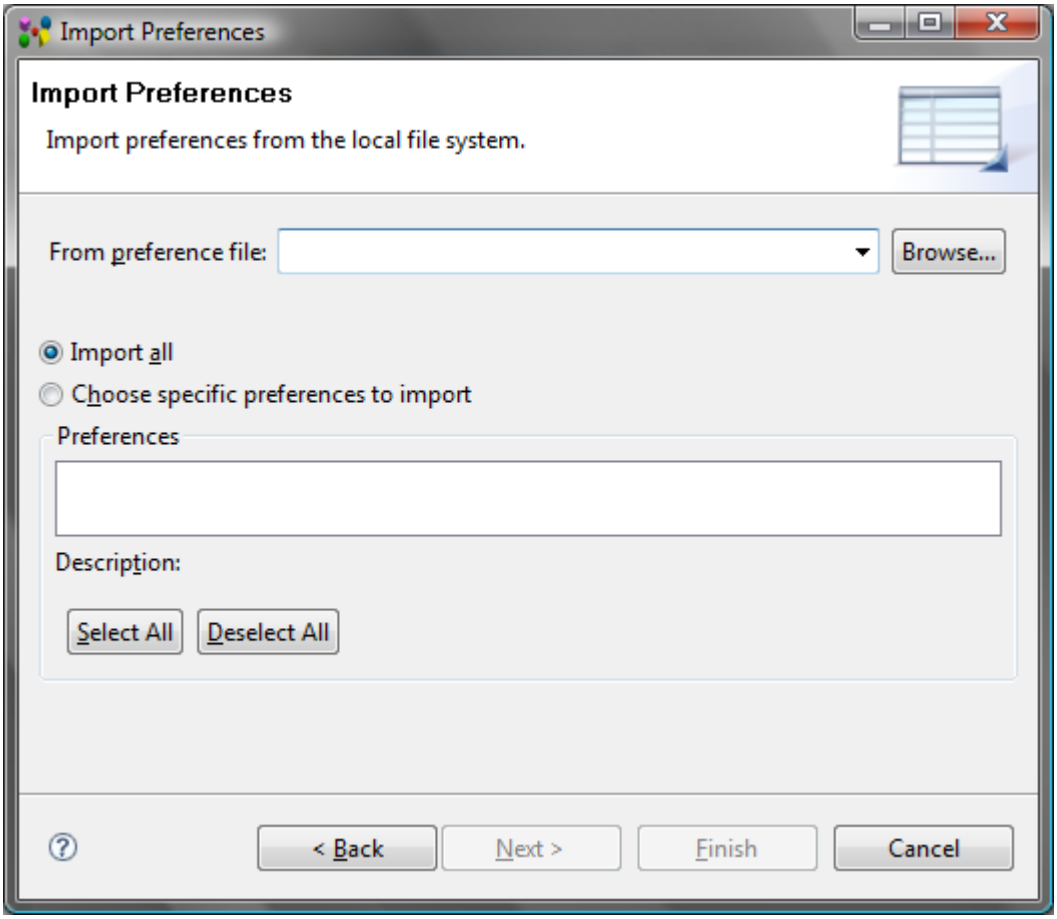

Figure 2. Import Preferences wizard page

- Click **Browse...** button next to the **From preference file** field and locate the preferences file on the local file system.
- Select one of the following radio buttons:
  - **Import all** - to accept all of the preferences defined in the file.
  - **Choose specific preferences to import** - to select only specified preferences defined in the file.
- You can read the description of the selected preference in the **Description** field.
- Also you can use **Select All** or **Deselect All** buttons to select or unselected preferences.
- Click **Finish** button.

The preferences from the specified file are imported into IDA-STEP.

To top

Exporting Preferences

With **Export / Preferences** some or all of the internal settings of IDA-STEP are stored in an "*.epf" file.

See also Export Wizard.

Follow the routine to export preferences:

- Select in the main menu **File -> Export** menu item. The **Export** wizard page appears on the screen.
- Select **Preferences** item (see Figure 3) and click **Next** button.

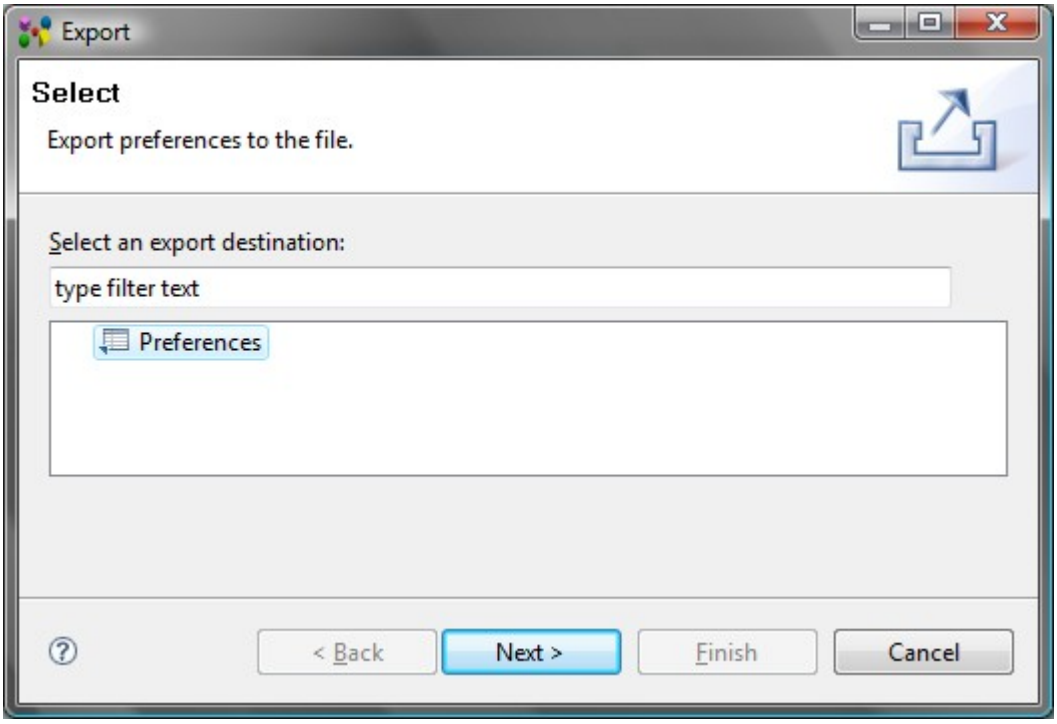

Figure 3. Export wizard, selected Preferences item

- The **Export Preferences** wizard page appears on the screen (see Figure 4):

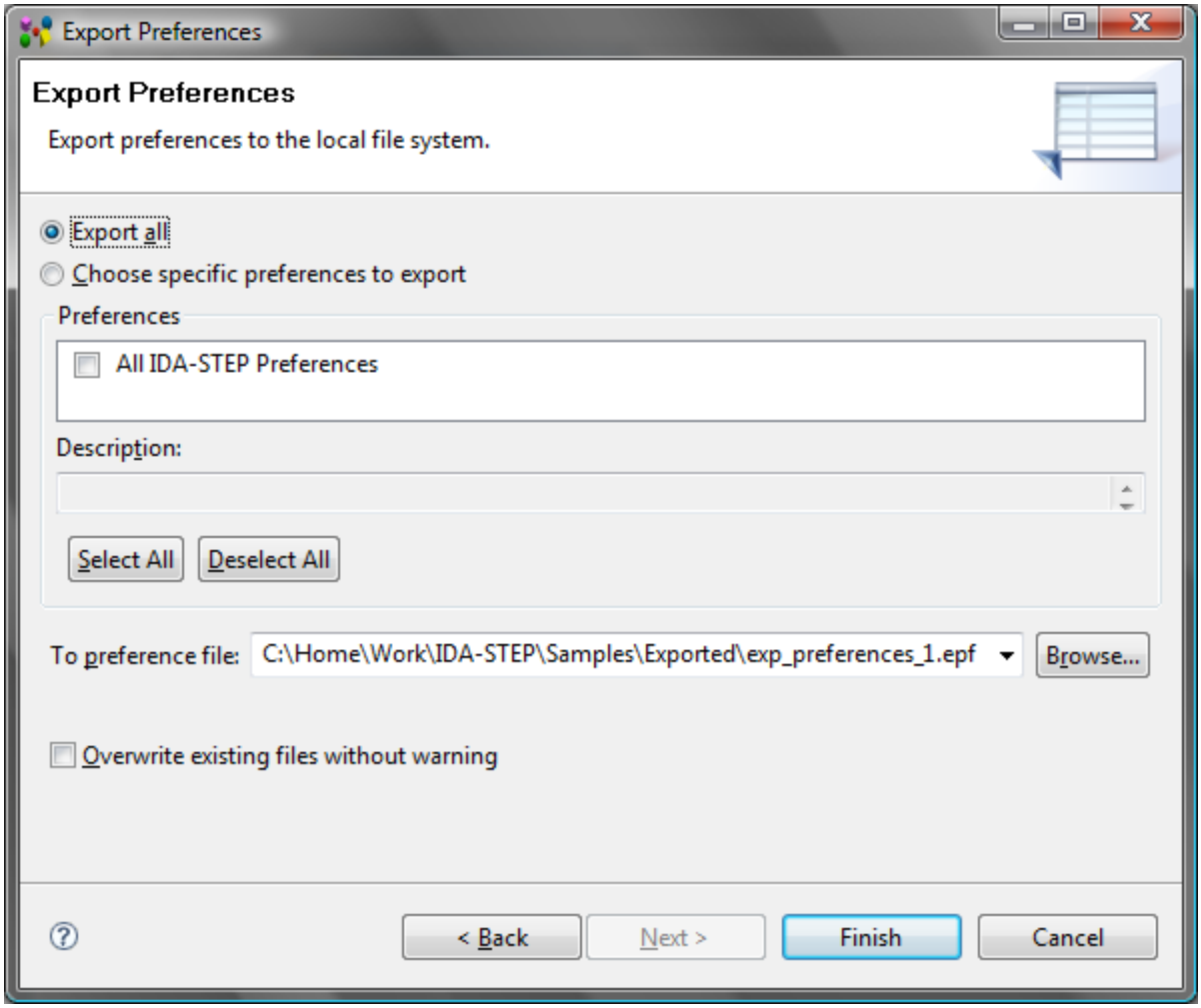

Figure 4. Export Preferences wizard page

- Select one of the following radio buttons:
  - **Export all** - to add all of the preferences to the file.
  - **Choose specific preferences to import** - to add only specified preferences to the

file.

- You can read the description of the selected preference in the **Description** field.
- Also you can use **Select All** or **Deselect All** buttons to select or unselected preferences.
- Click **Browse...** button next to the **To preference file** field and locate the preferences file on the local file system.
- You can select **Overwrite existing files without warning** to get no warning message that the specified preferences file already exists in your file system.
- Click **Finish** button.

The selected preferences are exported to the specified file.

To top

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6.3.3. Importing EAGLE

EAGLE Layout Editor is a tool from CadSoft for designing Printed Circuit Boards (PCB).
The **Import EAGLE** feature allows to convert EAGLE board files (*.brd) into IDA-STEP. This conversion process has two steps. The first step is to run a special ULP program inside EAGLE to dump out the binary board file into an ASCII file and also to generate a design rule file (*.dru). In the second step these files are then imported into IDA-STEP.
See Import Eagle for details on supported features.

First step: Data preparation in EAGLE

Start **EAGLE Layout Editor** and load a board file (*.brd) to generate these files:

- EAGLE data file (*.txt);
- EAGLE design rules file (*.dru).

First of all you need to generate a special **.txt* file. For this you need to execute a program, provided with IDA-STEP in the so called User Program Language (ULP) format. Follow the routine:

1. In the main tool bar press **Run** button.
2. Select one of the **.ulp* files, coming with IDA-STEP, in the appeared **Run** dialog. This file is located in IDA-STEP installation folder (*.../plugins/net.jsdai.xim.ap210.eagle/config*).
   - use *eagle_to_ASCII_40.ulp* - for Eagle 4.0 or lower version;
   - use *eagle_to_ASCII_41.ulp* - for Eagle 4.1 and higher version;
3. In the opened dialog select the output file and press **OK** button. As a result, a *.txt file is created.
   Note: Do not edit the exported file!

Next a design rule (**.dru*) file has to be generated. Follow the routine:

1. Select in the main menu bar **Edit -> Design Rules** menu item to open the **Design Rules** dialog window.
   1. Ensure the **File** tab is currently activated.
   2. Press **Save as** button and choose the location in your file system the **.dru* file to be saved in.

For more details on ULP execution and design rules refer Eagle Help.

Second step: Import *.txt and *.dru files into IDA-STEP

Follow the routine to go through the **Import EAGLE** wizard steps:

- Select in the main menu **File -> Import**.
- Select **Electronics ->** **EAGLE** in the appeared **Import** wizard page and press **Next** button. The **Import EAGLE** wizard page appears on the screen (see Figure 1):

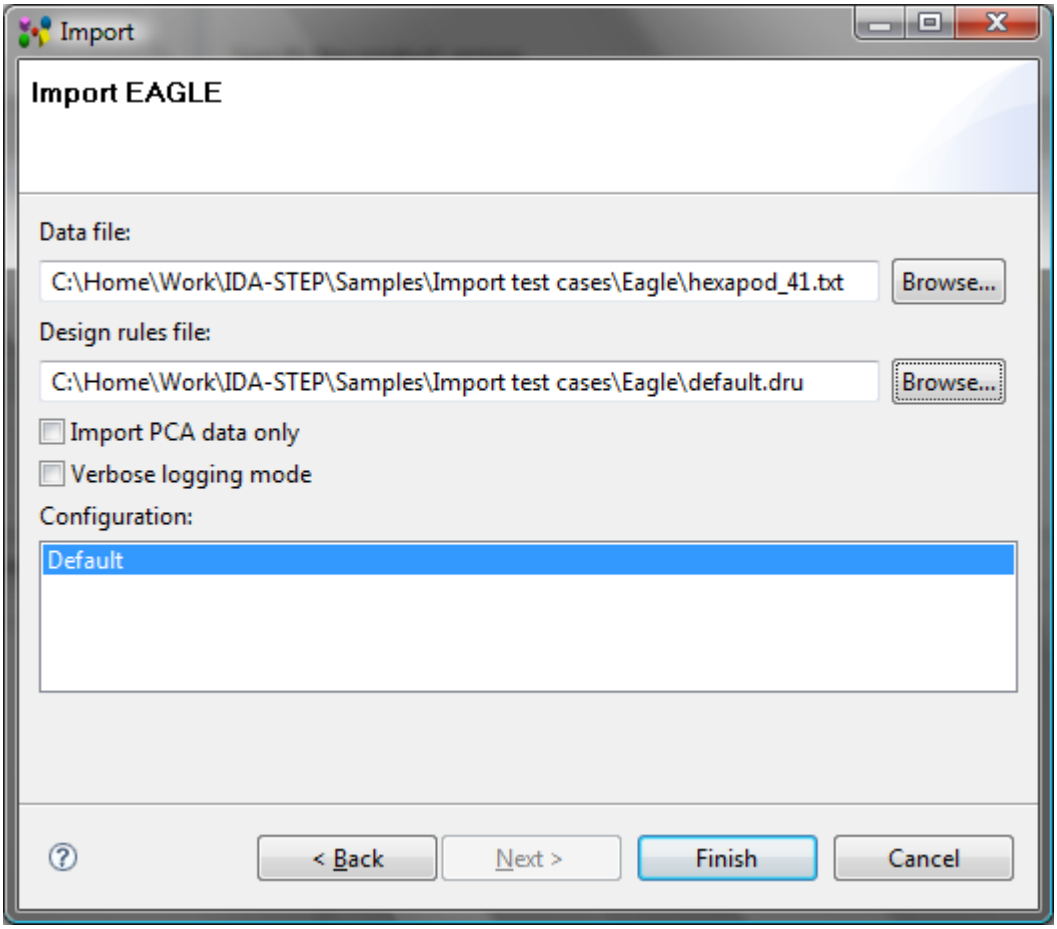

Figure 1. Import EAGLE wizard

- Select EAGLE data file (*.txt) by using **Browse** button next to the **Data file** field.
- Select EAGLE data rules file (*.dru) by using **Browse** button next to the **Design rules file** field.
- Choose whether the complete design PCA and PCB is imported, or import PCA data only by selecting/unselecting **Import PCA data only** option.
- Select the **Verbose logging mode** check box to get more detailed information in the *LOG* file.
- Details on how EAGLE data is converted into STEP are defined in special configuration files, which are provided by LKSoft. Available configurations are listed in the **Configuration** list. Typically the *Default* configuration is used.
- Press **Finish** button to import selected data.

After the import is completed, the generated data is opened in IDA-STEP (see Figure 2). Ensure that the **PCB/PCA** perspective is selected.

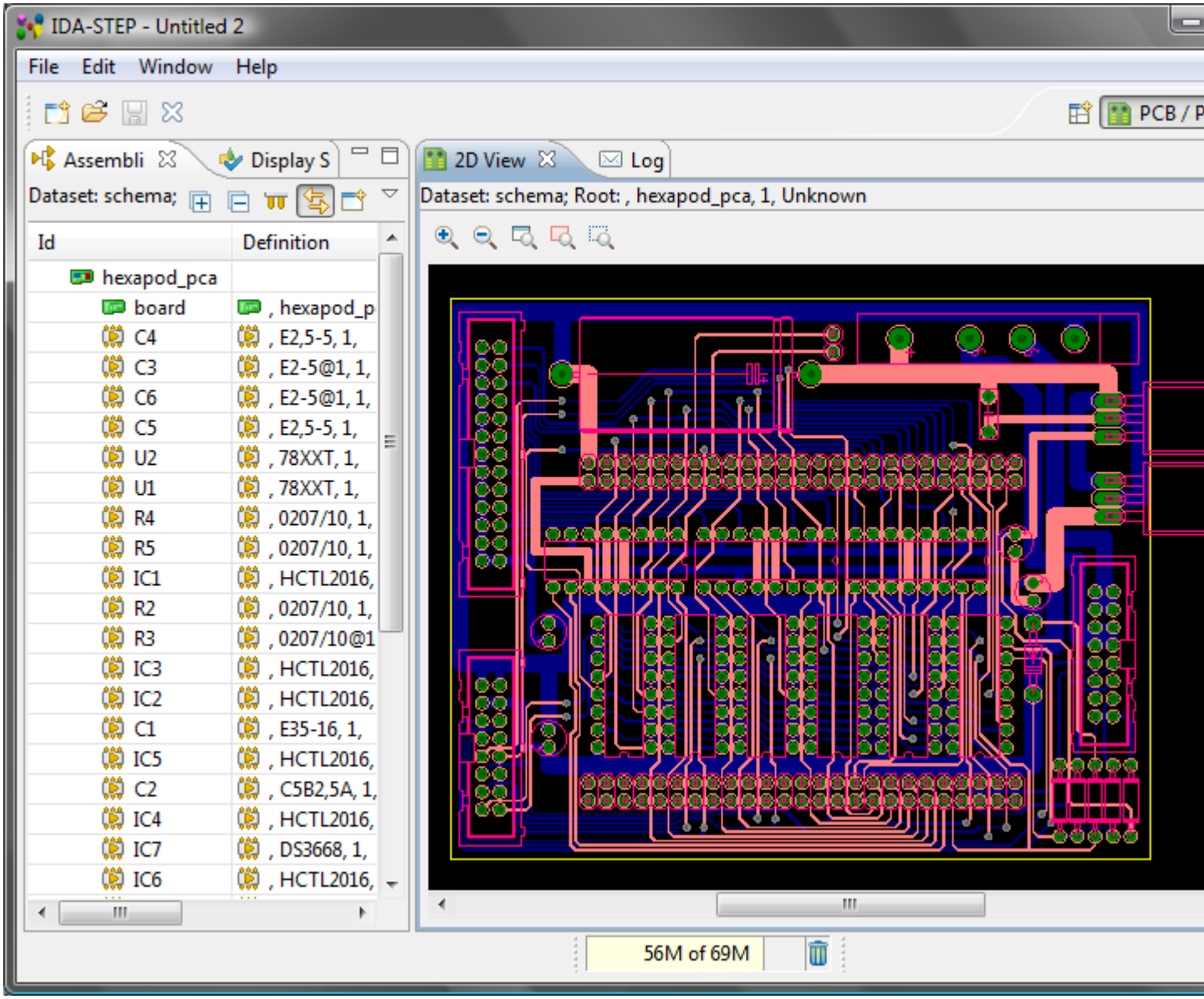

Figure 2. Imported EAGLE data (PCA and PCB) in IDA-STEP

To top

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6.3.4. Importing Data from CSV File

The CSV importer provides a powerful and flexible way to import data from a comma-separated file, which is a format supported by most applications (e.g. Microsoft Excel).
The **Import CSV File** wizard helps to import data from *CSV* file to IDA-STEP.
Follow the routine to go through the **Import CSV** wizard steps:

- Select in the main menu **File -> Import**.
- Select **Text -> CSV (Comma-Separated Values)** item in the appeared **Import** wizard page.
- The first **Import CSV** wizard page appears to select a *CSV* file (see Figure 1):

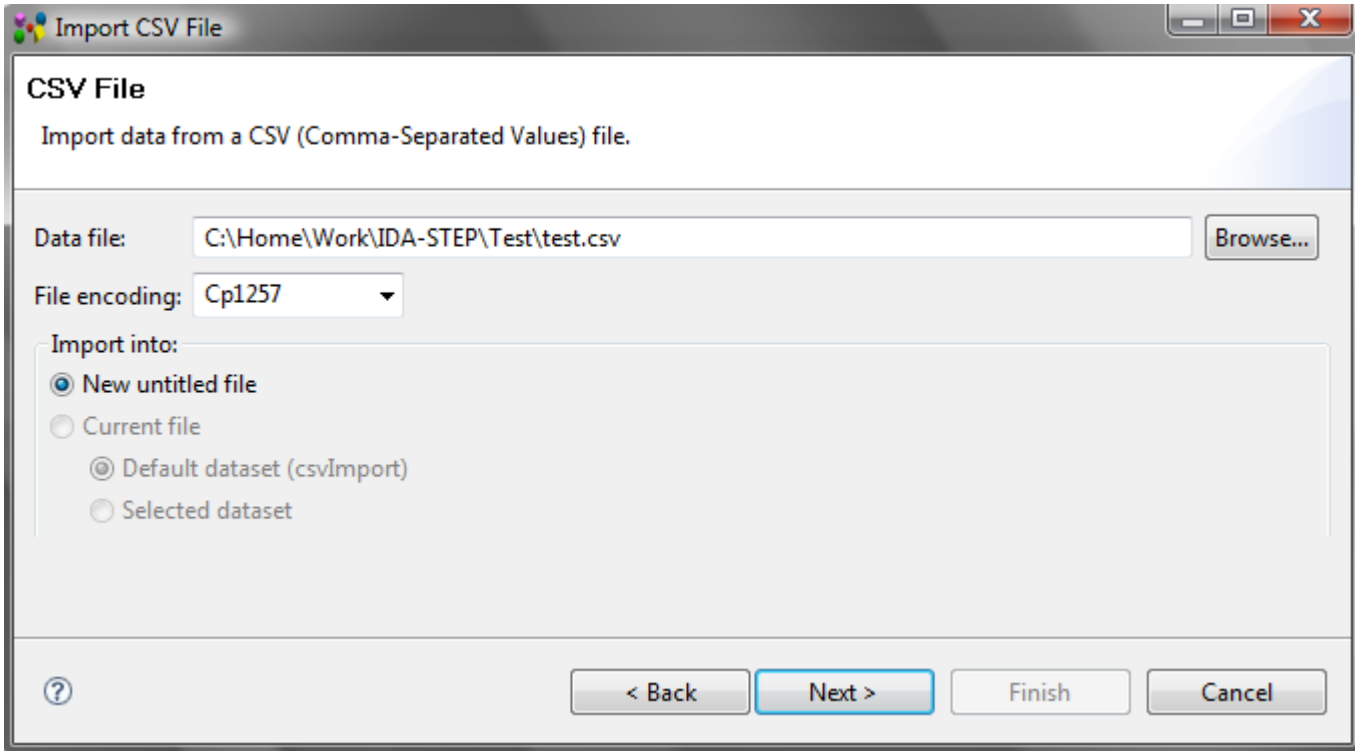

Figure 1. Import CSV file wizard page to select file to be imported

- Use **Browse** button next to the **Data file** filed to look for the CSV file in your file system.
- Select an applicable file encoding value in the **File encoding** drop down list.
- Select an applicable option in the **Import into** layout to define where data from CSV file should be imported:
    - **New untitled file** - import data into a new untitled file. This option is always enabled.
    - **Current file** - import data into the currently opened file (if no file is currently opened, this option is disabled). You have to select the dataset:
        - **Default dataset** - import data into the new dataset, named 'csvImport'.
        - **Selected dataset** - import data into the currently selected dataset (if no dataset is selected, this option is disabled).
- Press **Next** button to follow to the next wizard page (see Figure 2).

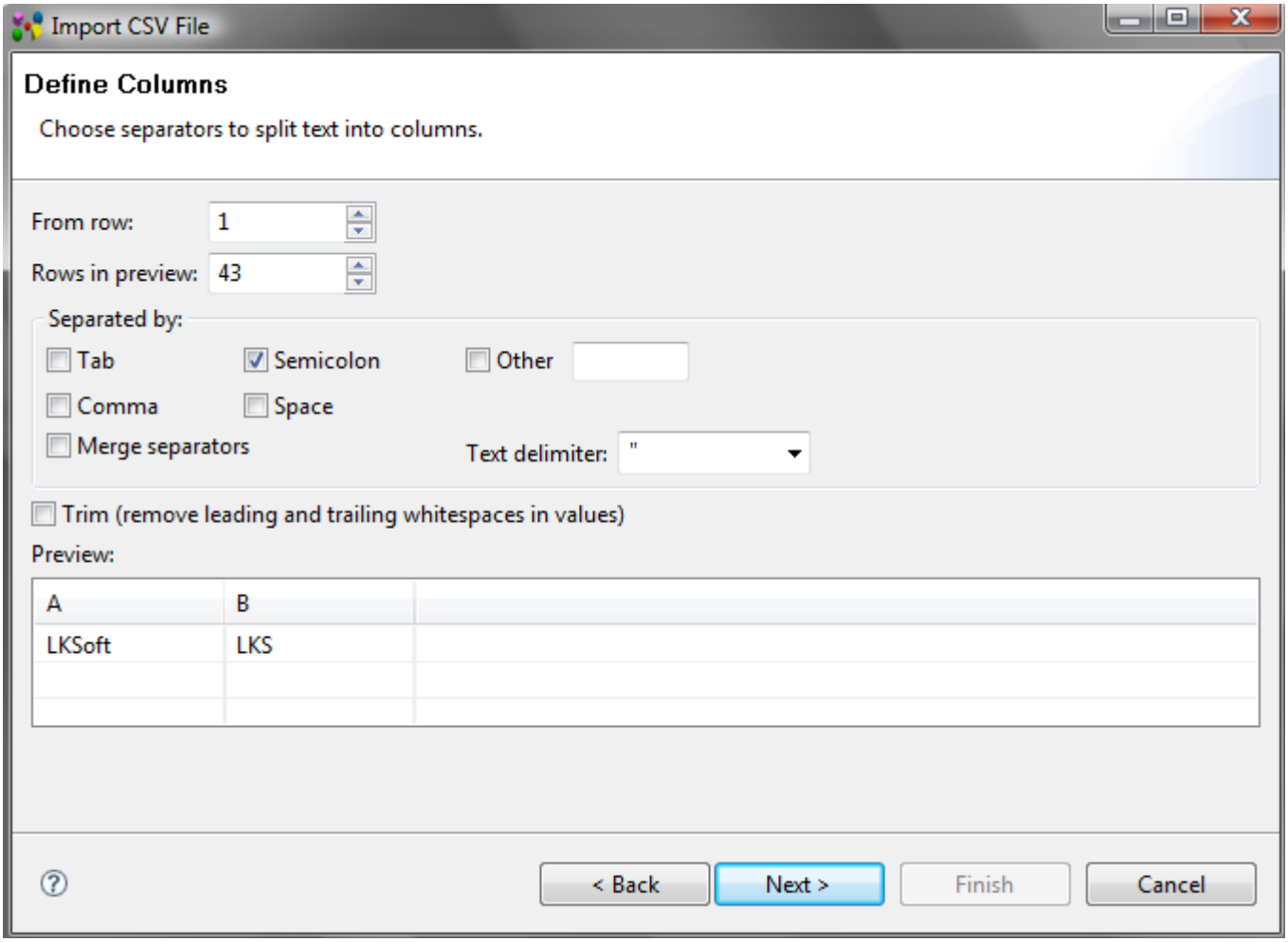

Figure 2. Import CSV file wizard page to define columns

- The **Define Columns** page appears on the screen to choose separators to split the text into columns:
  - Select the starting row number in the **From row** field by using arrows.
  - Select a number of rows available in preview in the **Rows in preview** filed by using arrows.
  - Select separator characters in the **Separated by** layout:
    - **Tab**,
    - **Comma**,
    - **Semicolon**,
    - **Space**,
    - **Other** - enter an applicable character to define it as a separator for the import file,
    - **Merge separators** - select this option to merge several separators,
    - **Text delimiter** - you can change the text delimiter value by selecting an applicable value from the drop down list.
  - Select **Trim** check box to remove leading and trailing whitespace in values.
  - The **Preview** table presents the content of the import file.
  - Press **Next** button to follow to the next wizard page (see Figure 3).

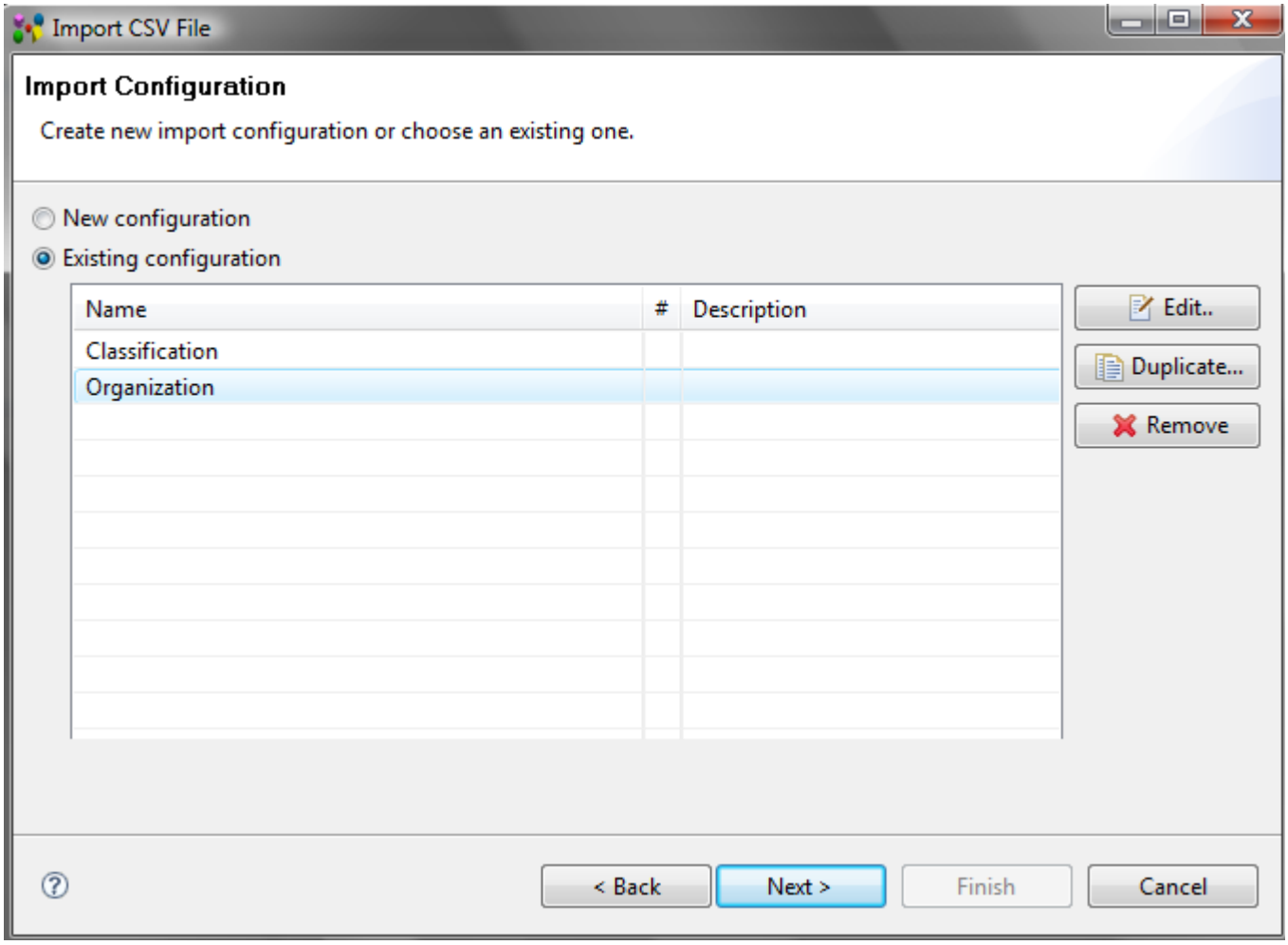

Figure 3. Import CSV file wizard page to define import configuration

- The **Import Configuration** page appears on the screen to create a new import configuration or choose an existing one:
    - Select one of:
        - **New configuration** - to create a new configuration for the import.
        - **Existing configuration** - to choose already existing configuration. After this radio button is selected. The list of available configurations appear in the table.
          Select an applicable configuration. You can edit (name and description), duplicate or remove the selected configuration bu using the particular buttons next to the table.
    - Press **Next** button to go to the next wizard page (see Figure 4).

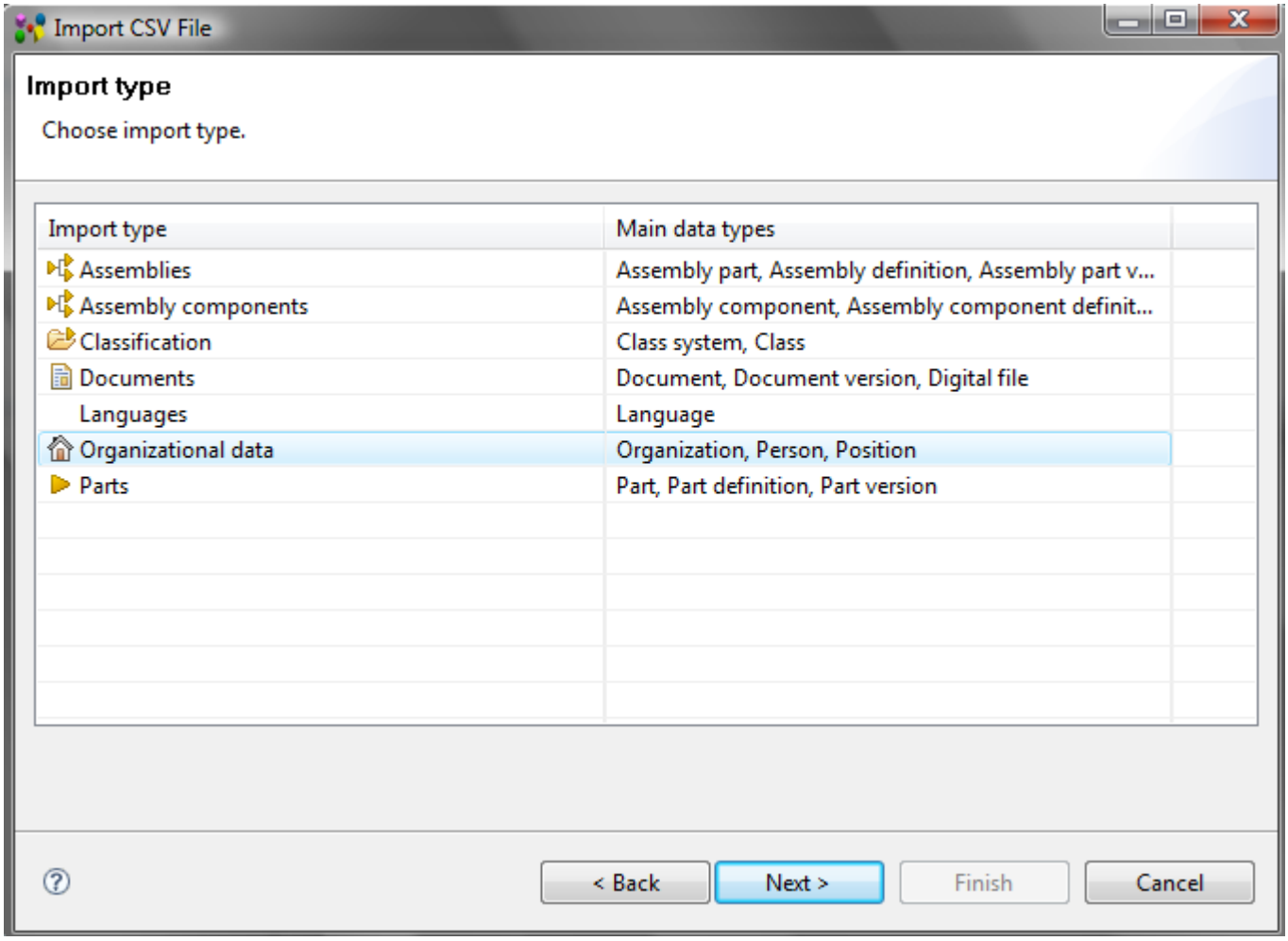

Figure 4. Import CSV file wizard page to define import type

- The **Import type** page appears on the screen to choose import type. The page present a table, which contain a list of available import types with specified data types:
    - Select an applicable import type in the table.
    - Press **Next** button to follow to the next wizard page (see Figure 5).

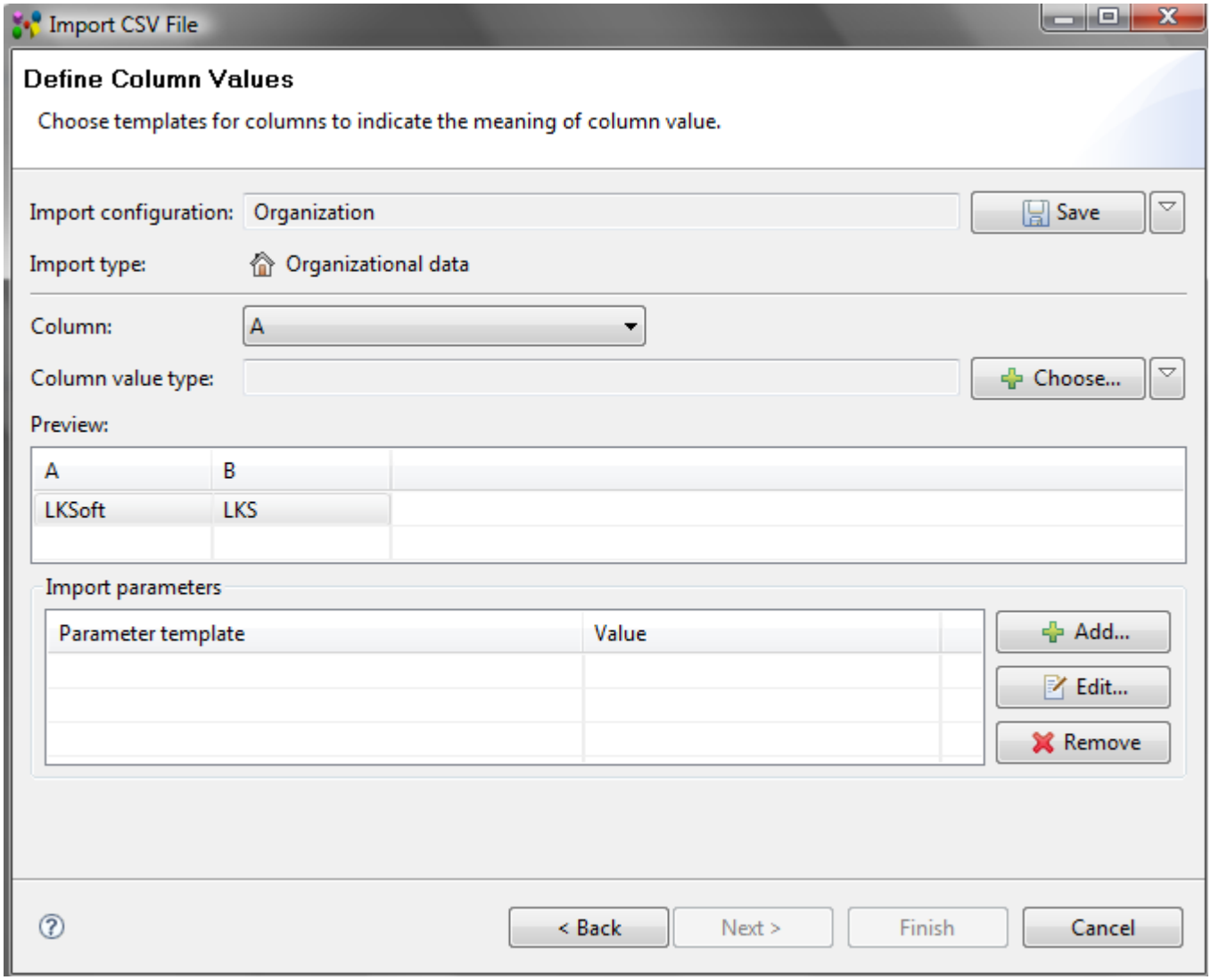

Figure 5. Import CSV file wizard page to define column values

- The **Define Column Values** page appears on the screen to choose templates for columns to indicate the meaning of a column value:
  - **Import configuration** field contains a name of selected configuration. If a new configuration was selected in the Import configuration page, you can save this configuration by clicking **Save** button. Specify a name and description of the configuration in the appeared wizard page. If you've edited the existing configuration, you can save it with a different name by clicking on the arrow next to the **Save** button and selecting **Save as** item.
  - **Import type** field presents the selected import type.
  - Select a column in the **Column** drop down list.
  - Press **Choose** button next to the **Column value type** field to define the column template. The **Choose template** wizard appears on the screen (see Figure 6):

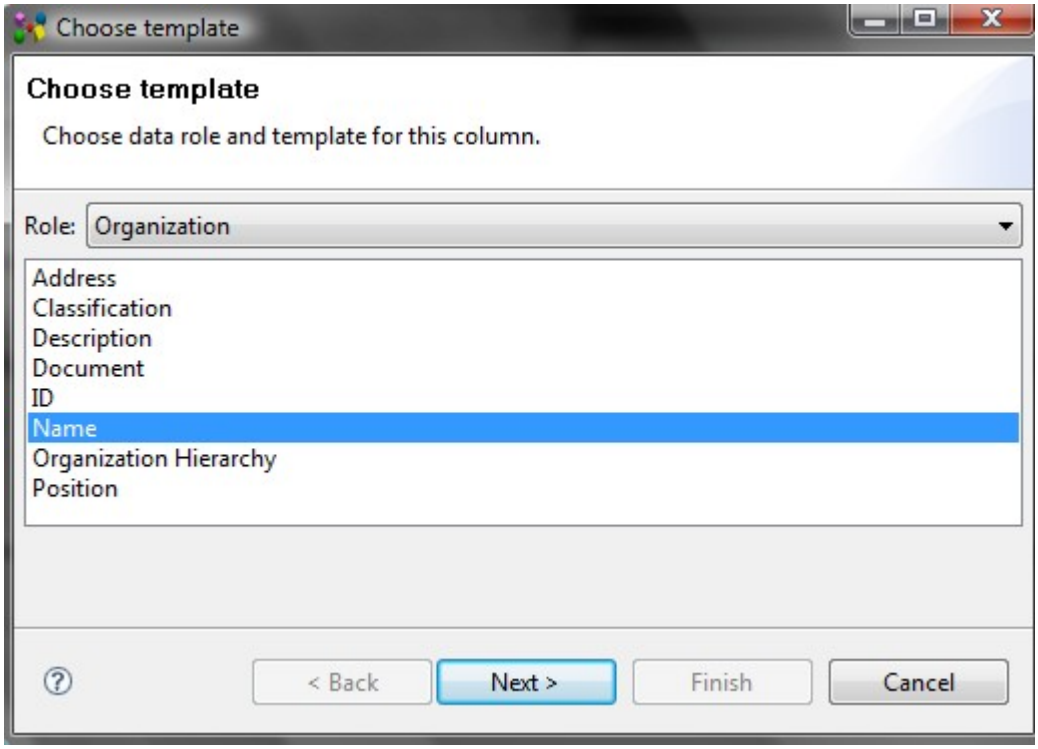

Figure 6. Choose template wizard page

- Select a data role in the **Role** drop down list. The listed values depends on the import type, selected in the previous steps.
- Select a template value from the list below the **Role** field.
- Press **Next** button to follow to the next **Choose template** wizard page (see Figure 7).

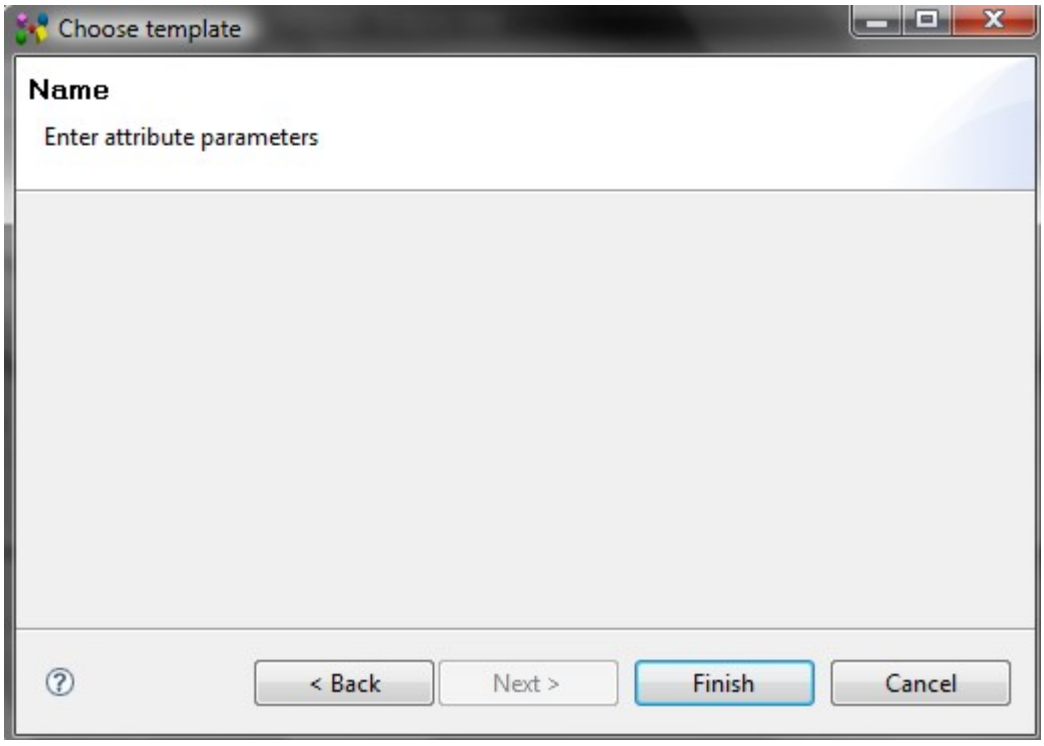

Figure 7. Column template parameters

- Enter needed parameters for the selected column template. In our example the template has no additional parameters, so the page is empty.
- Press **Finish** button. The selected template is assigned to the particular column (see Figure 8). You can review results in the **Preview** layout (**Define Column Values** page):
    - **Column header** - column index in CSV file, Data role and template.

- **Column value** - value from the CSV file.

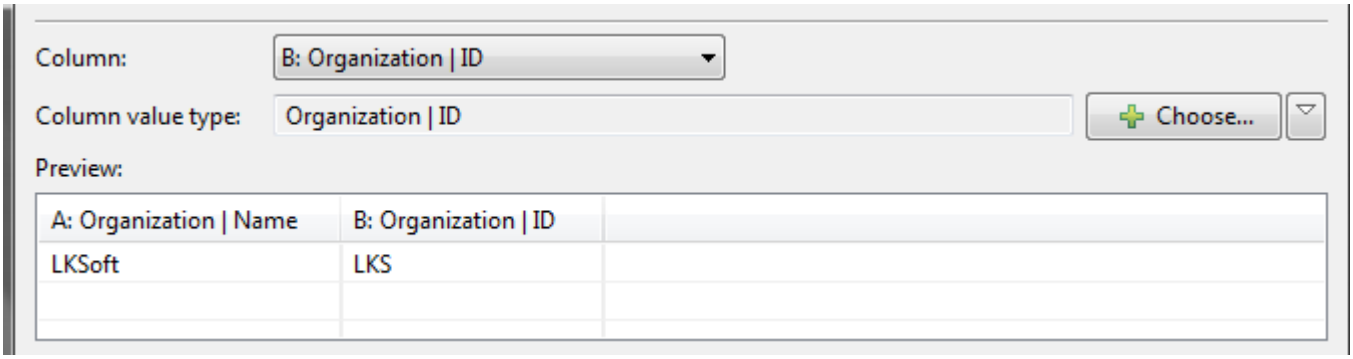

Figure 8. Assigned templates to the columns

- **Import Parameters Template** in the **Define Column Values** page (see Figure 5) presents a list of additional import parameters:
  - Press **Add** button add new parameters and assign the constant value.The **Add parameter** wizard page appears (see Figure 9):

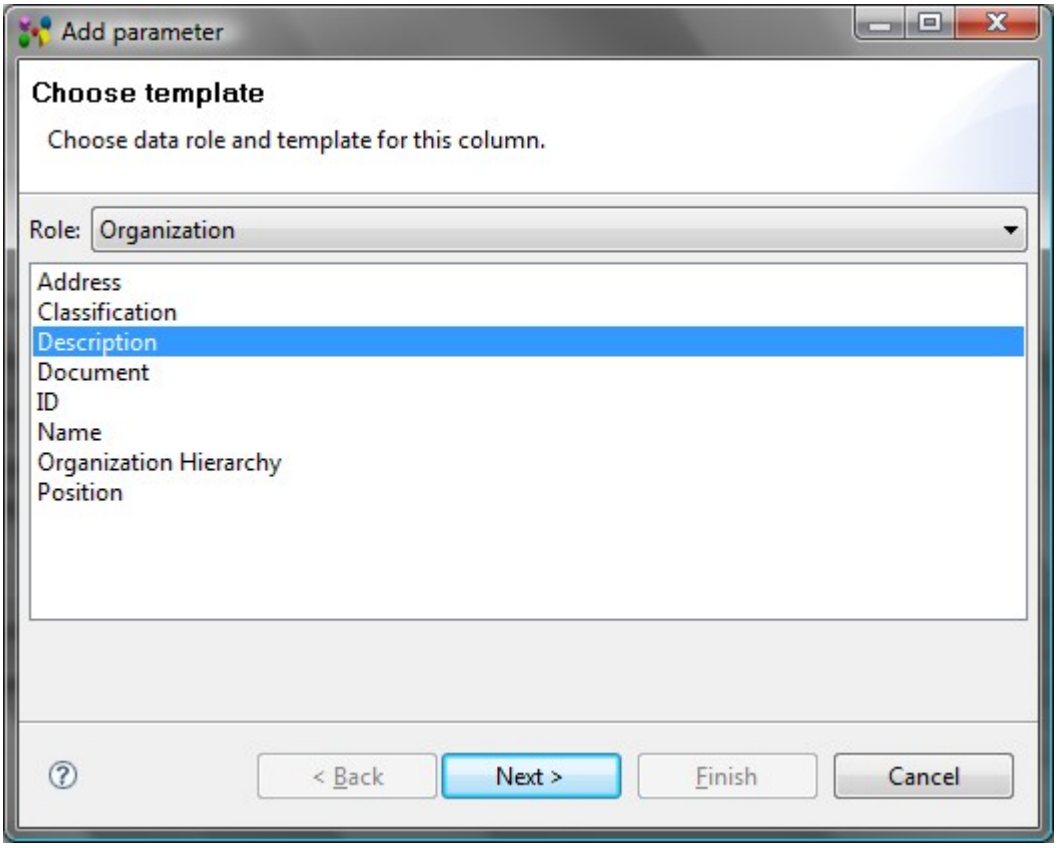

Figure 9. Add parameter wizard page

- Select a role of the parameter template in the **Role** drop down list.
- Select an applicable attribute from the list below the **Role** field.
- Press **Next** button. A new wizard page appears to enter parameter value (see Figure 10). If another template is selected, an additional page may appear to defined needed parameters.

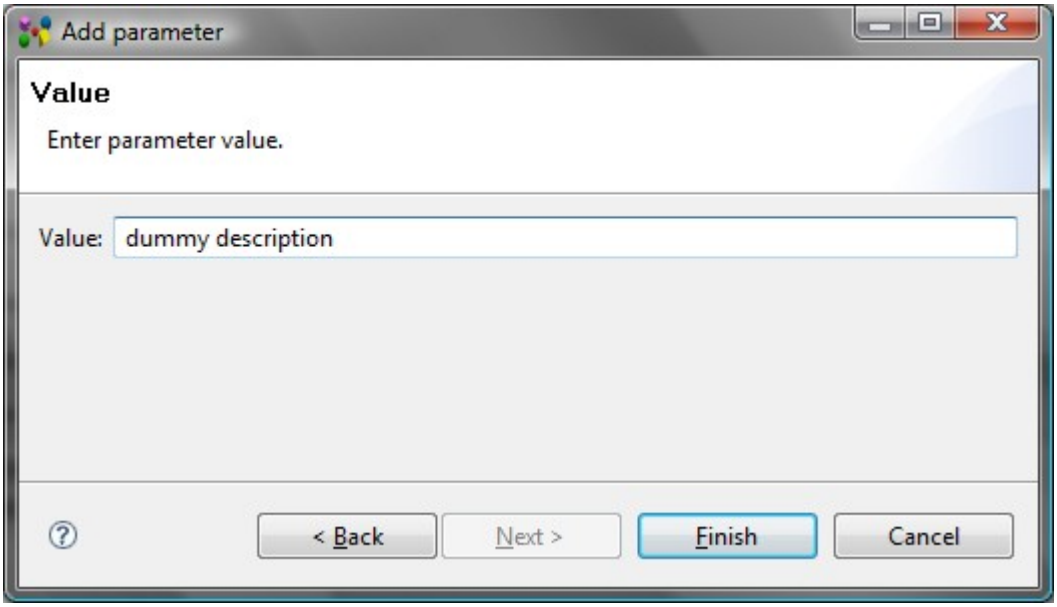

Figure 10. Add parameter wizard page to enter parameter value

- Enter an applicable parameter value into the **Value** field.
- Press **Finish** button. The defined parameter with entered value appears in the **Import parameters** layout table (see Figure 11).

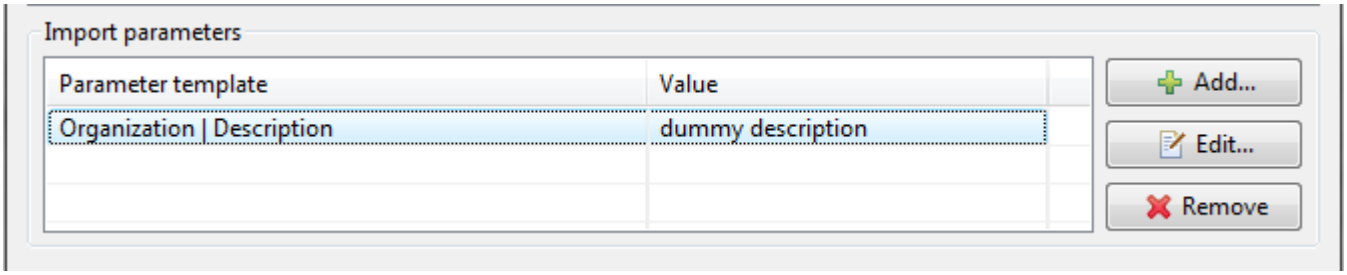

Figure 11. Define Column Values wizard page, Import parameters layout

- Press **Next** button to follow to the next **Import CSV File** wizard page (see Figure 12). The **Measure Units** page appears on the screen to define measure units.

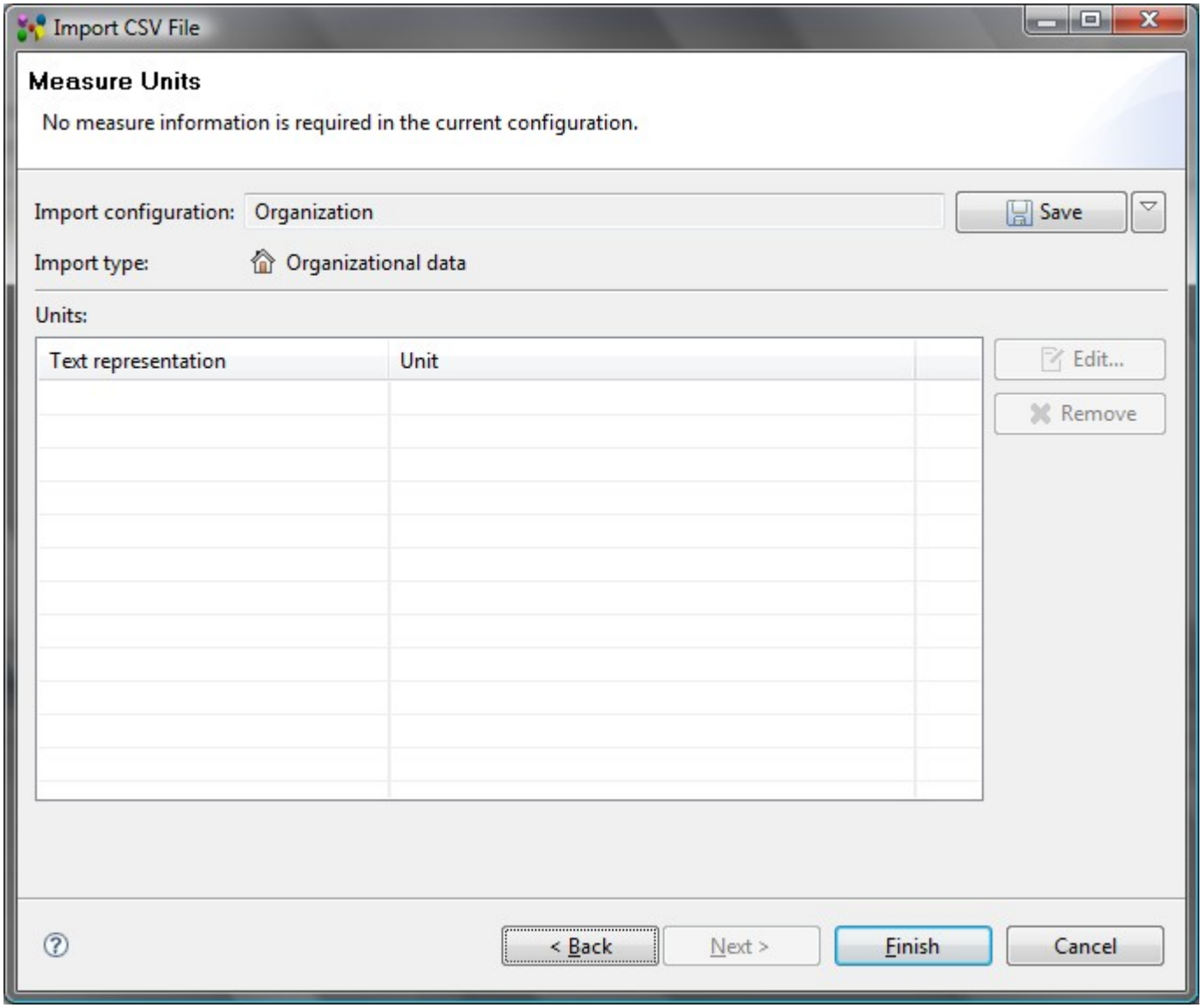

Figure 12. Import CSV file wizard page, Measure Units page

- After all needed parameters are set, press **Finish** button to import data form CSV file.

To top

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6.3.5. Importing Board Station

Board Station® is a PCB-CAD System from Mentor Graphics for designing Printed Circuit Boards (PCB).

The **Import Board Station** feature allows to convert Board Station design files into IDA-STEP. This conversion process has two steps. The first step is to get PCB design directory, which is created automatically after a particular PCB design is created in Board Station and also generate geometry ASCII file (*.txt). In the second step the PCB directory content and geometry file are then imported into IDA-STEP.

First step: Data preparation in Board Station

Start **Mentor Board Station** to get the following data, needed for the import to IDA-STEP:

- PCB directory,
- ASCII geometry file.

Create/load an applicable PCB design project in Mentor Board Station. Automatically the particular PCB design directory is created inside the project (if a new project was created, in other way this directory must already be present in the project). After that you need to generate geometry file in ASCII (*.txt) format. This file must be created using `Save ASCII geoms` command (for more details read Mentor BoardStation™ manual). **Note.** This geometry file may be named arbitrary.

Second step: Import PCB directory and geometry ASCII file into IA-STEP

Follow the routine to import Board Station PCB and Printed Circuit Assembly (PCA) data into IDA-STEP:

- Select in the main menu **File -> Import**.
- Select **Electronics ->** **Board Station** item in the appeared **Import** wizard page. The **Import Board Station** wizard page appears on the screen (see Figure 1):

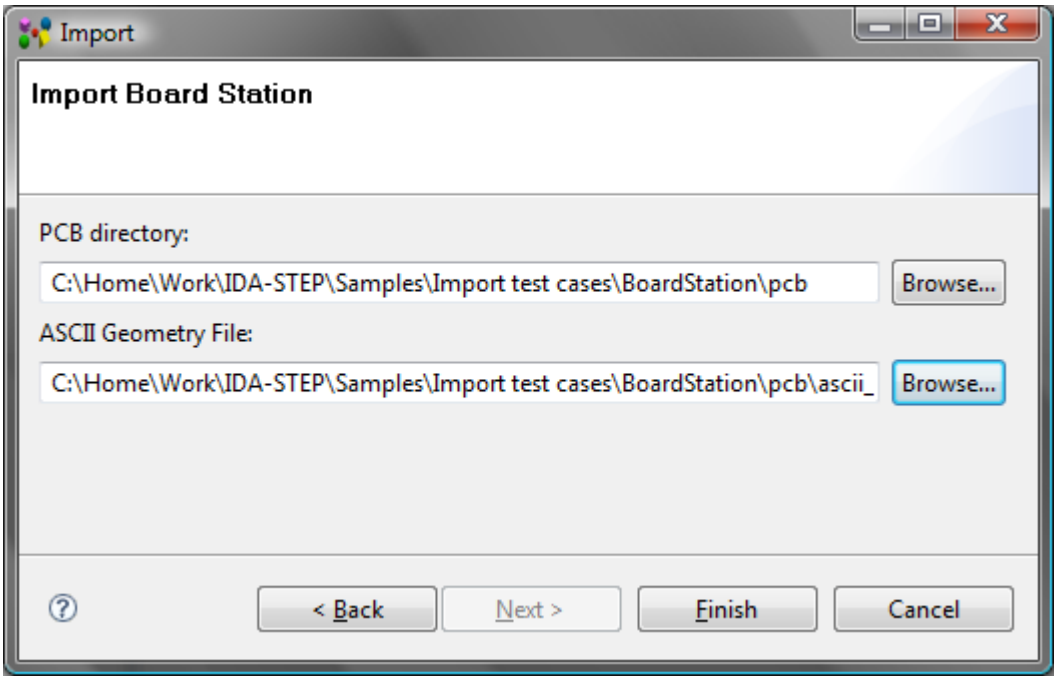

Figure 1. Import Board Station wizard page

- Locate a path of a *PCB* directory in Mentor BordStation™ project in the **PCB Directory** field by using **Browse** button.
- Locate a path to a *ASCII geometry file* in the **ASCII Geometry File** field. If this file is present in the recently selected **PCB directory**, the path of this file is located automatically into the **ASCII Geometry File** field.
- Press **Finish** button to confirm your selections and start import.

After the import is completed, the generated data are opened in IDA-STEP (see Figure 2). Ensure that the **PCB/PCA** perspective is selected.

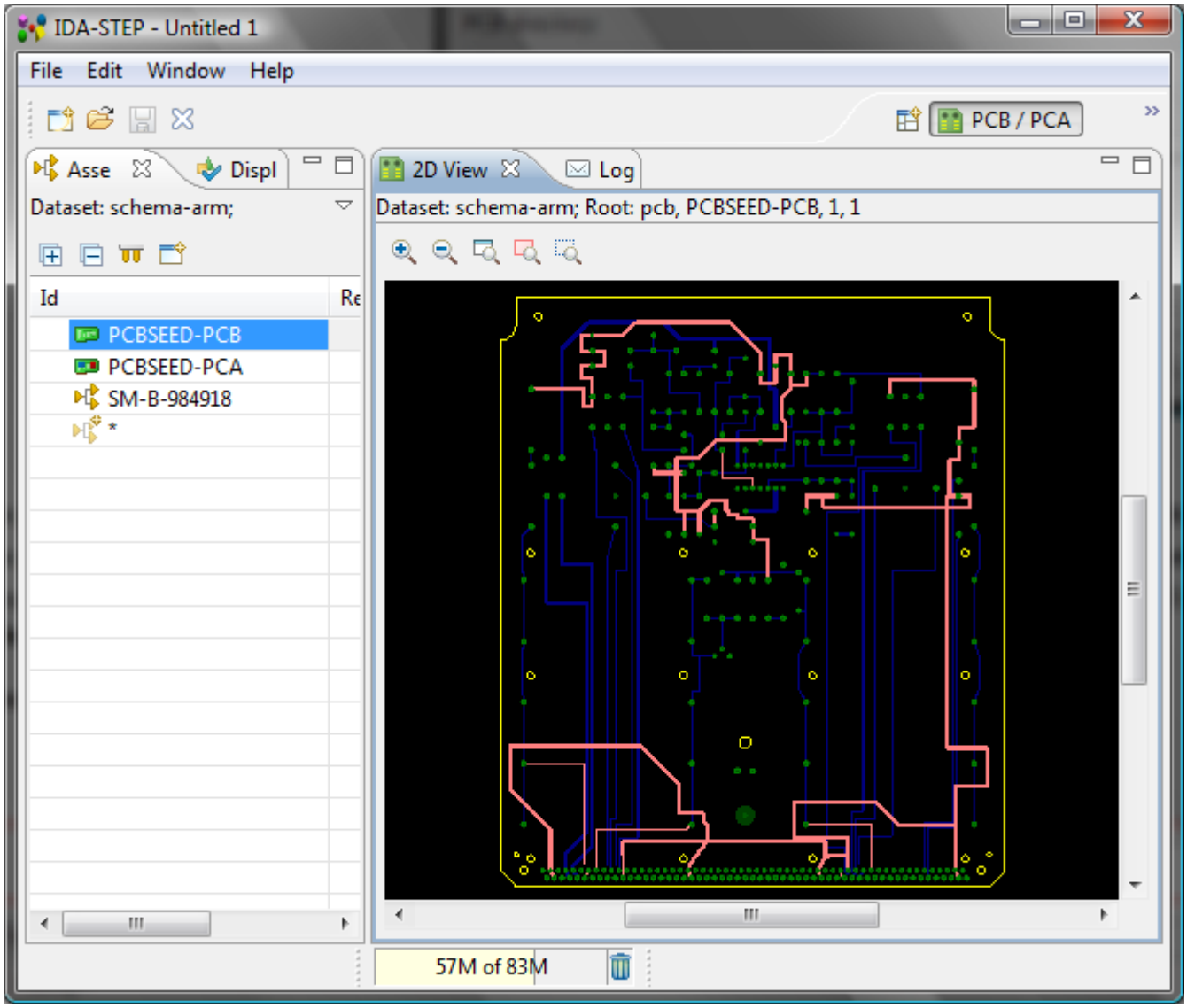

Figure 2. Imported PCB and PCA data in the PCB/PCA Perspective

To top

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6.3.6.1. Importing Allegro

Allegro™ PCB Editor is a PCB layout system from Cadence Design Systems used to design Printed Circuit Boards (PCB).
The **Import Allegro** feature allows to convert Allegro board files (*.brd) into IDA-STEP. This conversion process has two steps. The first step is to run a special `extracta` command inside Allegro PCB Editor to dump out ASCII files. In the second step these files are then imported into IDA-STEP. See Import Allegro for details on supported features.

First step: Data preparation in Allegro PCB Editor

Start Allegro PCB Editor and load board (*.brd) file to generate Allegro ASCII files (*.txt).
The ASCII text files have to be generated by using `extracta` command (refer Allegro PCB Editor User Guide for details). The specific script file, named *lksoft_allegro_extract.txt*, should be used while performing `extracta` command. This file is located in the *config* folder, included in IDA-STEP installation folder (*../plugins/net.jsdai.xim.ap210.allegro_*/config*). The exact number (7) of names of

the files to be generated, have to be specified while running the script. **Note:** You can specify any names for these files.
The order of files to be generated according the content:

1. Component,
2. Pins,
3. Pads,
4. Geometry,
5. Layers,
6. Nets,
7. Connectivity.

The Allegro ASCII text files of one design, should be located in a particular folder.

Second step: Import *.txt files into IDA-STEP

Follow the routine to import Allegro ASCII text files:

- Select **File -> Import** menu item in the main menu.
- Select **Import Allegro** item in the appeared **Import** wizard dialog window. The **Import Allegro** wizard dialog window is invoked to select data folder (see Figure 1).

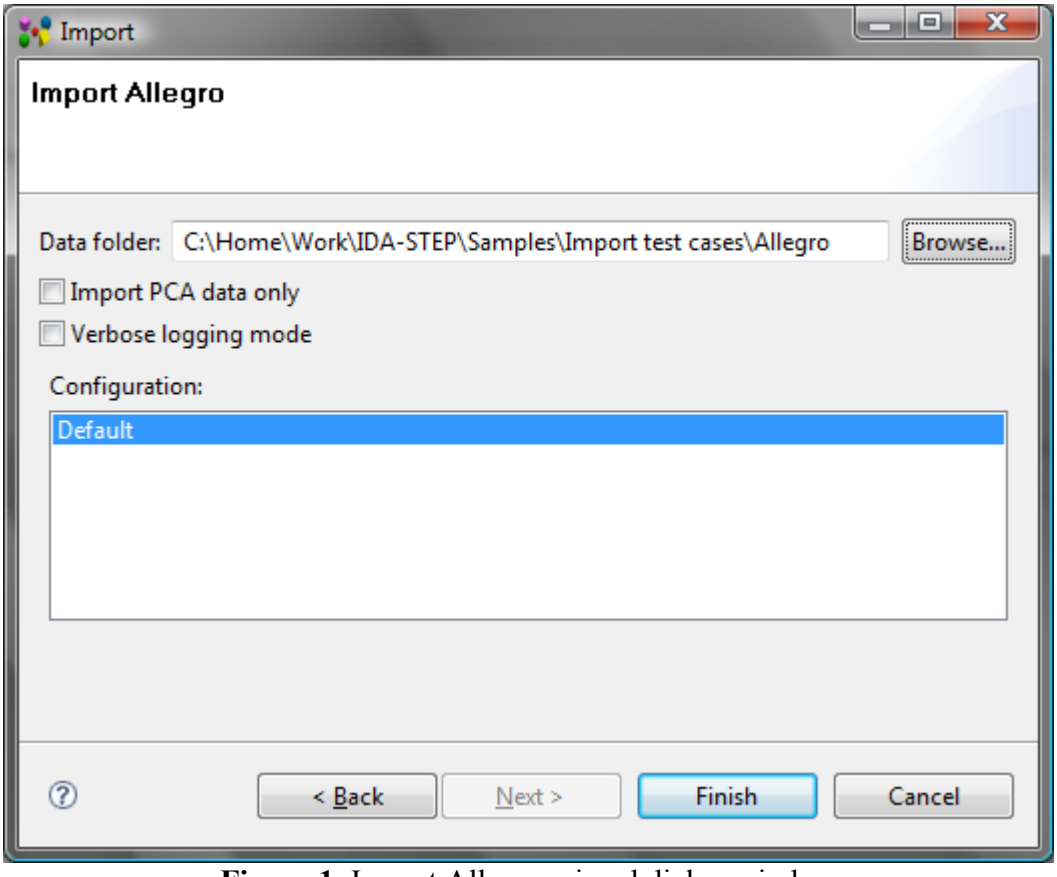

Figure 1. Import Allegro wizard dialog window

- Select a folder in the file system containing Allegro ASCII text files by using **Browse** button next to the **Data folder** field.
  Note: The folder must contain the ASCII text files of only one design.
- Choose whether the complete design PCA and PCB is imported, or import PCA data only by selecting/unselecting **Import PCA data only** option.
- Details on how Allegro data is converted into STEP are defined in special configuration files, which are provided by LKSoft. Available configurations are listed in the **Configuration** list. Typically the *Default* configuration is used.

- Click **Finish** button to start the import process.

After an import is completed, you can see the log details of the import (a path to the source folder, an import duration, a path to the log file) in the **Log** view, which is opened in the following way:

- Select **Window --> Show view --> Other** in the main menu. The **Show view** dialog window appears.
- Select **Other --> Log** item. The **Log** view appears on the screen (see Figure 2).

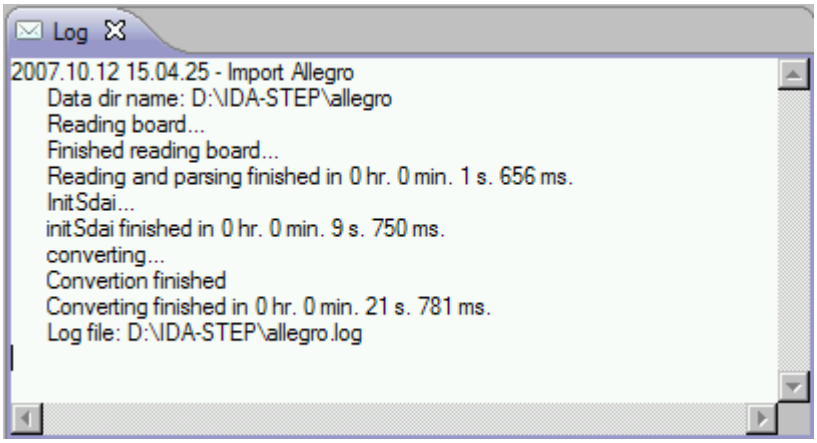

Figure 2. Log view

To view imported data in IDA-STEP (see Figure 3) use the **PCB/PCA** perspective (select **Window--> Open Perspective --> Other --> PCB/PCA**).

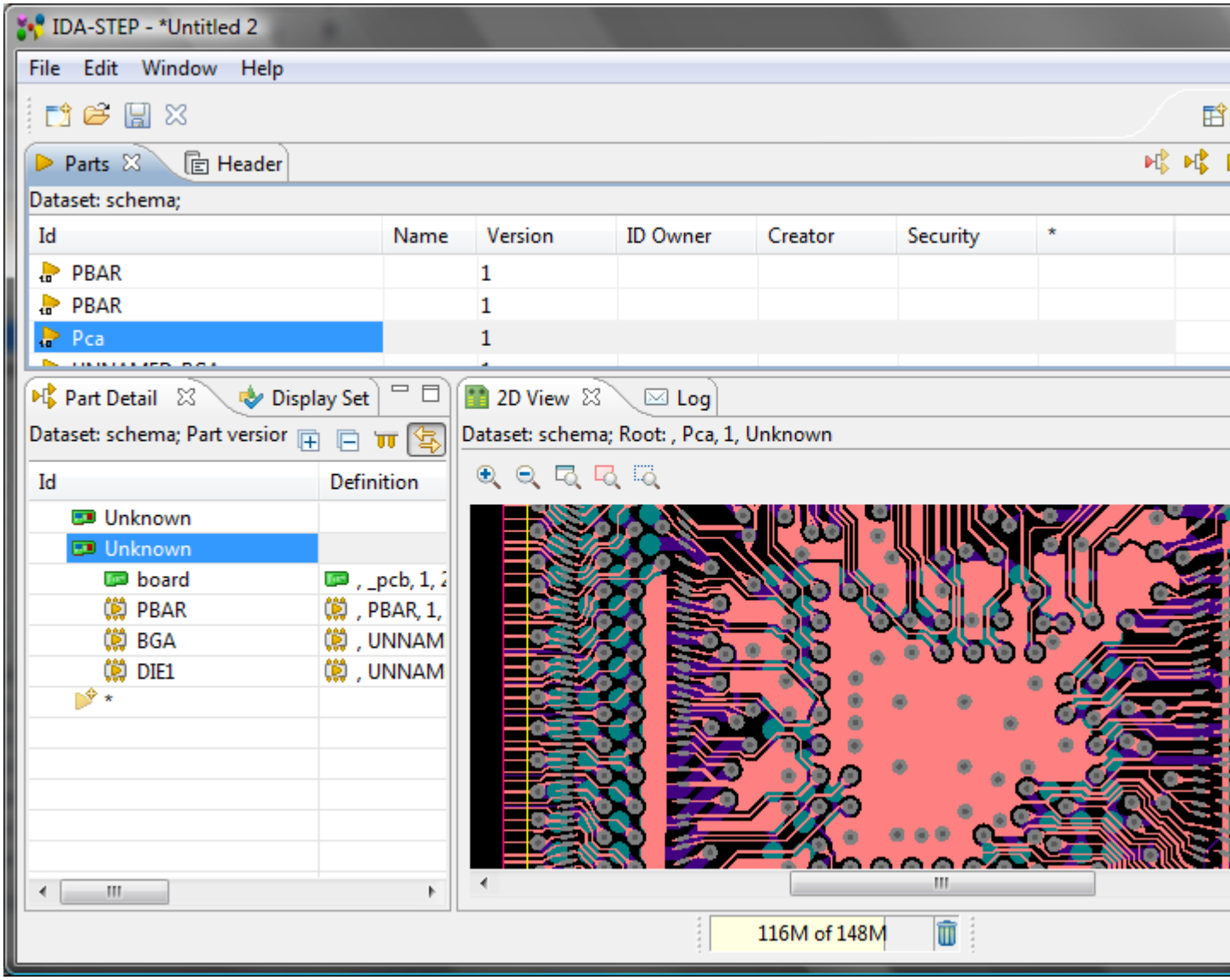

Figure 3. Imported Allegro data in IDA-STEP

To top

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6.3.6.2. Importing Example Data

Follow the routine to import example Allegro ASCII text files:

- Start **Import** wizard by selecting **File --> Import** in the main menu.
- Select **Import Allegro** item in the appeared layout. The second **Import** wizard dialog window appears to select data folder.
- Click **Browse** button to locate the folder. Browse to the IDA-STEP installation folder, enter into *.../plugins/net.jsdai.xim.ap210.allegro.imp.example* folder, select *cds_routed* folder (see Figure 1).
- The selected folder name appears in the **Folder** field.

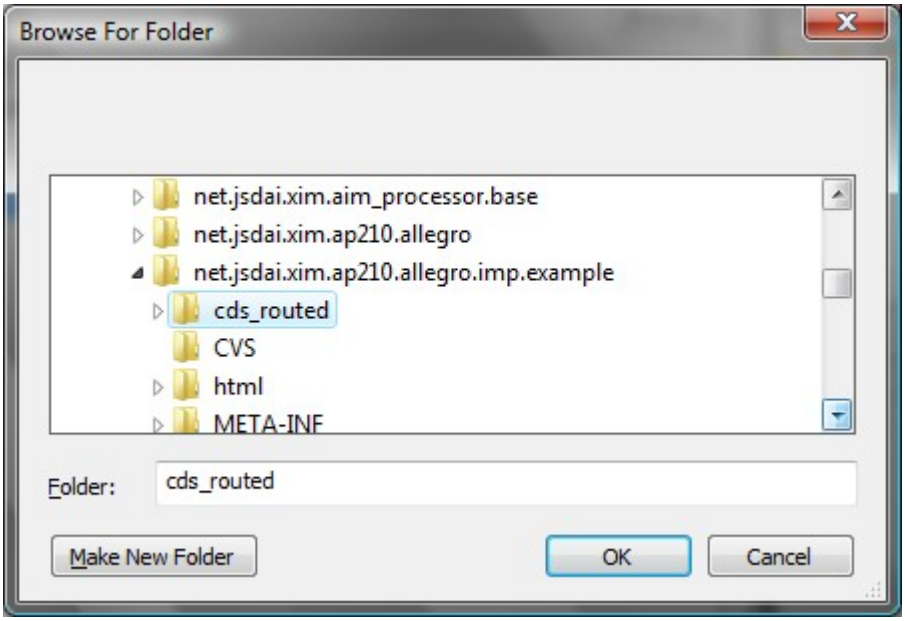

Figure 1. *cds_routed* folder with Allegro ASCII text files

- Click **OK** button to confirm the selection.
- A path to the selected data folder appears in the **Data source** field (see Figure 2).

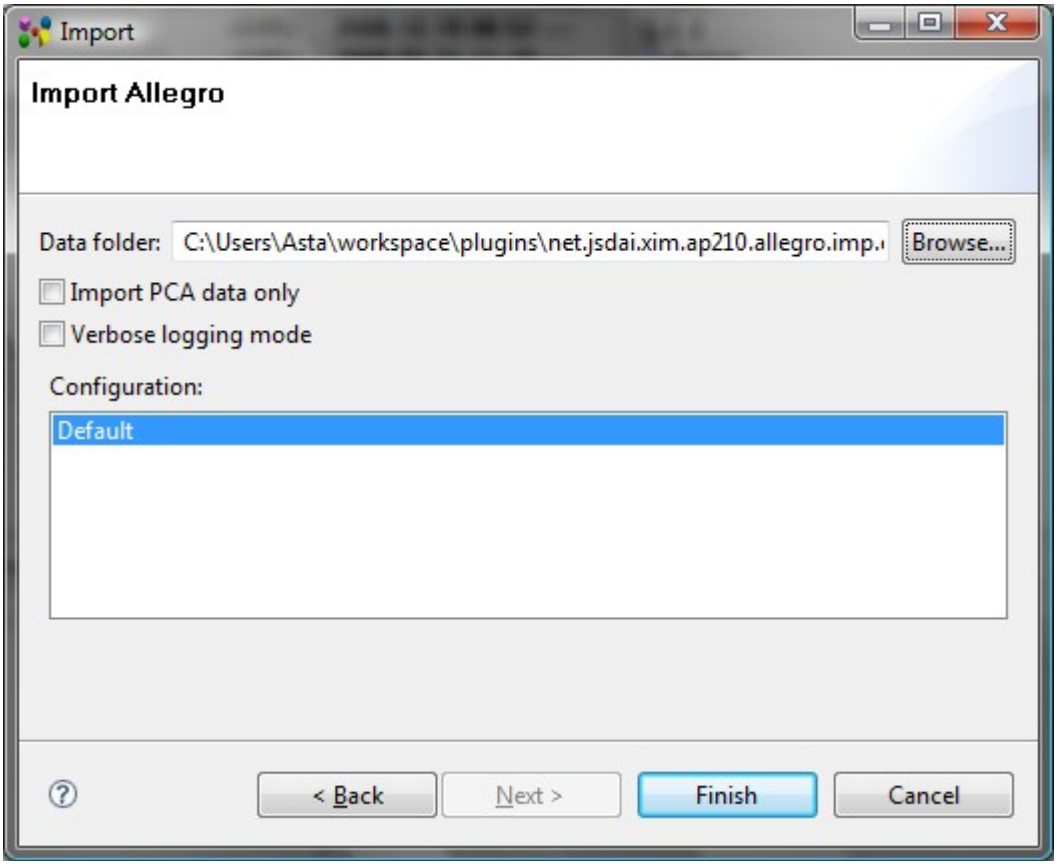

Figure 2. Import Allegro wizard dialog with a specified path to the data folder

- Choose whether the complete design PCA and PCB is imported, or import PCA data only by selecting/unselecting **Import PCA data only** option.
- Details on how Allegro data is converted into STEP are defined in special configuration files, which are provided by LKSoft. Available configurations are listed in the **Configuration** list. Typically the *Default* configuration is used.
- Click **Finish** button to import data.

To view imported data use the **PCB/PCA** perspective (select **Window--> Open Perspective --> Other --> PCB/PCA**).

To top

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6.3.7. Importing CircuitCAM

LPKF CircuitCAM PCB is a CAM system from LKSoft / LPKF Laser & Electronics AG, which is used for designing Printed Circuit Boards (PCB).
The **Import CircuitCAM** feature allows to convert CircuitCAM files (*.cam) into IDA-STEP. This conversion process has two steps. The first step is to make an auxiliary XML design file. In the second step these files are then imported into IDA-STEP.
See Import CircuiCAM for details on supported features.

First step: Auxiliary XML file creation

You need to specify auxiliary (further we will use name AUX) file in order to import CircuitCAM designs into IDA-STEP.

See below a sample AUX file . This file, named *info.xml,* can be found in IDA-STEP installation folder (*.../plugins/net.jsdai.xim.ap210.ccam/config*):

```
<?xml version="1.0" encoding="UTF-8" ?>
-    <design_info layout="Main">
-        <stratum_stack>
      <stratum name="s01" template="soldermask" level="s01" thickness="200" />
      <stratum name="l01" template="electrical" level="l01" thickness="35" />
      <stratum name="dielectric1" template="dielectric" thickness="1000" />
      <stratum name="l02" template="electrical" level="l02" thickness="35" />
      <stratum name="s02" template="soldermask" level="s02" thickness="200" />
    </stratum_stack>
    <board_outline level="BoardOutline" />
  </design_info>
```

In essence AUX file is a special XML file, that contains:

1. identification of CircuitCAM layout that contains design information;
2. mapping between CircuitCAM levels and IDA-STEP stratums;
3. identification of CircuitCAM level that contains board outline information.

Populate an AUX file for a new design by editing the sample AUX file. In order to specify (1) you need to edit layout attribute of `design_info` element. In most cases a name for a layout, containing design information, is `Main` and this is exactly what is specified in the sample AUX file. Here is the appropriate snippet of AUX file:

```
<design_info layout="Main">
```

In order to specify (2) you need to edit `stratum_stack` element. This element contains a list of information for each stratum in the design. Each stratum is represented by stratum element. This element has four attributes:

- **name** - this mandatory attribute specifies a name of a stratum. In most cases this name is the same as a name of an appropriate level in CircuitCAM;
  Note. The value of this attribute should be unique.
- **template** - this mandatory attribute specifies a type of a stratum. Currently CircuitCAM IDA-STEP importer supports three types of stratums:
  - **electrical** - a stratum, containing circuit geometry;

        - **soldermask** - a stratum, containing solder-mask geometry;
        - **dielectric** stratum should contain no stratum and is present in stratum stack to represent dielectric/core material.
    - **level** - this optional attribute specifies a name of a CircuitCAM level, which geometry will be used for this stratum. **Note**, it is illegal to specify a level for dielectric stratums;
    - **thickness** - this optional attribute specifies a stratum's thickness in micro meters. If it is not provided, then the default thickness of 0.1 mm is used.

Here is the appropriate snippet of AUX file:

```
  <stratum_stack>
    <stratum name="s01" template="soldermask" level="s01" thickness="200" />
    <stratum name="l01" template="electrical" level="l01" thickness="35" />
    <stratum name="dielectric1" template="dielectric" thickness="1000" />
    <stratum name="l02" template="electrical" level="l02" thickness="35" />
    <stratum name="s02" template="soldermask" level="s02" thickness="200" />
</stratum_stack>
```

This snippet specifies the stratum stack consisting of 5 stratums. **Note**, it is important to specify stratums in the correct order going from the top to the bottom of a board. The top solder-mask stratum **`s01`** uses the geometry from the level `s01`. The bottom solder-mask stratum `s02` uses the geometry from the level `s02`. The top circuit/electrical stratum `l01` uses the geometry from the level `l01`. The bottom circuit/electrical stratum `l02` uses the geometry from the level `l02`. The stratum `dielectric1` has no geometry and represents the core of PCB. **Note**, currently it is impossible to specify a thickness for a stratum and as a result the CircuitCAM IDA-STEP importer uses the default thicknesses (0.1 mm). This issue will be addressed in the upcoming release by allowing the user to specify a new thickness attribute for a stratum.

In order to specify (3) you need to edit a level attribute of the `board_outline` element. Here is an appropriate snippet of the AUX file specifying the `BoardOutline` level as the source for the board outline geometry:

```
<board_outline level="BoardOutline" />
```

Additionally it is recommended to perform the *union* operation for the geometry in each CircuitCAM level before importing CircuitCAM file into IDA-STEP. This would provide much better final result when converting geometry to 3D.

Second step: Import *.cam and *.xml files into IDA-STEP

Follow the routine to import CircuitCAM PCB design data into IDA-STEP:

- Select in the main menu **File -> Import**.
- Select **Electronics ->** **CircuitCAM** item in the appeared **Import** wizard page. The **Import CircuitCAM** wizard page appears on the screen (see Figure 1):

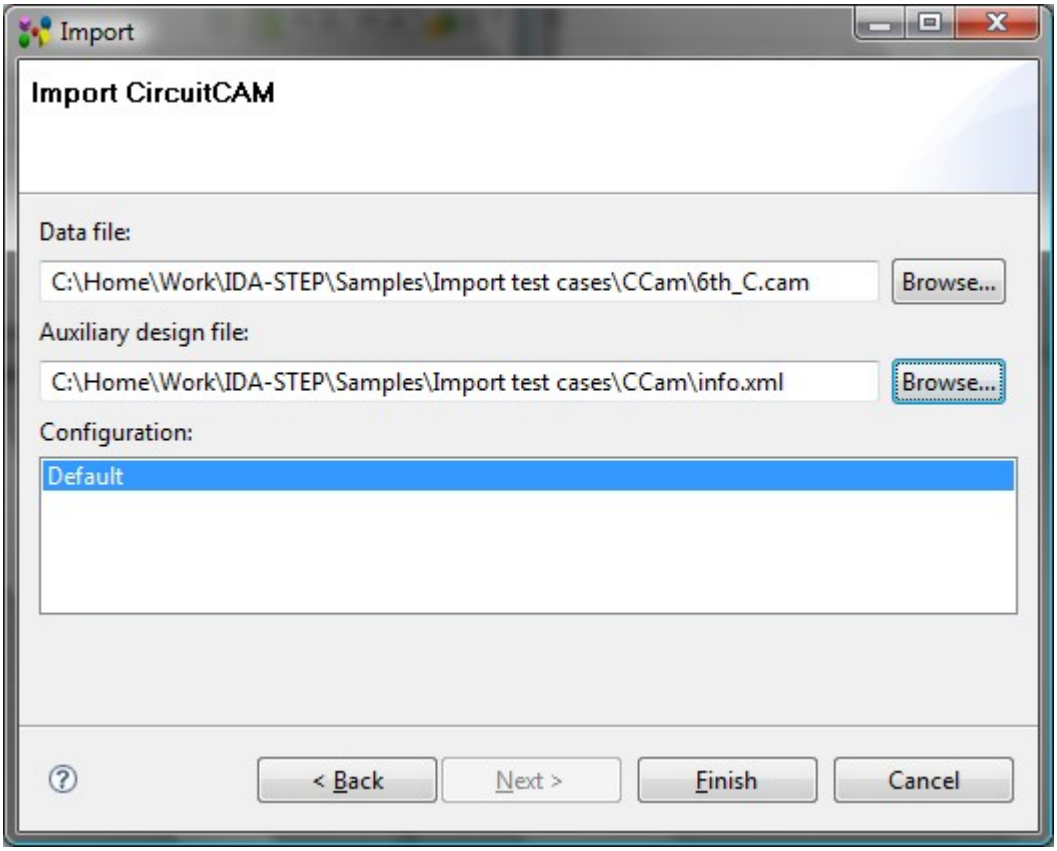

Figure 1. Import CircuitCAM wizard page

- Locate a path to a *CAM* file in the **Data file** field by using **Browse** button.
- Locate a path to an auxiliary *XML* file in the **Auxiliary design file** field by using **Browse** button.
- Details on how CircuitCAM data is converted into STEP are defined in special configuration files, which are provided by LKSoft. Available configurations are listed in the **Configuration** list. Typically the *Default* configuration is used.
- Press **Finish** button to import selected data.

After the import is completed, the generated data are opened in the IDA-STEP (see Figure 2). Ensure that the **PCB/PCA** perspective is selected.

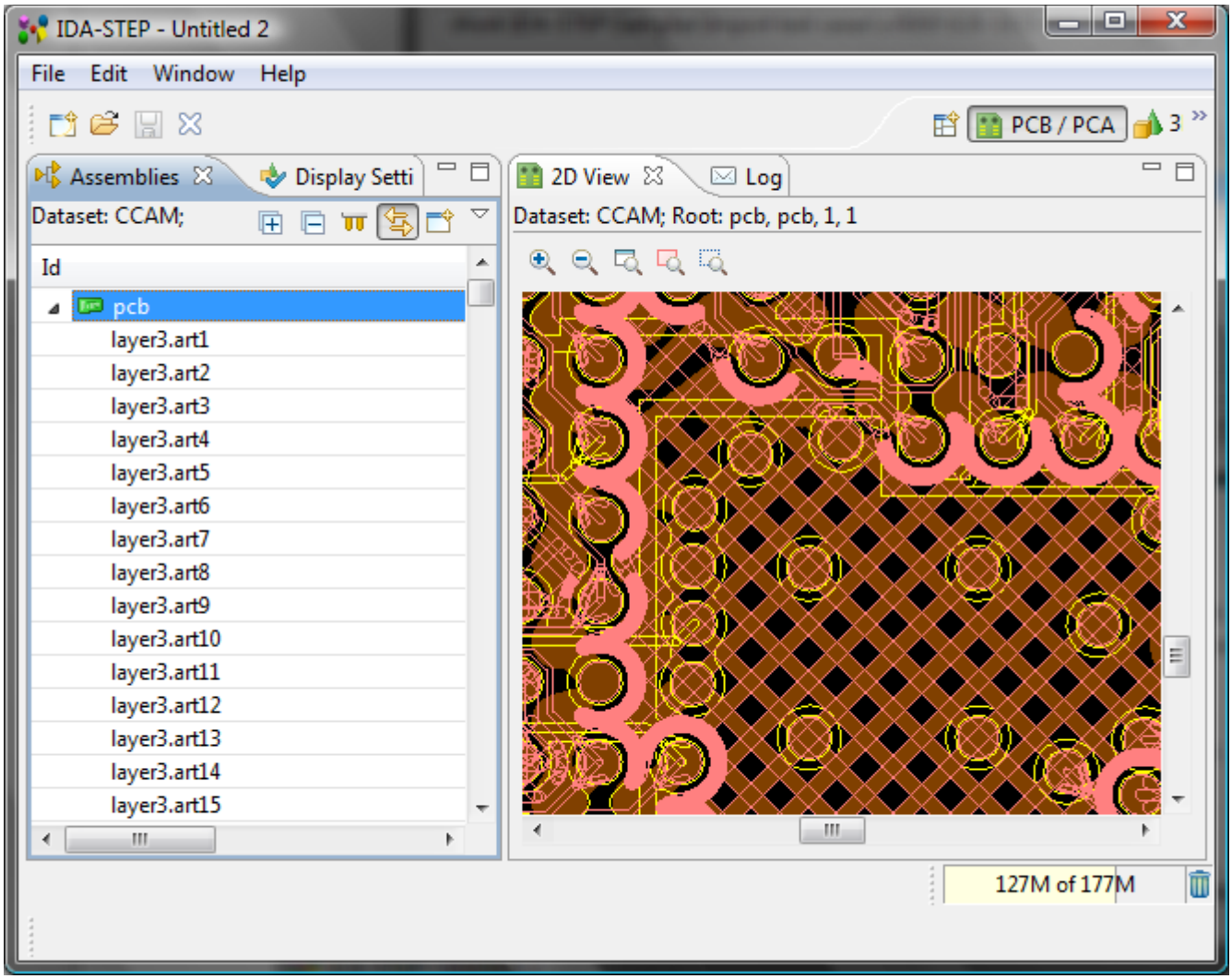

Figure 2. Imported CircuitCAM PCB data in IDA-STEP

To top

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6.3.8. Importing PADS

PADS® is a PCB-CAD System from Mentor Graphics for designing Printed Circuit Boards (PCB). The **Import PADS** wizard allows to import design from PADS data files (*.asc) into IDA-STEP. It requires the PADS ASCII data files (*.asc) to import data. **Note.** While exporting data from PADS Layout® to ASCII (*.asc) file, make sure you have selected to export all available data from the PADS design to PADS ASCII file.
Follow the routine to import PADS design data into IDA-STEP:

- Select in the main menu **File -> Import**.
- Select **Electronics ->** **PADS** item in the appeared **Import** wizard page. The **Import PADS** wizard page appears on the screen (see Figure 1):

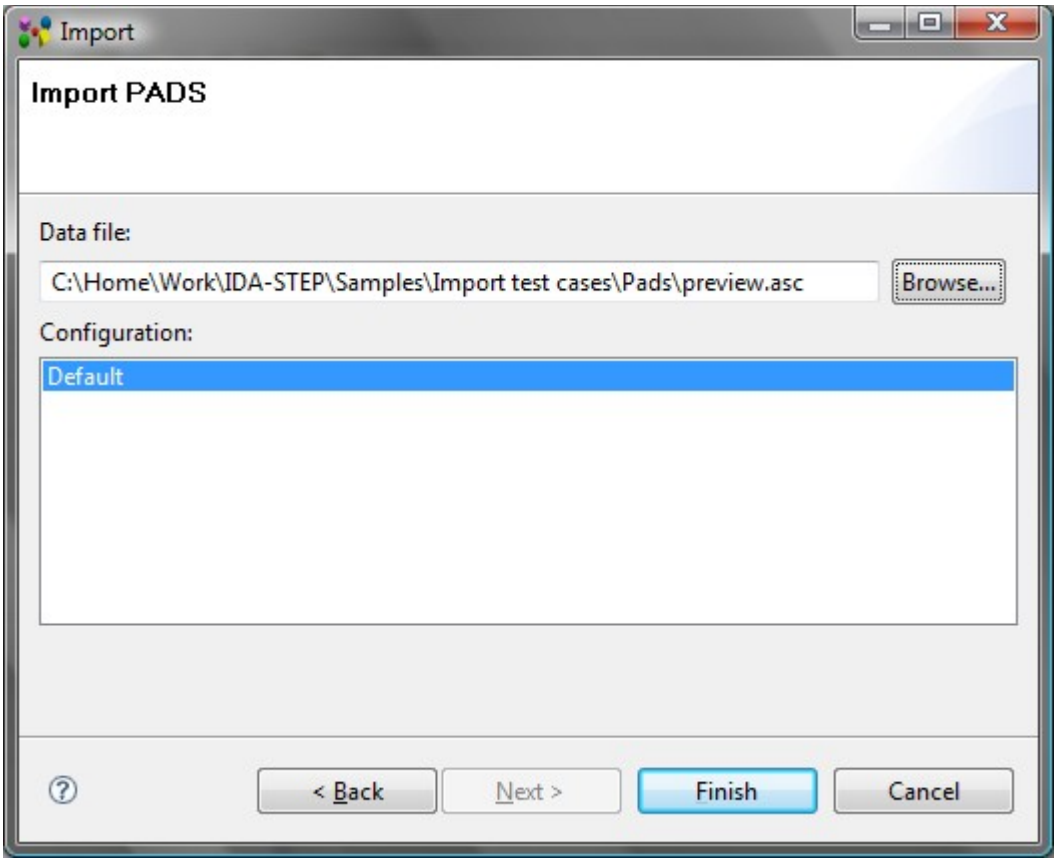

Figure 1. Import PADS wizard page

- Locate a path to an *ASC* file in the **Data file** field by using **Browse** button.
- Details on how EAGLE data is converted into STEP are defined in special configuration files, which are provided by LKSoft. Available configurations are listed in the **Configuration** list. Typically the *Default* configuration is used.
- Press **Finish** button to import selected data.

After the import is completed, the generated data are opened in the IDA-STEP (see Figure 2). Ensure that the **PCB/PCA** perspective is selected.

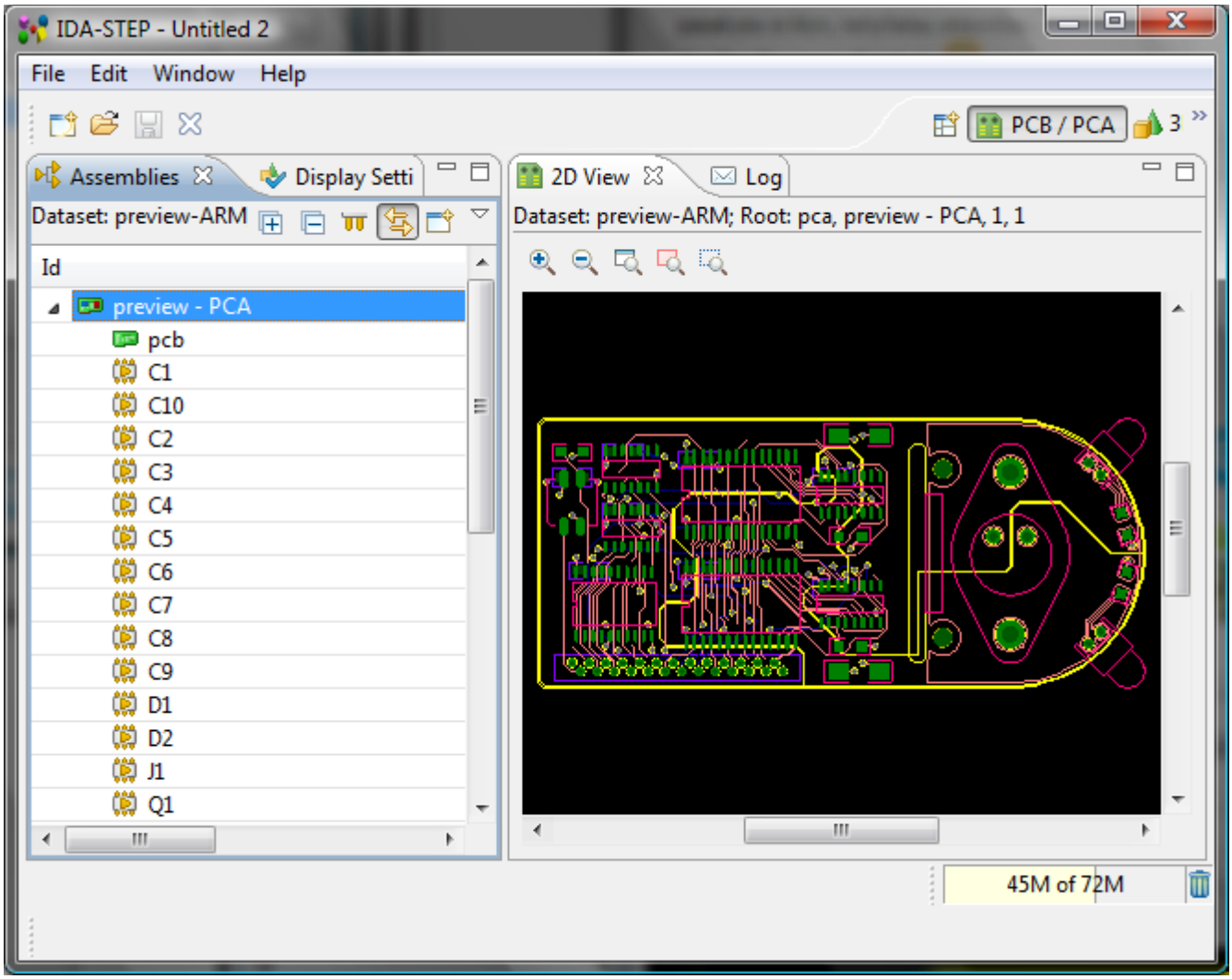

Figure 2. Imported PADS data (PCA and PCB) in IDA-STEP

To top

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6.3.9. Importing CR-5000

CR-5000 is Zuken's printed circuit board (PCB) and advanced packaging layout suite of customizable tools aimed at the Enterprise market.
The **Import CR-5000** wizard allows to import CR-5000 design into IDA-STEP. It requires the following data from CR-5000:

- **PCB ASCII File** (*PCF* extension) - the ASCII file describing the contents of the PCB Database;
- **Footprint ASCII File** (*FTF* extension) - the ASCII file describing the contents of the Footprint Library.

Follow the routine to import CR-5000 PCB and Printed Circuit Assembly (PCA) data into IDA-STEP:

- Select in the main menu **File -> Import**.
- Select **Electronics ->** **CR-5000** item in the appeared **Import** wizard page. The **Import CR-5000** wizard page appears on the screen (see Figure 1):

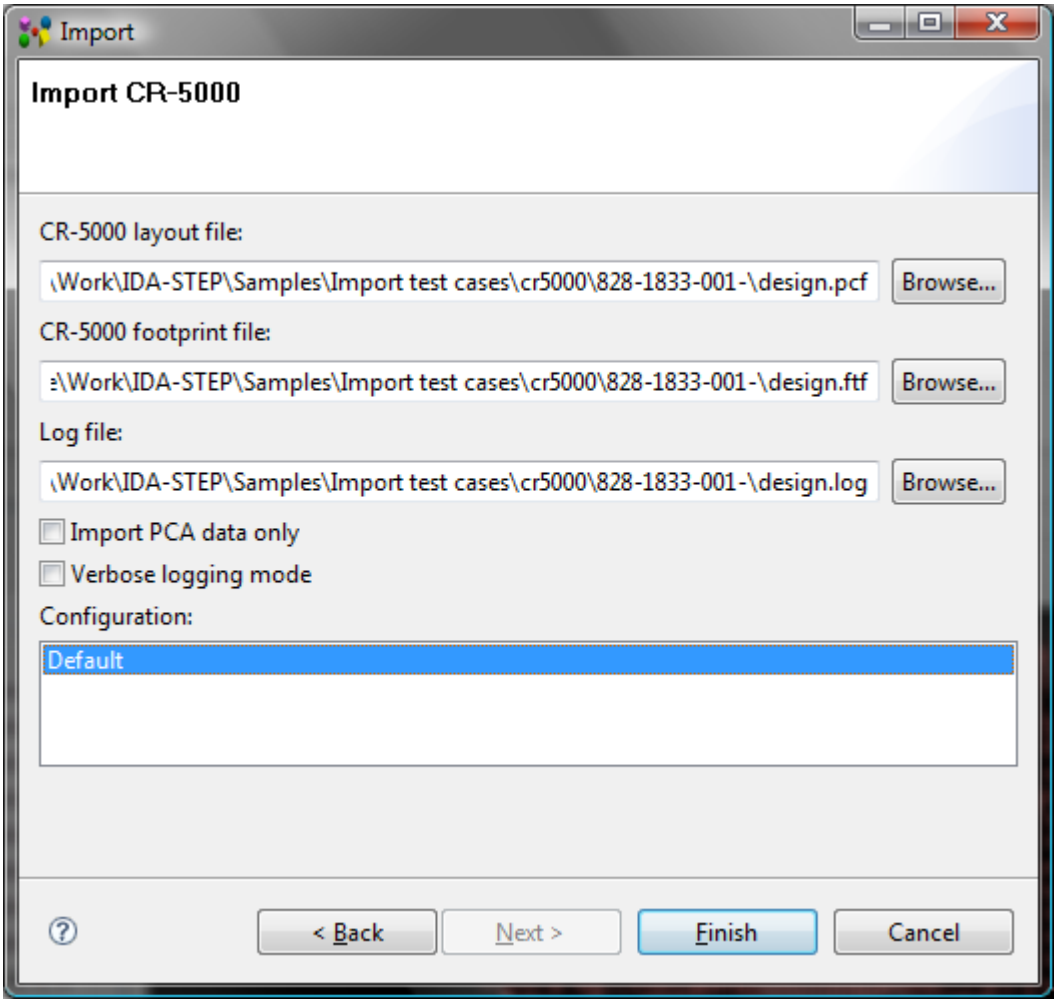

Figure 1. Importing CR-5000 wizard page

- Locate a path to a *PCF* file in the **CR-5000 layout file** field by using **Browse** button.
- Locate a path to a *FTF* file in the **CR-5000 footprint file** field by using **Browse** button.
- Locate a path to a log (*TXT*) file in the **Log file** field by using **Browse** button. This file will contain the import log output. The short import log will be available in the **Log** view.
- Choose whether the complete design PCA and PCB is imported, or import PCA data only by selecting/unselecting **Import PCA data only** option.
- Select the option **Verbose logging mode** to get more detailed information in the log file.
- Details on how CR-5000 data is converted into STEP are defined in special configuration files, which are provided by LKSoft. Available configurations are listed in the **Configuration** list. Typically the *Default* configuration is used.
- Press **Finish** button to import selected data.

After the import is completed, the generated data are opened in the IDA-STEP (see Figure 2). Ensure that the **PCB/PCA** perspective is selected.

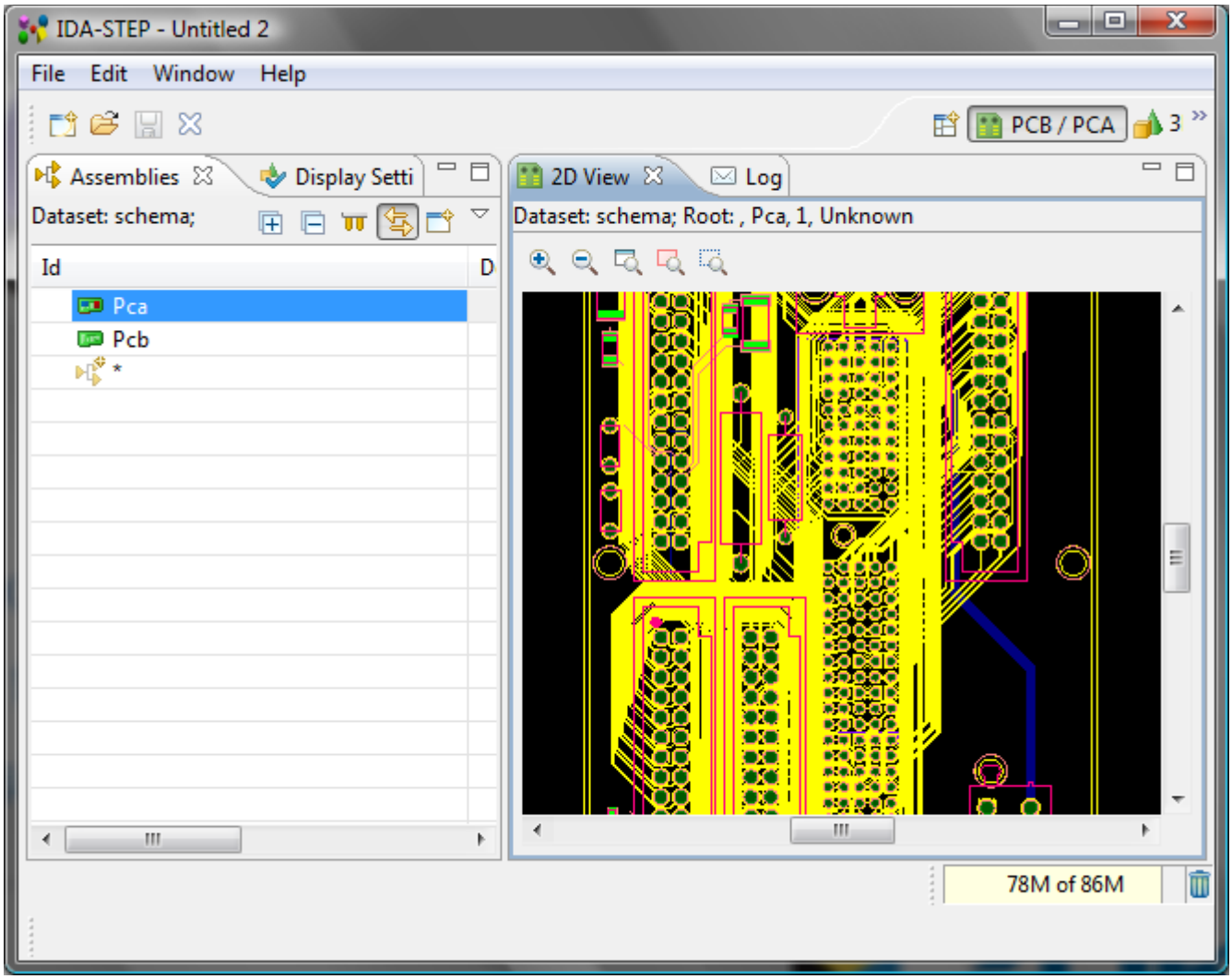

Figure 2. Imported CR-5000 data (PCB and PCA) to IDA-STEP

To top

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6.3.10. Importing Expedition

Expedition™ is a PCB-CAD System from Mentor Graphics for designing Printed Circuit Boards (PCB). The **Import Expedition** wizard allows to import design from Expedition data files into IDA-STEP. It requires the Expedition PCB data folder with the following files, which can be find in one of the the Expedition PCB project folders:

- Cell.hkp,
- JobPrefs.hkp,
- Layout.hkp,
- NetClass.hkp,
- NetProps.hkp,
- Padstack.hkp,
- PDB.hkp.

Follow the routine to import Expedition PCB and Printed Circuit Assembly (PCA) data into IDA-STEP:

- Select in the main menu **File -> Import**.
- Select **Electronics ->** **Expedition** item in the appeared **Import** wizard page. The **Import**

Expedition wizard page appears on the screen (see Figure 1):

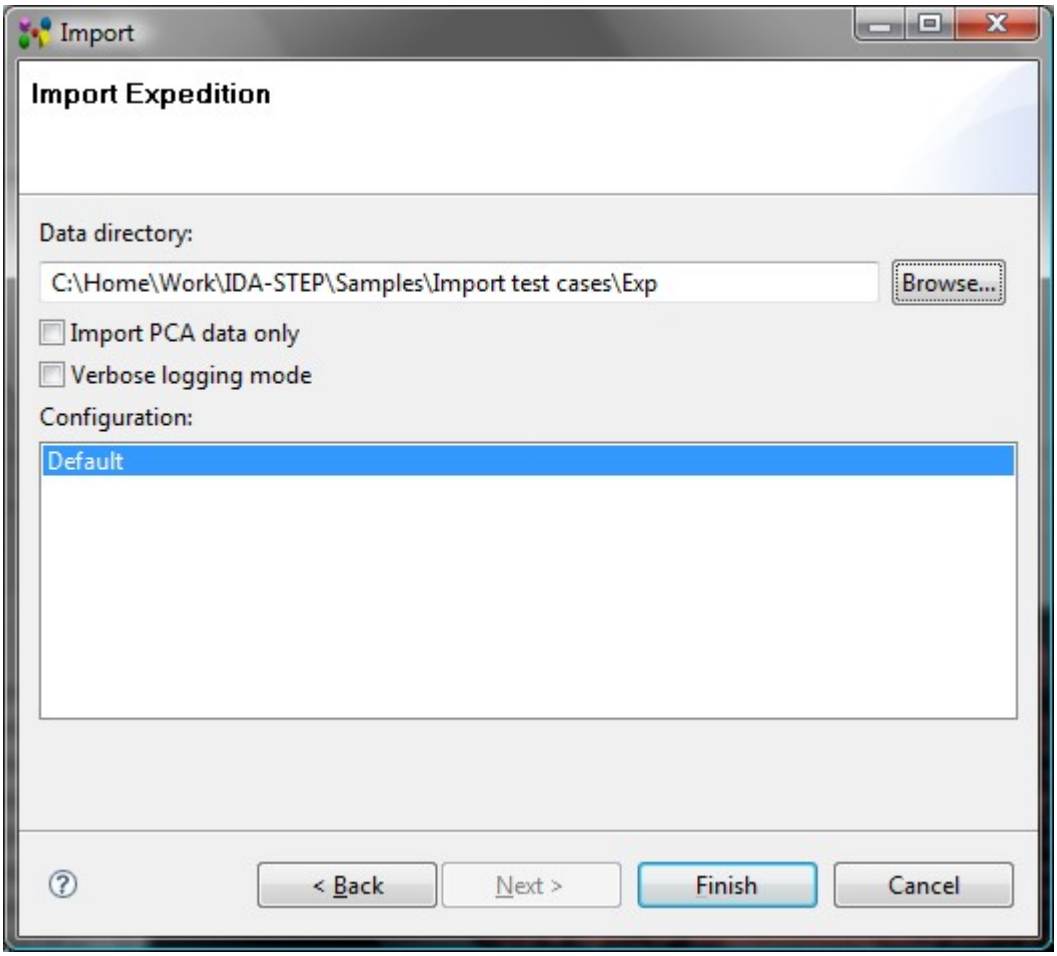

Figure 1. Import Expedition wizard page

- Locate a path to the Expedition folder in the **Data directory** field by using **Browse** button.
- Choose whether the complete design PCA and PCB is imported, or import PCA data only by selecting/unselecting **Import PCA data only** option.
- Select the option **Verbose logging mode** to get more detailed information in the *LOG* file.
- Details on how Expedition data is converted into STEP are defined in special configuration files, which are provided by LKSoft. Available configurations are listed in the **Configuration** list. Typically the *Default* configuration is used.
- Press **Finish** button to import selected data.

After the import is completed, the generated data are opened in the IDA-STEP (see Figure 2). Ensure that the **PCB/PCA** perspective is selected.

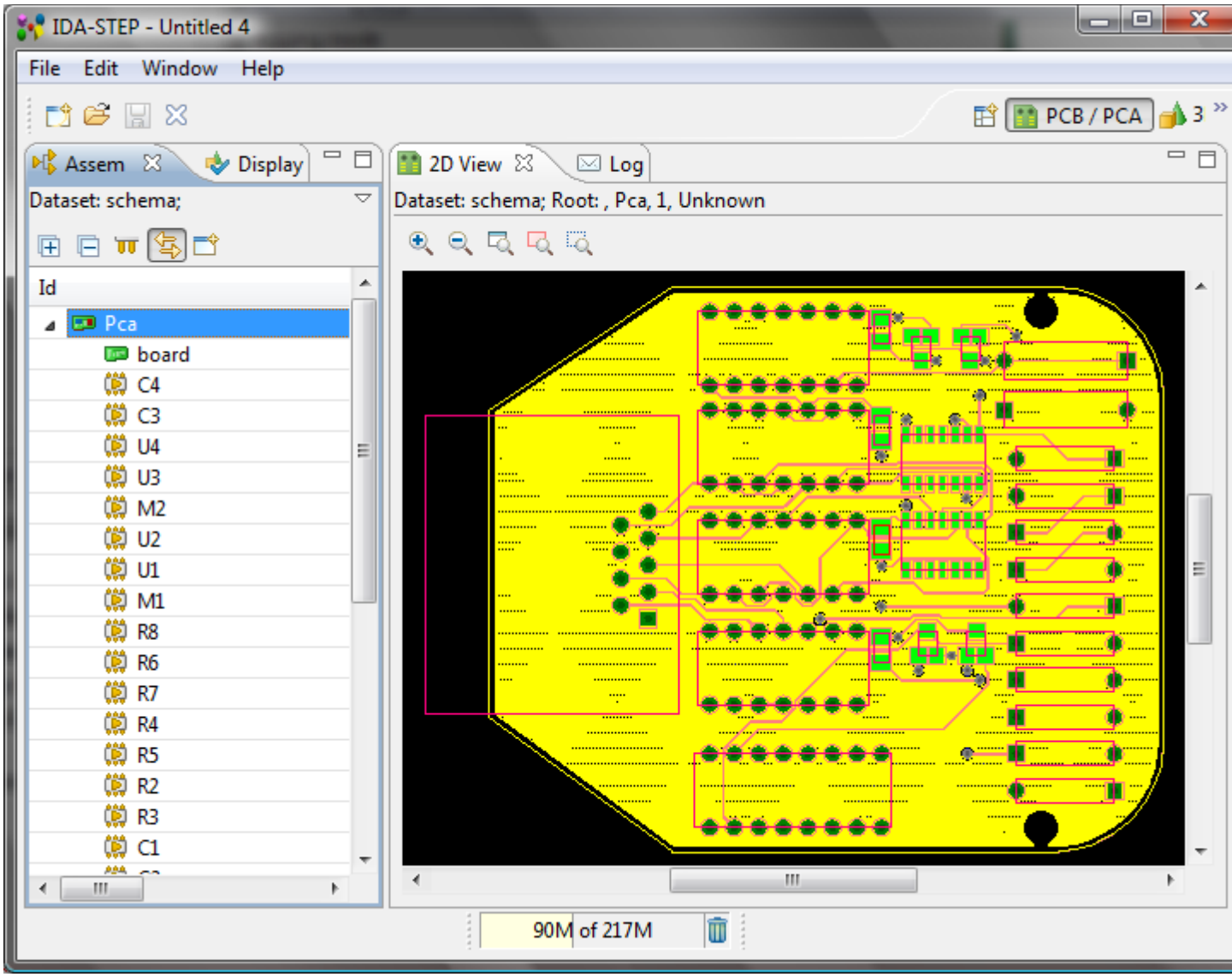

Figure 2. Imported Expedition data in IDA-STEP

To top

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6.3.11. Importing CADIF

CADIF (*.paf) is an ASCII neutral file, which is used for storing PCB, PCA and other information. This file is used by ECAD systems CR-5000, VISULA and CADSTAR from Zuken.
The **Import CADIF** feature allows to import design stored in CADIF file into IDA-STEP. It requires only the CADIF (*.paf) file.
Follow the routine to import CADIF PCB and Printed Circuit Assembly (PCA) data into IDA-STEP:

- Select in the main menu **File -> Import**.
- Select **Electronics ->** **CADIF** item in the appeared **Import** wizard page. The **Import CADIF** wizard page appears on the screen (see Figure 1):

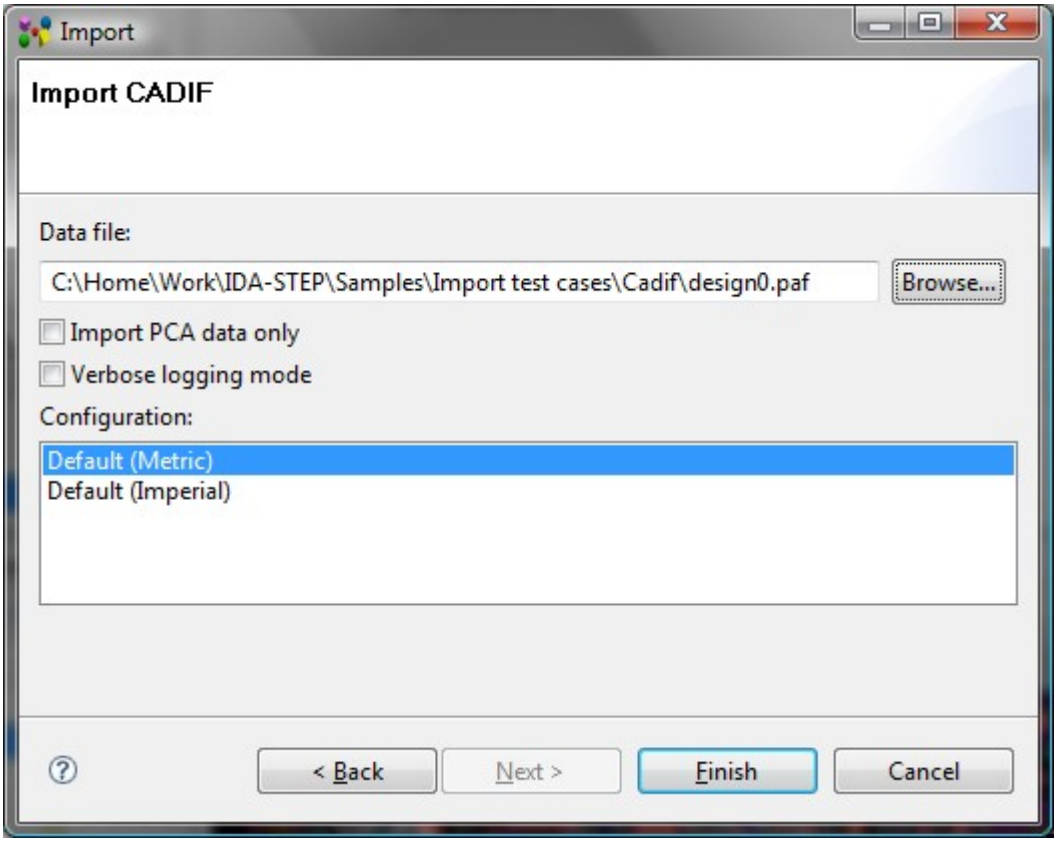

Figure 1. Import CADIF wizard page

- Locate a path to a *PAF* file in the **Data file** field by using **Browse** button.
- Choose whether the complete design PCA and PCB is imported, or import PCA data only by selecting/unselecting **Import PCA data only** option.
- Select the option **Verbose logging mode** to get more detailed information in the *LOG* file.
- Appropriate configuration needs to be chosen in the **Configurations** layout depending on whether imperial or metric units are used for attributes (e.g. height attribute for packages) in CADIF file.
- Press **Finish** button to confirm your selections and start import.

After the import is completed, the generated data are opened in the IDA-STEP (see Figure 2). Ensure that the **PCB/PCA** perspective is selected.

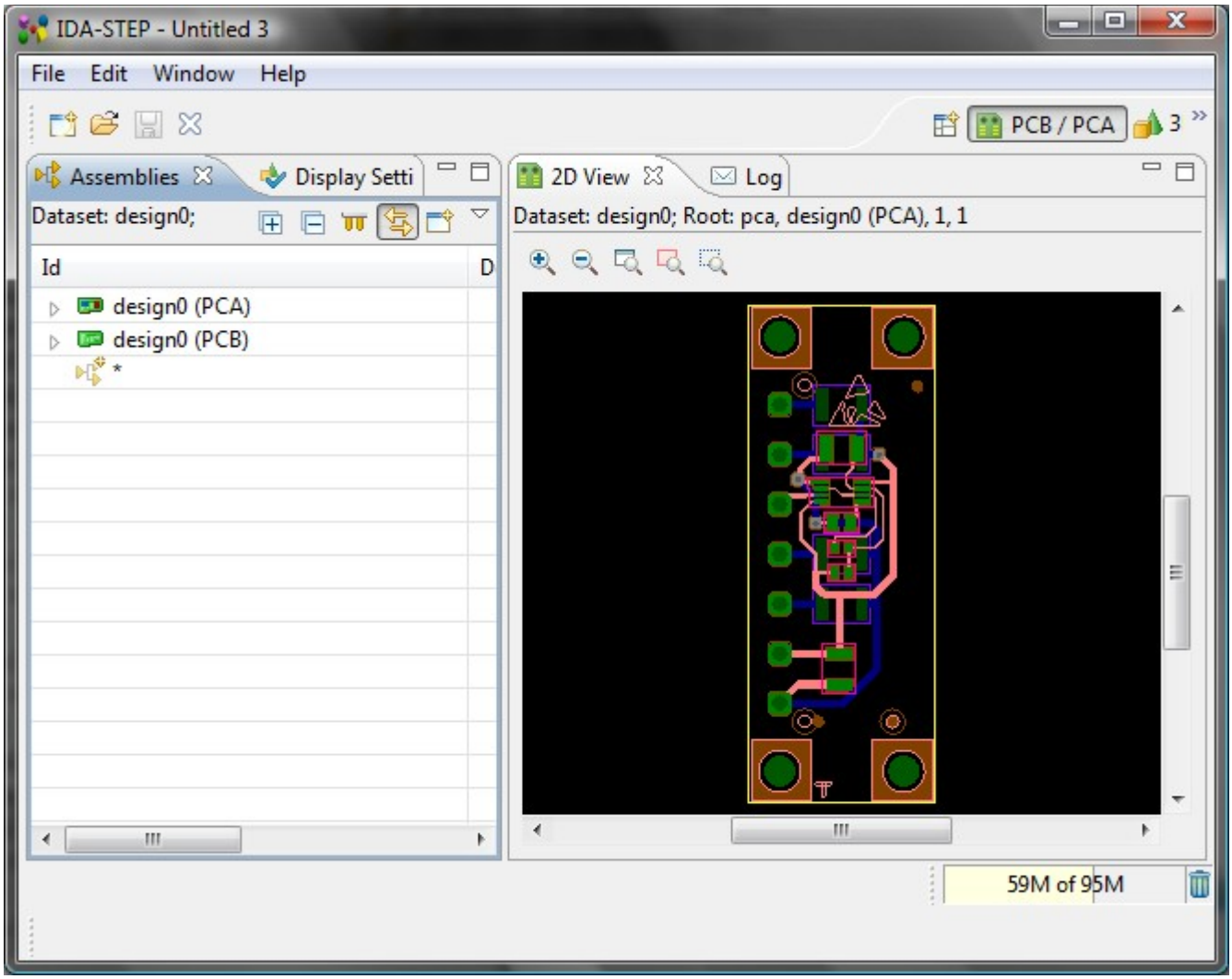

Figure 2. Imported CADIF data (PCB and PCA) to IDA-STEP

To top

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6.4.1. Export Wizard

The **Export** wizard helps to extract or convert data out of IDA-STEP (see Figure 1) into formats other than XIM (for more information on XIM reference http://www.jsdai.net/xim). To start the **Export** wizard select in **IDA-STEP** main menu bar **File -> Export**.

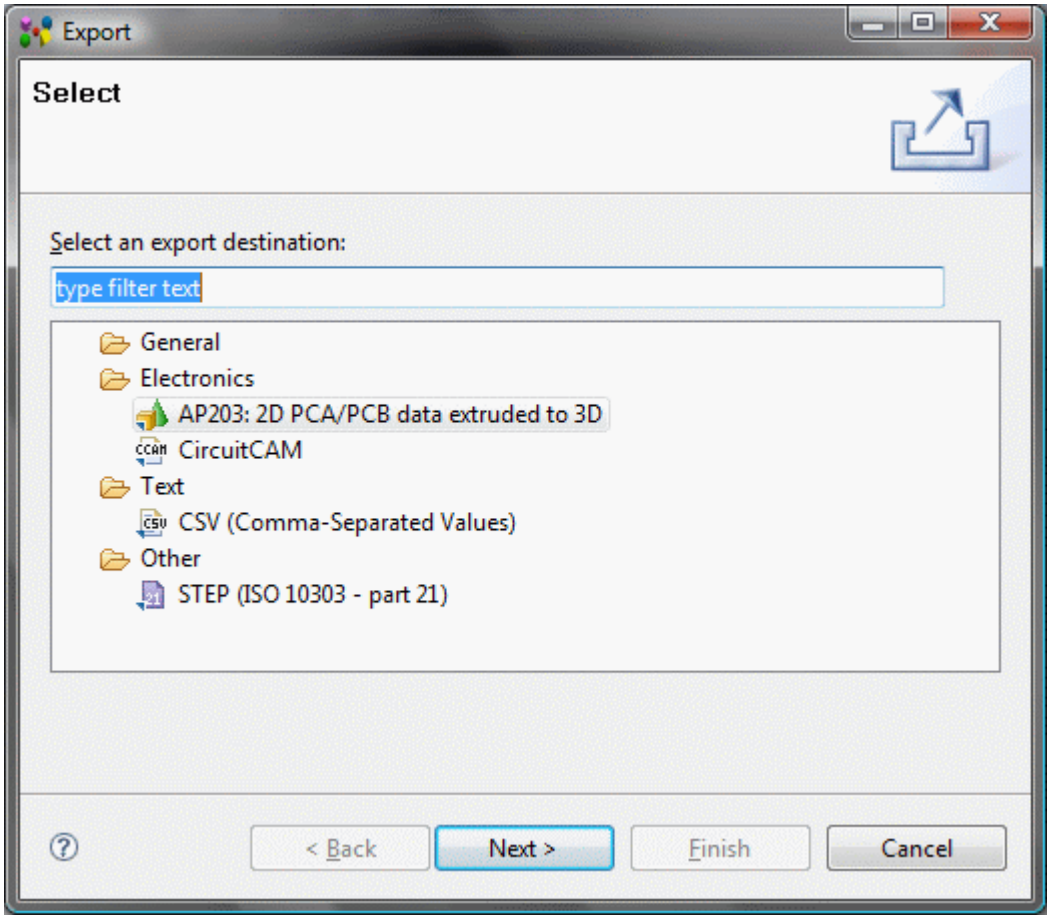

Figure 1. Export wizard page

When the **Export** wizard page appears on the screen you must select what type of export to do. To assist in locating a particular wizard, the text field can be used to show only the wizards that match the entered text.
After an export type is selected, click **Next** button to launch a particular wizard page.

To top

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6.4.3. STEP: 2D PCA/PCB data extruded to 3D

The **STEP: 2D PCA/PCB data extruded to 3D** export allows to generate 3D shape for the selected PCA/PCB and export it as AP203 STEP file. This STEP file can then be opened by almost any mechanical CAD system. The generated STEP file contains not only PCA/PCB 3D shape, but also it's assembly structure.
Follow the routine to export an applicable PCA/PCB design to the AP203 STEP file:

- A particular PCA/PCB design has to be opened or imported in IDA-STEP (see Figure 1).

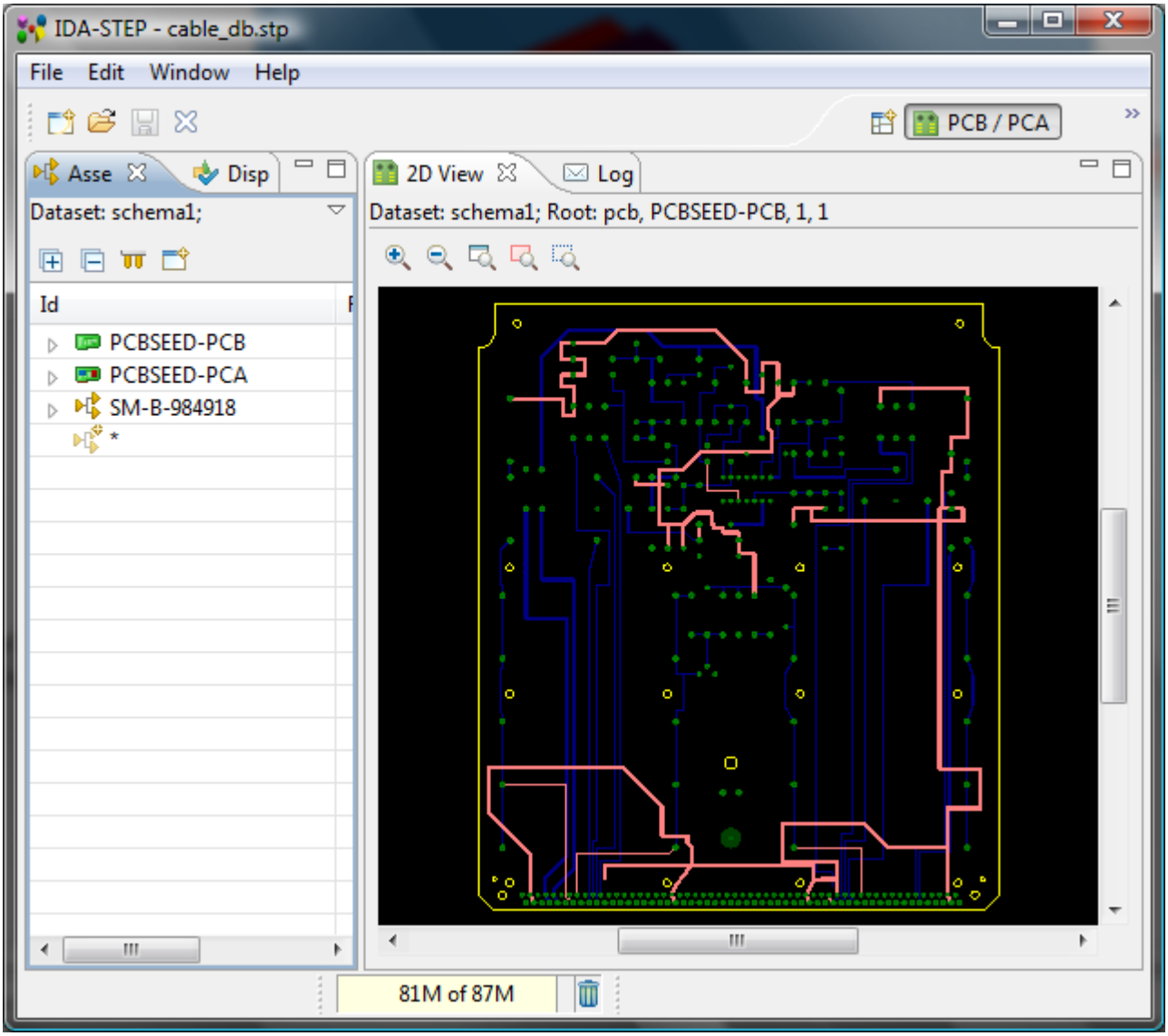

Figure 1. Opened STEP file with PCA and PCB design data and 2D view

- Select in the main menu **File -> Export**. The **Export** wizard appears on the screen.
- Select **Electronics ->** **STEP: 2D PCA/PCB data extruded to 3D** item and press the **Next** button on the bottom section of the wizard page.
- The **Export 3D PCA/PCB Extrusion** wizard page appears (see Figure 2):

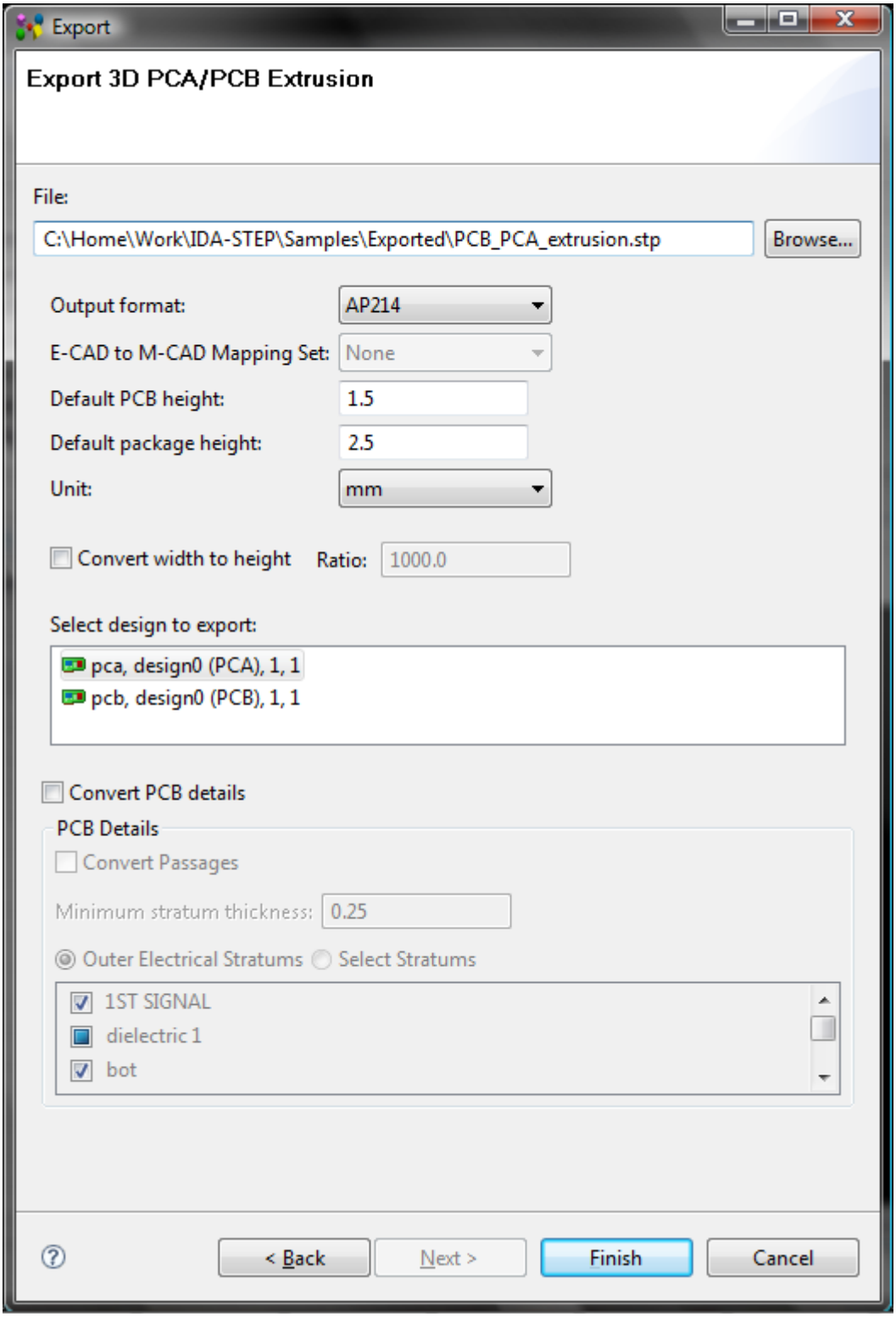

Figure 2. Export 3D PCA/PCB Extrusion wizard page

- Click **Browse** button next to the **File** field to locate the target STEP file in your file system.
- Select an applicable data output format AP214 or 203ed2 by selecting a value in the **Output format** drop down list.
- To generate 3D shape for PCA using "real" (not simply extruded) 3D package shapes, select which package library to use (if any) when generating PCA 3D shape in the **E-CAD to M-CAD Mapping Set** drop down list. How to manage the E-CAD to M-CAD Mapping library, refer clause E-CAD to M-CAD Mapping.
- Enter an applicable number value into the **Default PCB height** field - the PCB height that will be used for 3D shape generation if there is no data about PCB height in the design or the height data in the design is less than specified in this field..
- Enter an applicable number value into the **Default package height** field - the package height that will be used for 3D shape generation if there is no data about package height in

the design or the height data in the design is less than specified in this field.

- Select a unit in the **Unit** combo box - the unit for default PCB and package heights, also for PCB minimum stratum thickness.
- The **Convert width to height** option (see Figure 3) can be used when the design has special package shapes. Each shape consists of one or many closed curves with specific widths. The package 3D shapes are generated so, that each closed curve is extruded with a height, that is equal to it's width multiplied by the **Ratio**. Refer to the documentation for specific IDA-STEP electronics importers to see, if and how they support this feature (see Import Eagle Feature documentation if the **Import Eagle** feature is installed).

Figure 3. Convert width to height option and Ratio field

You can see what a difference between generated 3D shapes, when only **Default package height** is used for conversion (see Figure 4) and when the **Convert width to height** option is selected while exporting data (see Figure 5).

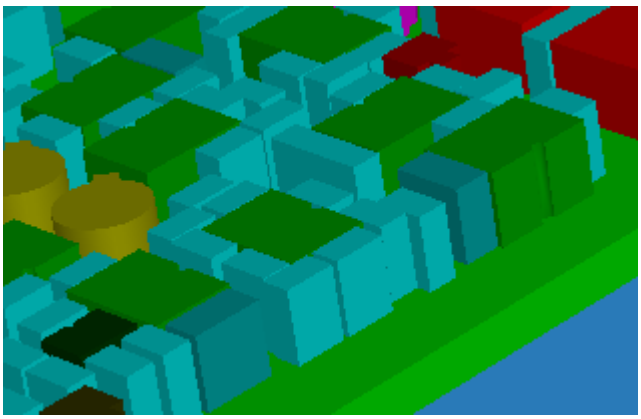

Figure 4. Generated 3D shapes with default PCB and default package heights

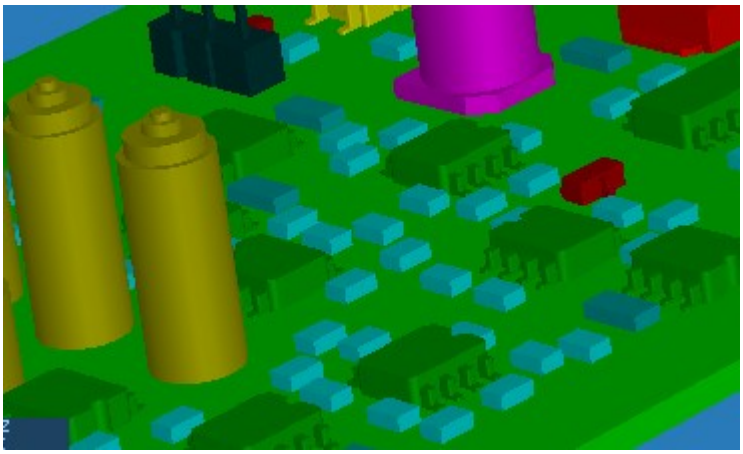

Figure 5. Generated 3D shapes with converted width to height

- Select an applicable design in the **Select design to export** list. You can select PCA or PCB designs (if any is present in the list).
    - If the PCA design is selected, then the user has a possibility to convert PCA and/or PCB designs (see Figure 6).

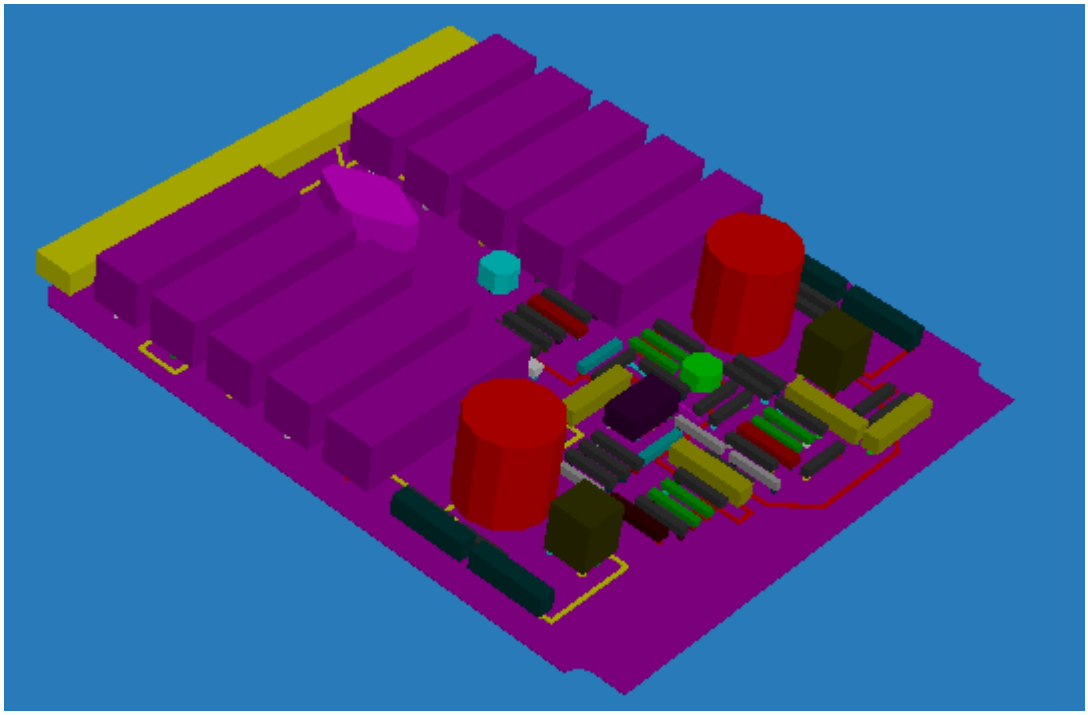

Figure 6. PCA and PCB 3D view

In Figure 2 it is selected the PCA design. You can see the enabled check box **Convert PCB details**. If this check box is not selected, than only PCA details should be converted (see Figure 7).

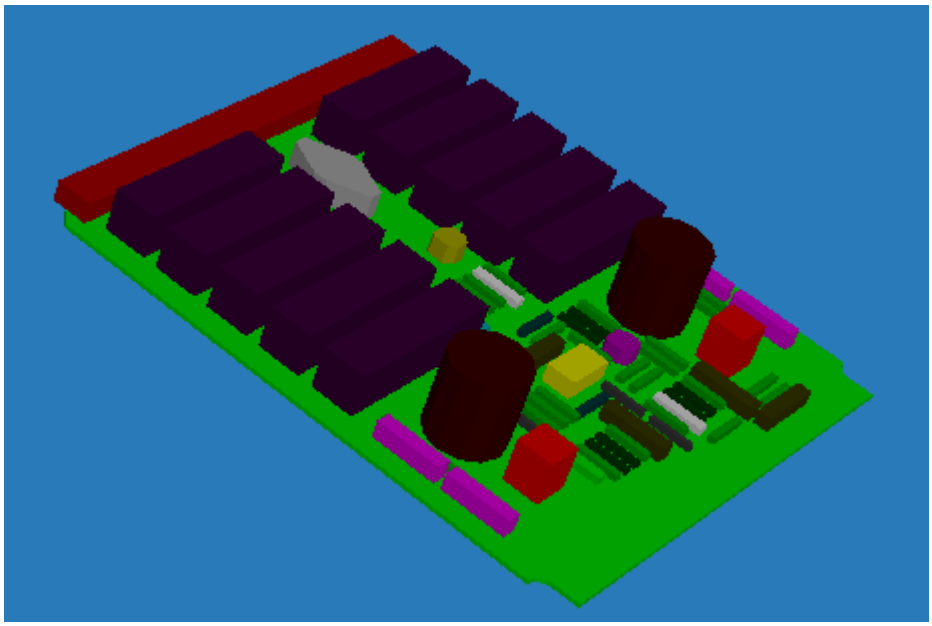

Figure 7. PCA design 3D view

- If PCB design is selected - only PCB data will be converted to 3D (see Figure 8):

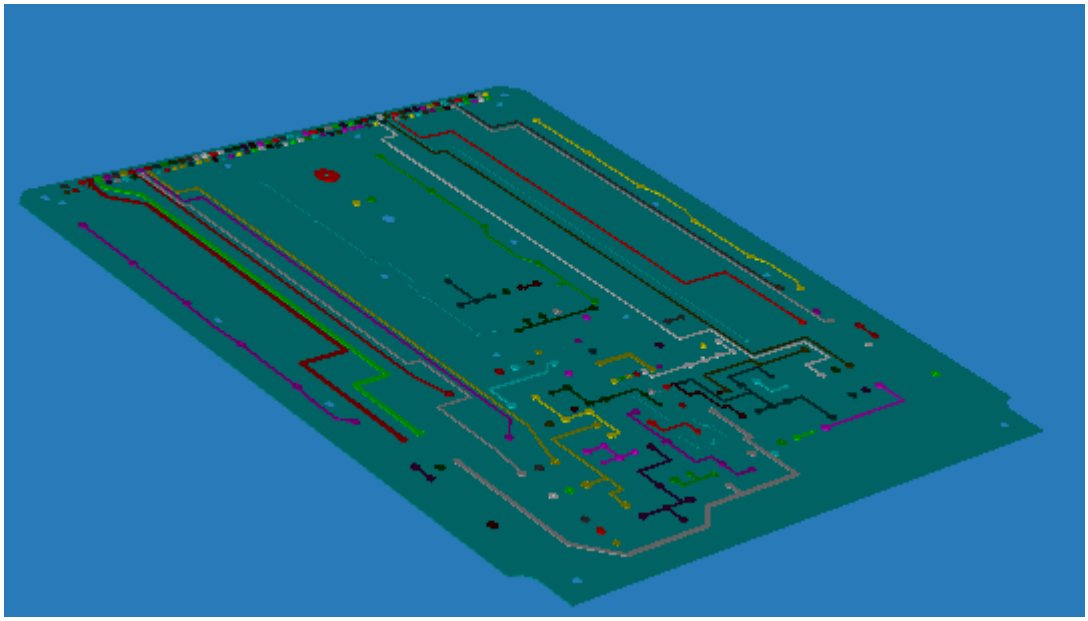

Figure 8. PCB design 3D view

- The **Convert PCB details** check box is selected automatically (see Figure 9). Options in the **PCB Details** layout are enabled for selection. You can select applicable PCB details to be converted:

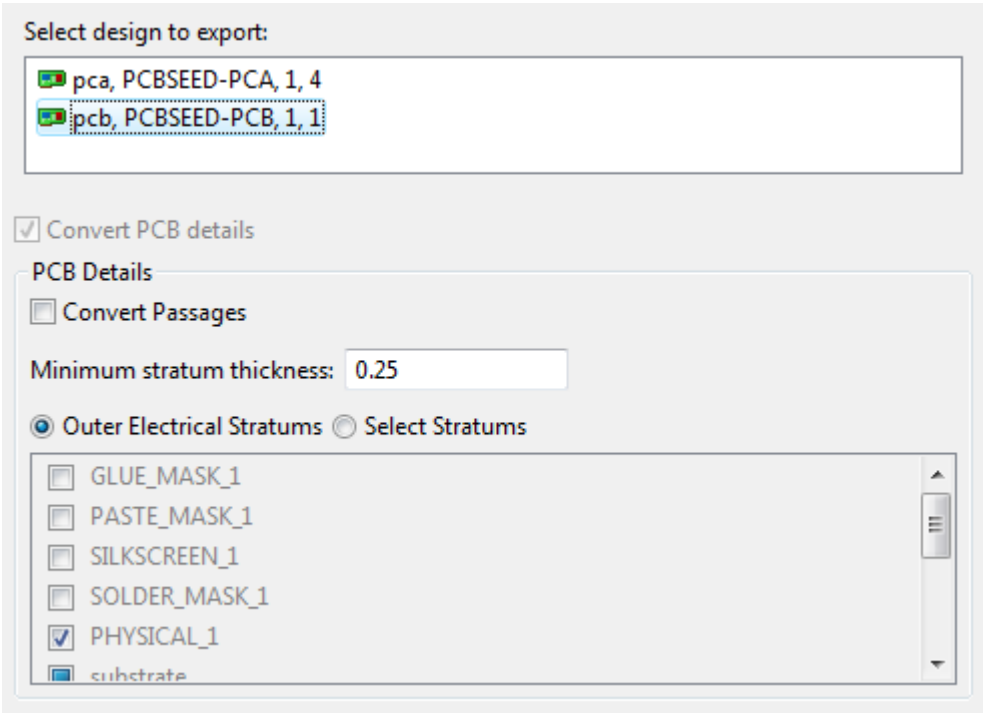

Figure 9. Convert PCB details

- Select **Convert Passages** check box to convert passages.
- Enter a number value of the minimum stratum thickness that will be used for 3D shape generation into the **Minimum stratum thickness** field. This value is used, if no minimum stratum thickness data is present in the selected design or the value in the design is smaller then the currently entered value.
- Select applicable stratums to be converted:
  - **Outer Electrical stratums** - the stratums are selected automatically: outer stratums - with details (☑ check box icon) and stratums between them - without details (▣ check box icon).
  - **Select Stratum** option allows to select all available stratums and their representation (see Figure 10). Select this radio button in order

to add more needed stratums to be converted:

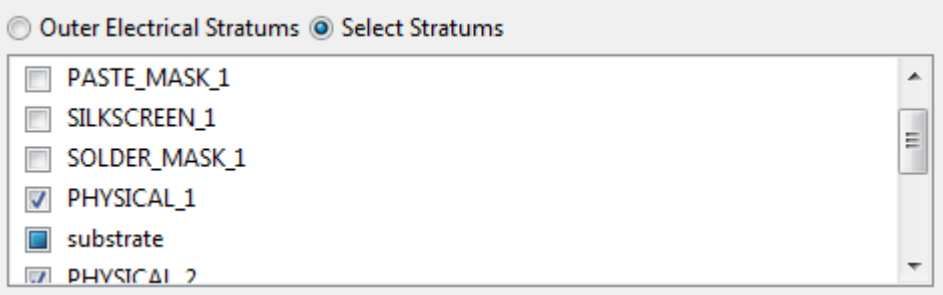

Figure 10. Select Stratums

- The list of available stratums is enabled for selection. The representation of each stratum can be selected by clicking on the check box in front of the particular stratum:
  - "unselected" check box icon - the stratum is not selected for converting.
  - "partially selected"check box icon - a stratum without details (see Figure 11).

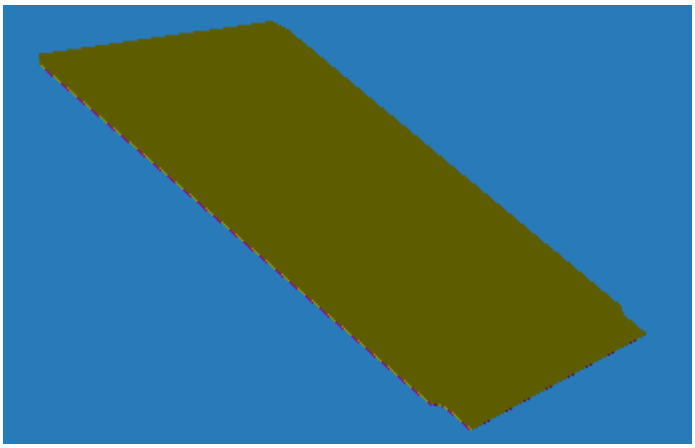

Figure 11. Converted stratum converted without details

  - "fully selected" check box icon - a stratum with details (see Figure 12 - the same stratum as in Figure 11).

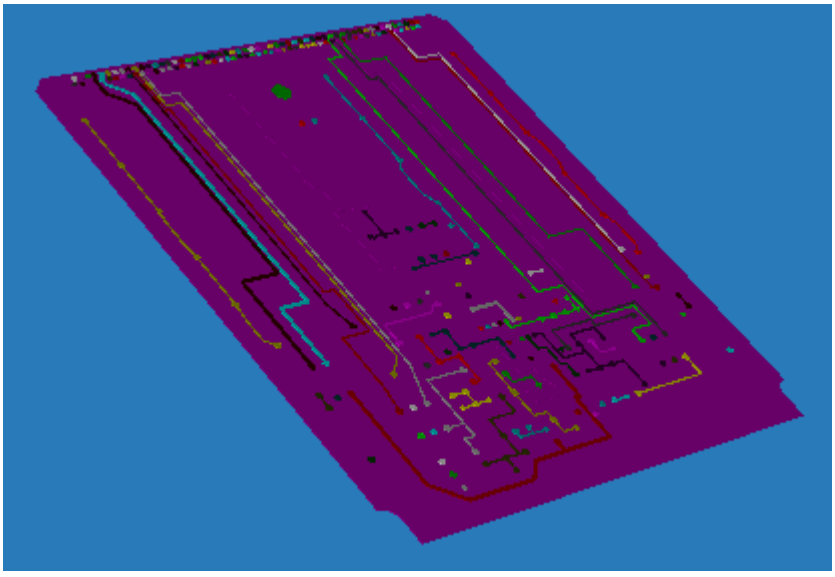

Figure 12. Stratum converted with details

- Click **Finish** to confirm your selections and start export. After the export is completed, the wizard page is closed automatically. The specified target STEP file appears in the specified location.

To top

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6.4.4. Exporting STEP (ISO 10303 - part 21)

STEP-File is the most widely used data exchange form of STEP (Standard for the Exchange of Product model data). Due to its ASCII structure it is easy to read with typically one instance per line. The format of a STEP-File is defined in ISO 10303-21 *Clear Text Encoding of the Exchange Structure*.

ISO 10303-21 defines the encoding mechanism on how to represent data according to a given EXPRESS schema, but not the EXPRESS schema itself. STEP-File are also called *p21-File* and *STEP Physical File*. The file extensions *.stp* and *.step* indicate that the file contain data conforming to STEP Application Protocols. For more information on STEP file reference http://en.wikipedia.org/wiki/ISO_10303-21.

Follow the routine to export STEP data:

- Applicable STEP data have to be loaded in IDA-STEP.
- Select in the main menu **File -> Export**. The **Export** wizard appears on the screen:
  - Select **Other ->** **STEP (ISO 10303 - part 21)** item and press **Next** button.
  - The **Export STEP** wizard dialogue window appears on the screen (see Figure 1):

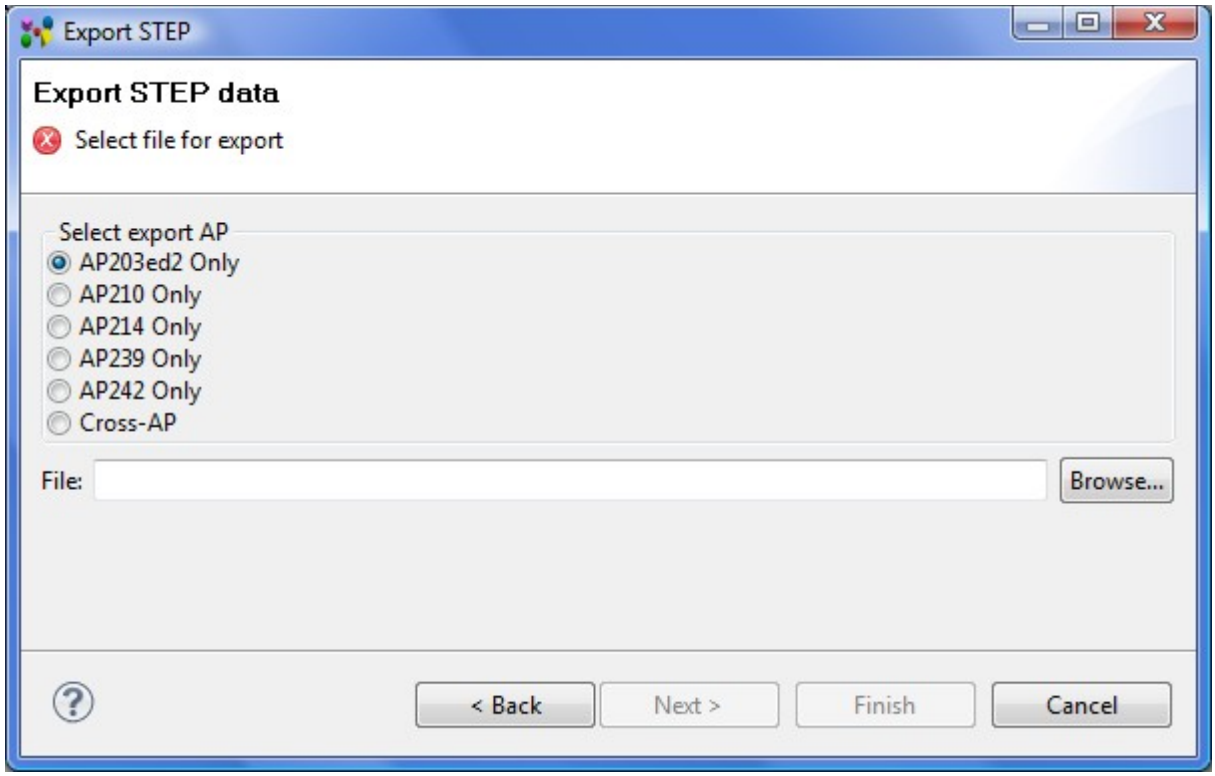

Figure 1. Export STEP dialogue window

- Select an applicable Application Protocol (AP) in the **Select export AP** layout:
  - **AP203 ed2 Only** - export STEP data according AP203 ed2 (Configuration controlled 3D designs of mechanical parts and assemblies) format;
  - **AP210 Only** - export STEP data according AP210 (Electronic assembly,

interconnection, and packaging design) format;
    - **AP214 Only** - export STEP data according AP214 (Core data for automotive mechanical design processes) format;
    - **AP239 Only** - export STEP data according AP239 (Product life cycle support) format;
    - **AP242 Only** - export STEP data according AP242 (Managed Model-Based 3D Engineering) format;
    - **Cross-AP** - export data as it is, independent of a particular AP.
- Select a destination file by using **Browse** button next to the **File** field.
- Press **Finish** button.

To top

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6.4.5. Exporting to CSV

The **Export to CVS** tool generates the CSV (Comma Separated Values) file, which is a format supported by most applications (e.g. Microsoft Excel).
Follow the routine to export the currently opened STEP file data to a CSV file:

- A particular data have to be opened/imported into IDA-STEP (see Figure 1).

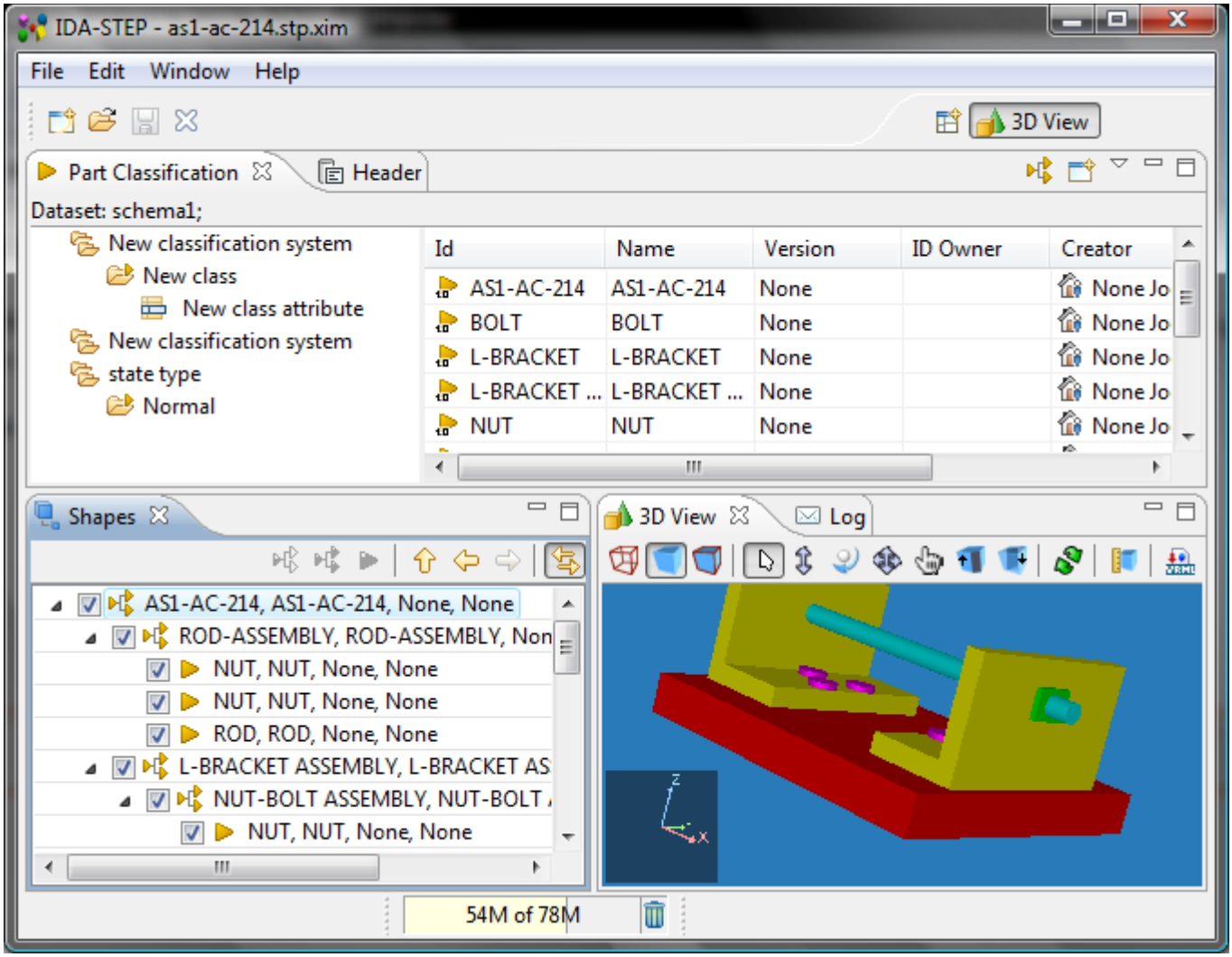

Figure 1. STEP file with data opened in IDA-STEP

- Select in the main menu **File -> Export**. The **Export** wizard appears on the screen.
- Select **Text ->** **CSV (Comma Separated Values)** item in the list.
- The **Export view** to set export settings and select which view to export appears on the screen (see Figure 2):

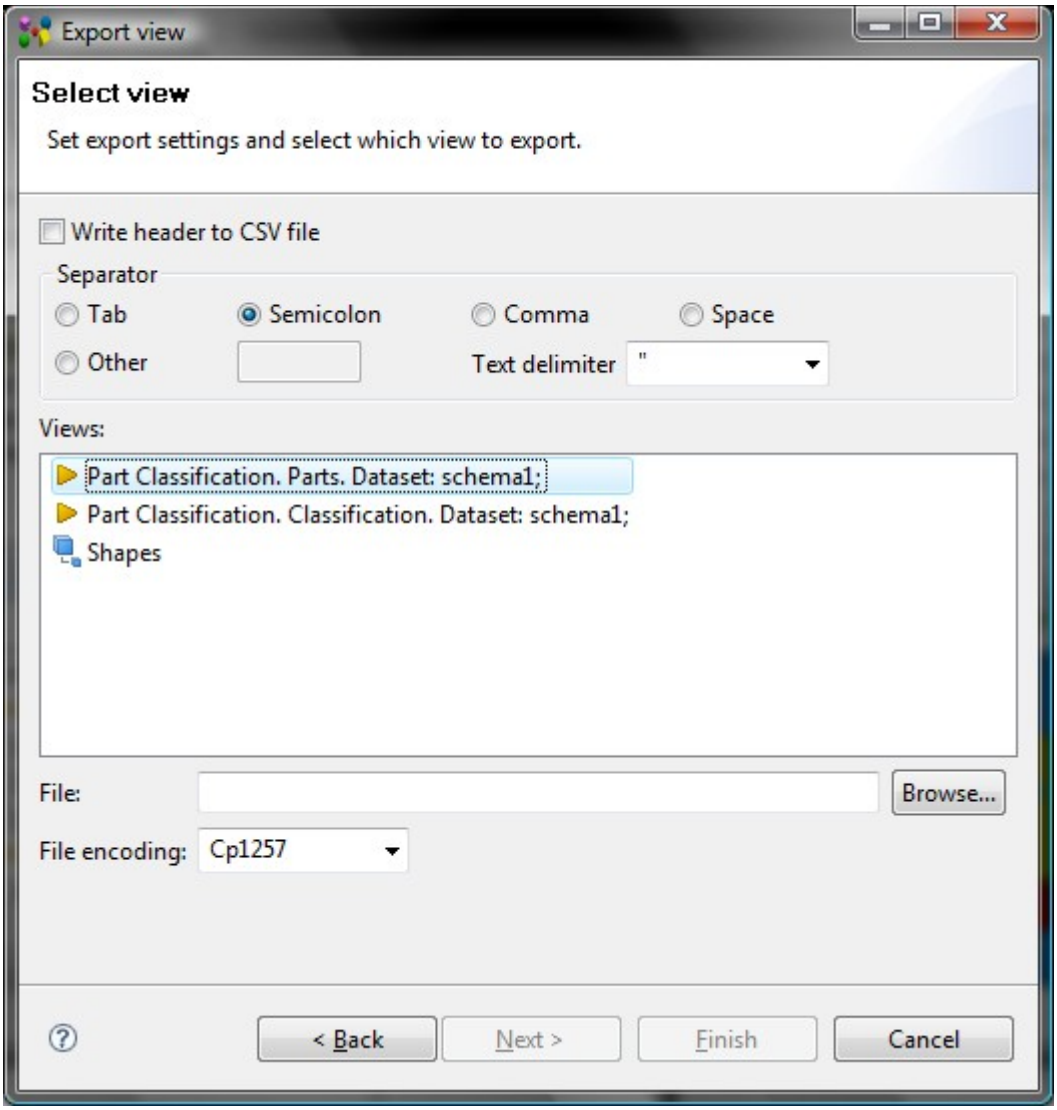

Figure 2. Export to CSV wizard window

- Select the **Write header to CSV file** check box to add a header for the table to be generated (see Figure 3).

Id	Name	Version	ID Owner	Creator	Security
AS1	AS1	None		None Non	unclassified
BOLT	BOLT	None		None Non	unclassified
L-BRACKE	L-BRACKE	None		None Non	unclassified
L_BRACKE	L_BRACKE	None		None Non	unclassified
NUT	NUT	None		None Non	unclassified

Figure 3. Generated CSV file table with header

- Select an applicable separator character in the **Separator** layout by selecting a particular radio button:
  - **Tab** - the text in the generated table should be separated by tab character;
  - **Semicolon** - the text in the generated table should be separated by column character (see Figure 4);

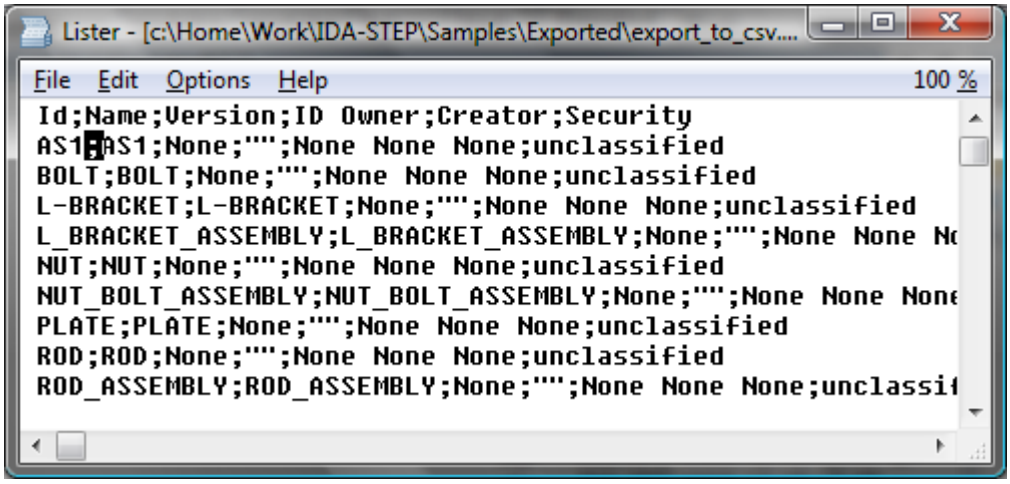

Figure 4. The text separator semicolon (';')

  - **Comma** - the text in the generated table should be separated by comma character;
  - **Space** - the text in the generated table should be separated by space character;
  - **Other** - if this radio button is selected, enter an applicable character into the text field;
  - **Text delimiter** - select an applicable value from the drop down list.
- Select one of available views, which data should be exported to CSV file in the **Views** layout.
- Click **Browse** button next to the **File** field to locate the target CSV file in your file system.
- You can select an applicable **File encoding** value in the drop down list.
- Press **Finish** button to confirm your selections and start export. After the export is completed, the wizard page is closed automatically. The specified target CSV file appears in the specified location.

To top

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6.4.6. Exporting to STL

The **Export to STL** tool generates binary *.stl* (stereolitography) file. STL files describe only the surface geometry of a three dimensional object without any representation of colour, texture or other common CAD model attributes.
Follow the routine to export the currently opened STEP file data to a *STL* file:

- The STEP file with an applicable 3D model data has to be opened in IDA-STEP (see Figure 1).

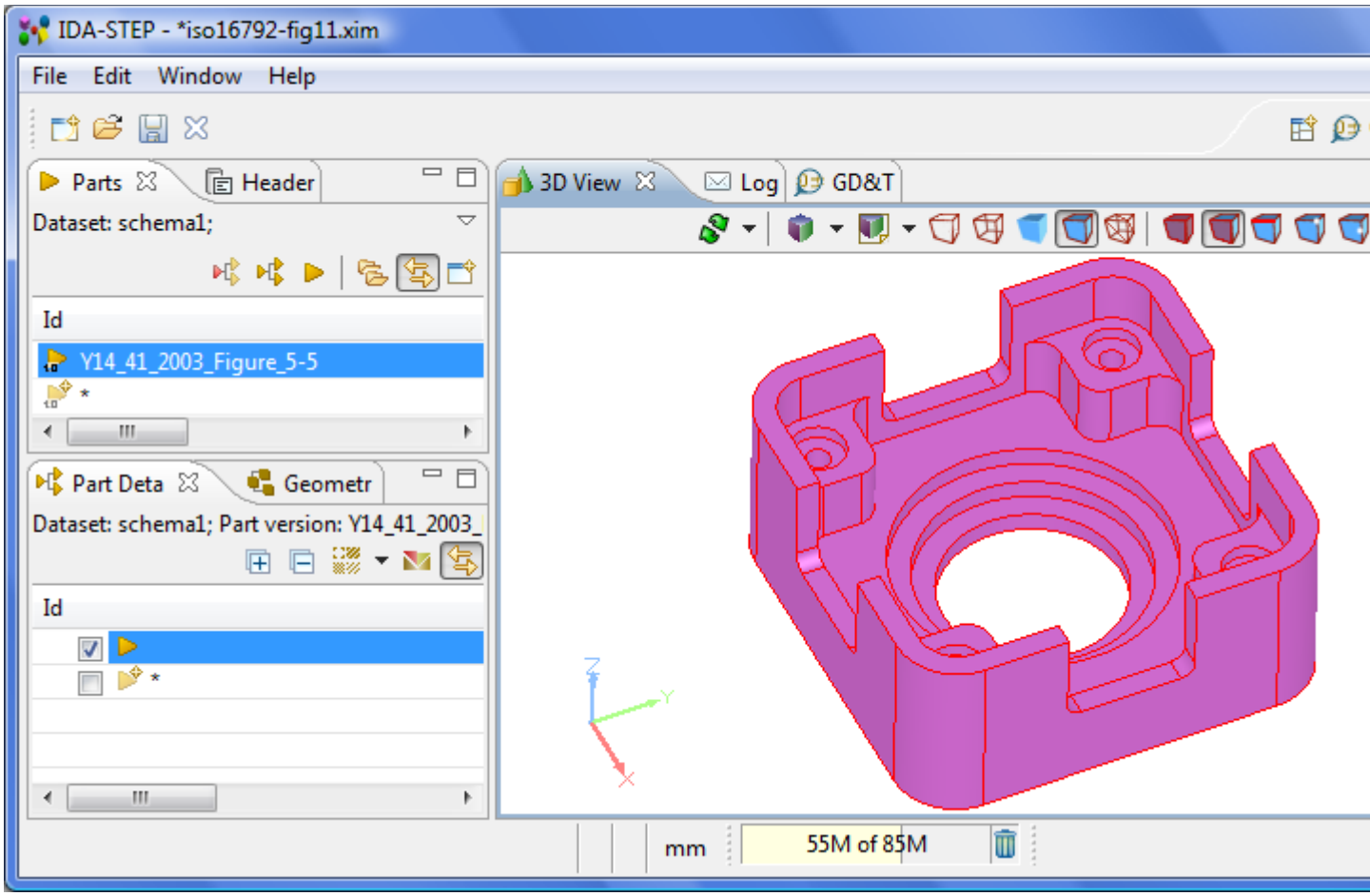

Figure 1. STEP file with 3D model data

- Select in the main menu **File -> Export**. The **Export** wizard appears on the screen:
  - Select **Other -> Export STL** item.
  - The **Export STL** wizard page appears to locate the file to be created (see Figure 2):

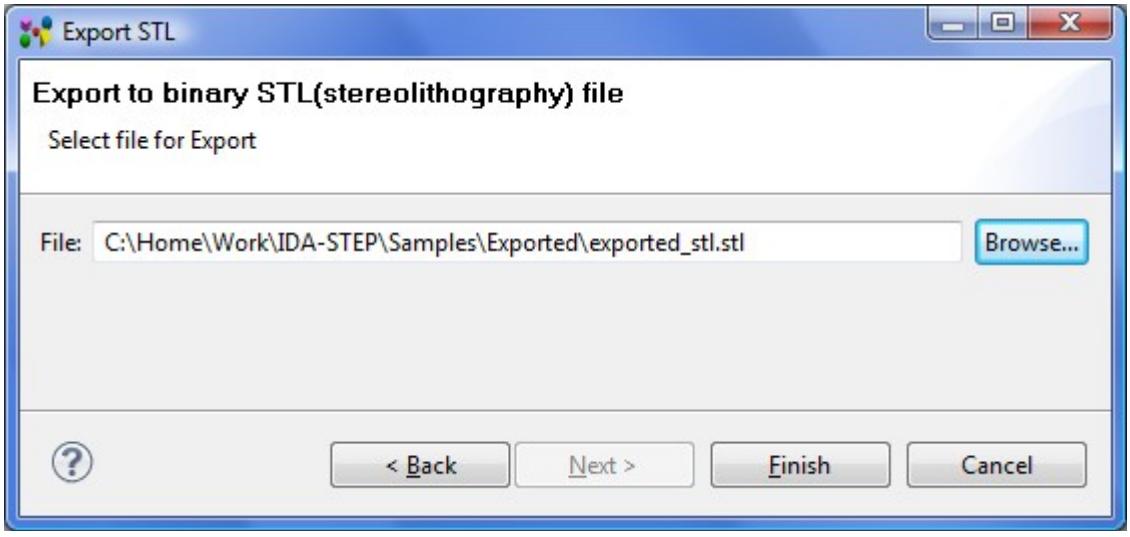

Figure 2. Export STL wizard page to locate stl file

  - Click **Browse** button next to the **File** field to locate the *STL* file in your file system. After the export is finished, the specified *STL* file appears in the selected location.
  - Click **Finish** button to confirm your selection and start export. After the export is completed the wizard page is closed automatically. The created *STL* file is available in the specified location.

To top

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6.4.7. Exporting to VRML

The **Export to VRML** tool generates *.wrl* file (virtual world created in VRML (Virtual Reality Modeling Language-97) version 2.0), which can be navigated in three dimensions, contains coordinates and colours that define each object and shape; also includes the viewpoint coordinates for the initial view of the 3D scene.
Follow the routine to export the currently opened STEP file data to a *WRL* file:

- The STEP file with an applicable 3D model data has to be opened in IDA-STEP (see Figure 1).

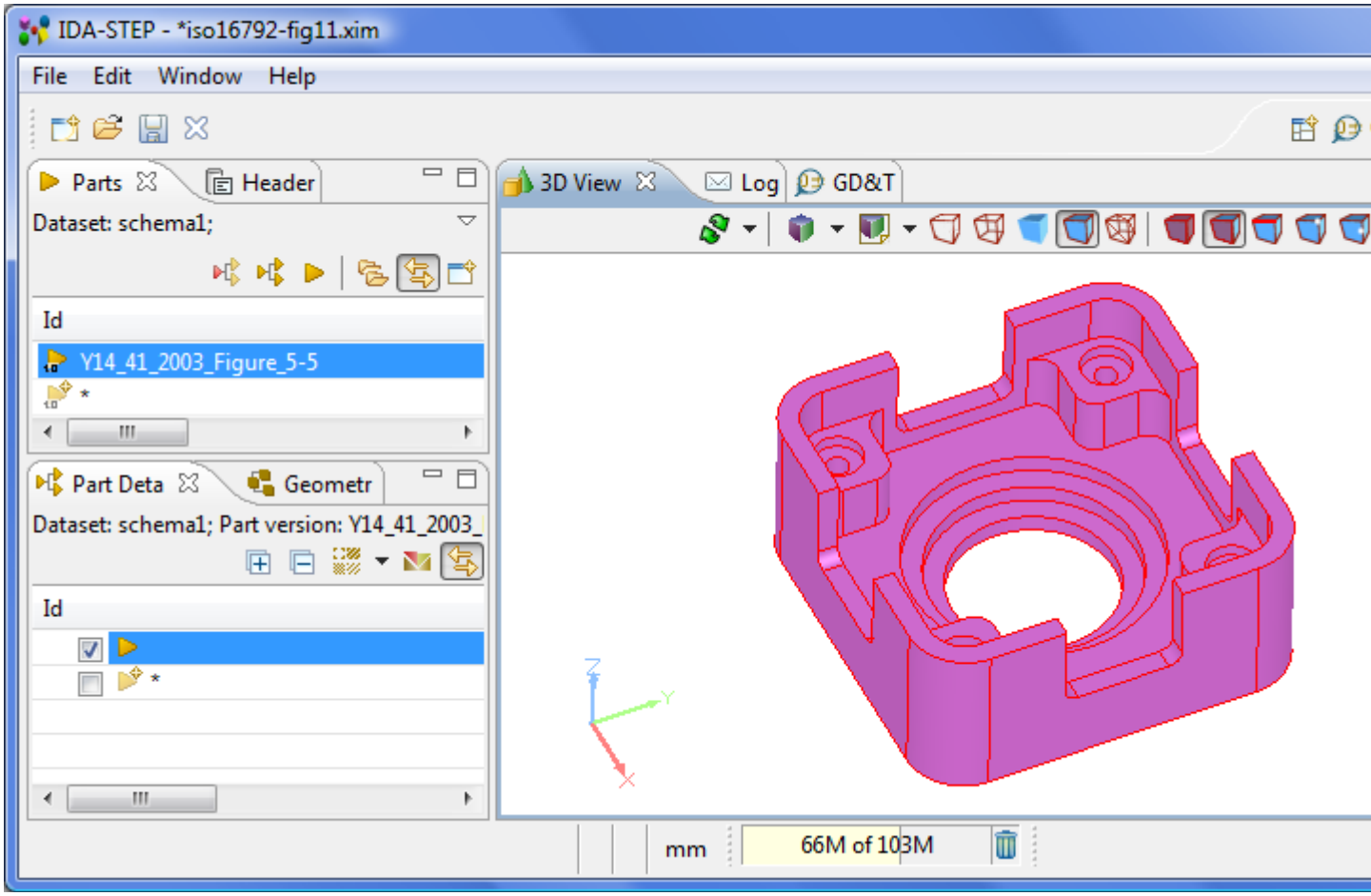

Figure 1. STEP file with 3D model data

- Select in the main menu **File -> Export**. The **Export** wizard appears on the screen:
  - Select **Other -> Export VRML** item.
  - The **Export VRML** wizard page appears to locate the file to be created (see Figure 2):

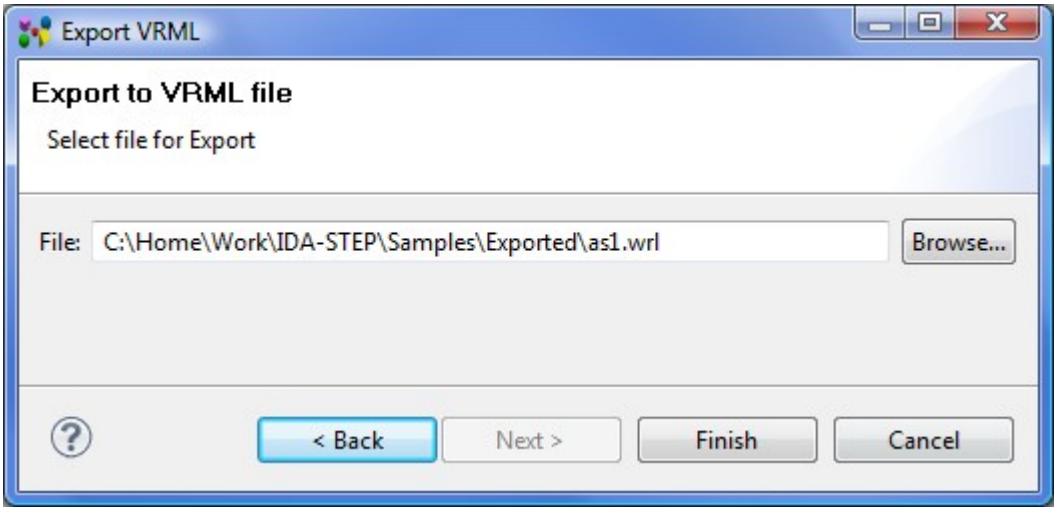

Figure 2. Export VRML wizard to locate wrl file

- Click **Browse** button next to the **File** field to locate the *WRL* file in your file system. After the export is finished, the specified *WRL* file appears in the selected location.
- Click **Finish** button to confirm your selection and start export. After the export is completed the wizard page is closed automatically. The created *WRL* file is available in the specified location.

To top

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6.4.8. Exporting CircuitCAM

Computer Aided Manufacturing (**CAM**) files, also known as *Gerbers* or *Gerber files*, are the standard format for describing your board design.

Follow the routine to export the PCB design data to *CAM* file:

- The STEP file with the desired PCB data has to be opened in the IDA-STEP (see Figure 1).

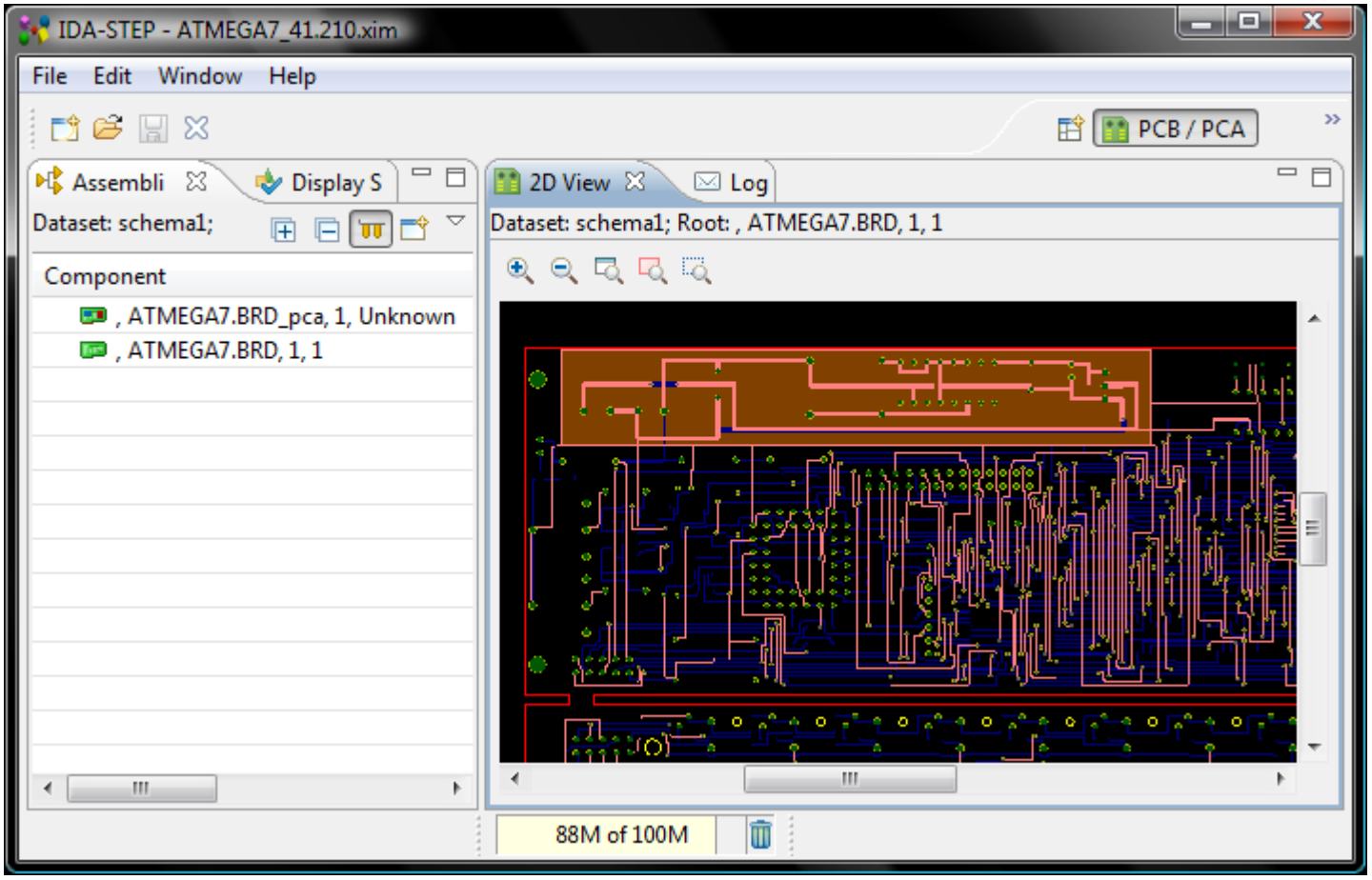

Figure 1. STEP file with PCB data

- Select in the main menu **File -> Export**. The **Export** wizard appears on the screen:
  - Select **Other ->** **Export CircuitCAM** item.
  - The **Export CircuitCAM** wizard page appears to make selections (see Figure 2):

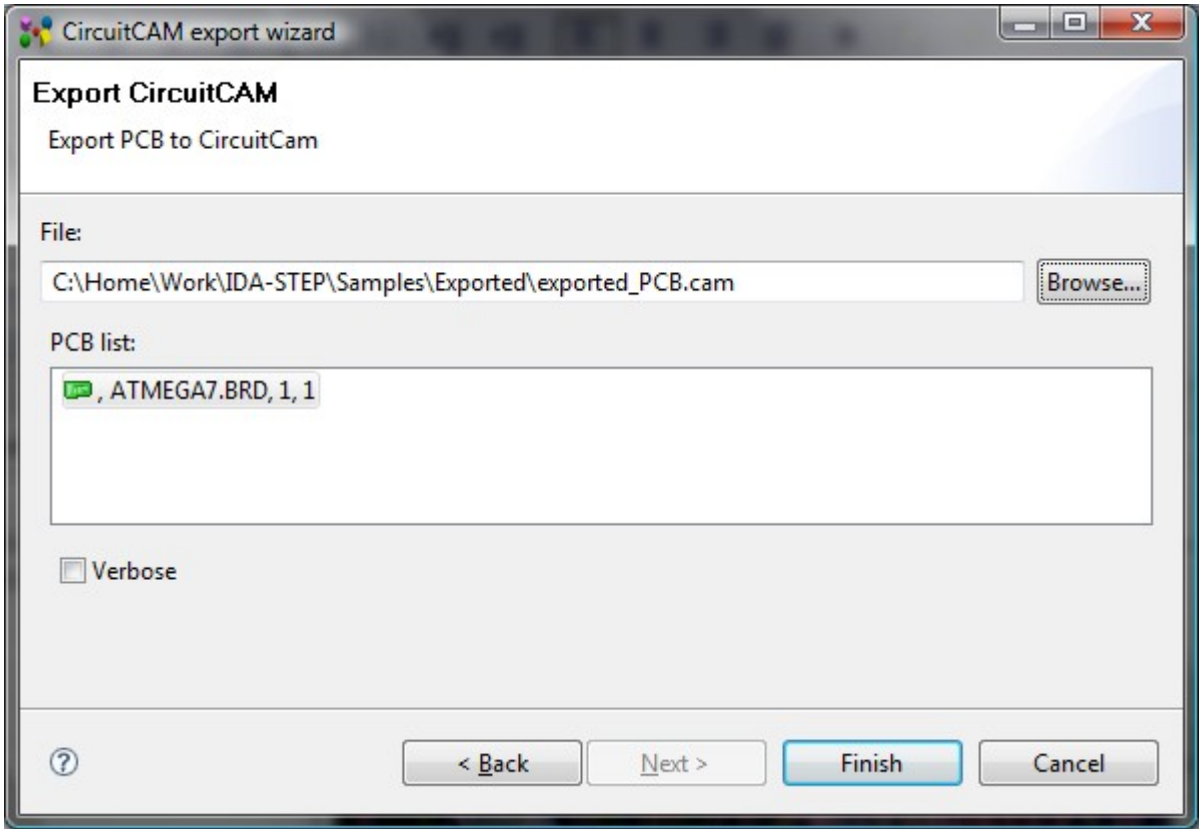

Figure 2. Export CircuitCAM wizard page

- Click **Browse** button next to the **File** field to locate the *CAM* file in your file

system. After the export is finished, the specified *CAM* file appears in the selected location together with the *LOG* file with the same name, containing general log information.

- Select a PCB in the **PCB list**.
- Select the **Verbose** check box to get more detailed information in the *LOG* file.
- Click **Finish** button to confirm your selections and start export. After the export is completed the wizard page is closed automatically. The created files are available in the specified location (see Figure 3).

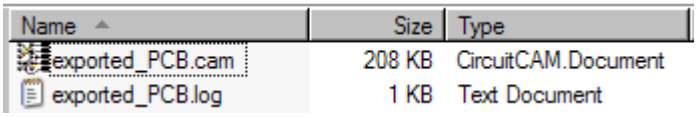

Figure 3. Created files in the specified folder

To top

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6.5.1. 3D Part Libraries

IDA-STEP presents possibility to set-up and maintain libraries of 3D parts using **Manage 3D Part Libraries** wizard (see Figure 1). These parts could be used by various IDA-STEP tools (e.g. PCA 3D shape generation tool (refer clause E-CAD to M-CAD Mapping) as a provider of package 3D shapes). To open the wizard page use the main menu bar **File -> Manage Libraries -> 3D Part Libraries...**.

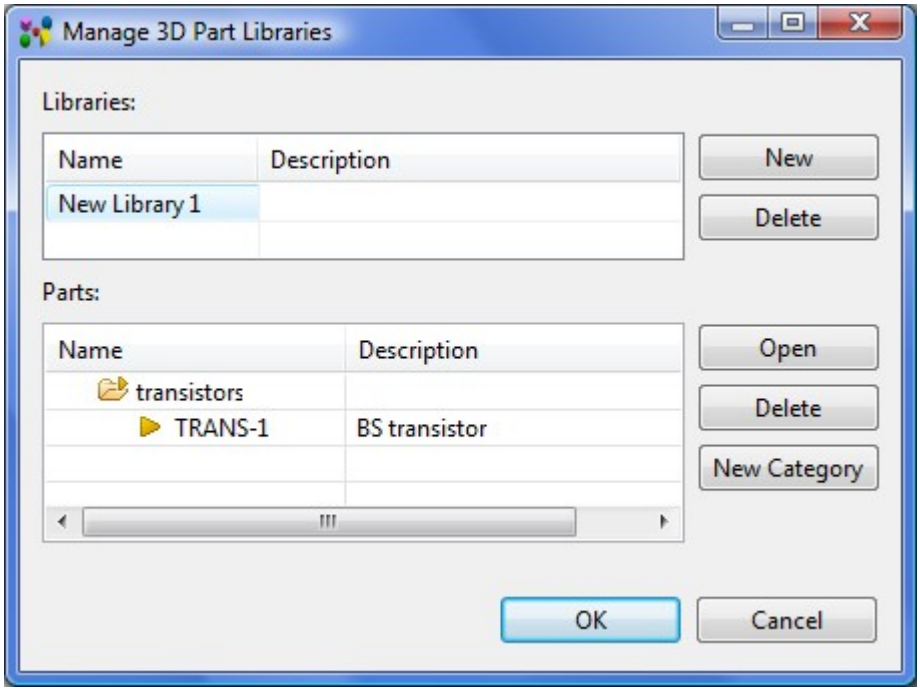

Figure 1. Manage 3D Part Libraries wizard page

The **Libraries** table presents the list of libraries with the specified name and description:

- To *view a content* of an applicable library - select the library entry in the table. The content of the selected library is presented in the **Parts** table.
- To *create* a new library - press the **New** button next to the **Libraries** table. After the button is released, a new entry with the default name *New Library <number>* appears in the table. To edit the name - double click on the **Name** cell and enter a new text. Additional information about the library can be entered into the **Description** column - double click on the column cell and enter an applicable text. Press enter to quit the editing mode.

- To *remove* a redundant or an incorrect library entry - select a library entry in the table and press **Delete** button, located next to the table.

The **Parts** table presents the hierarchy of the selected library - 3D parts, which can be classified into particular categories. Each category and part are defined by a name and a description (optionally). See below, how to manipulate with data, presented in the **Parts** table:

- To *open a 3D part data* in a new IDA-STEP window:
  - Select an applicable part in the **Parts** table.
  - Press the **Open** button next to the table. A new IDA-STEP window appears on the screen with loaded data of the selected part.
- To *create a new category* - press the **New Category** button next to the **Parts** table. The **New Category** dialogue window appears on the screen (see Figure 2):

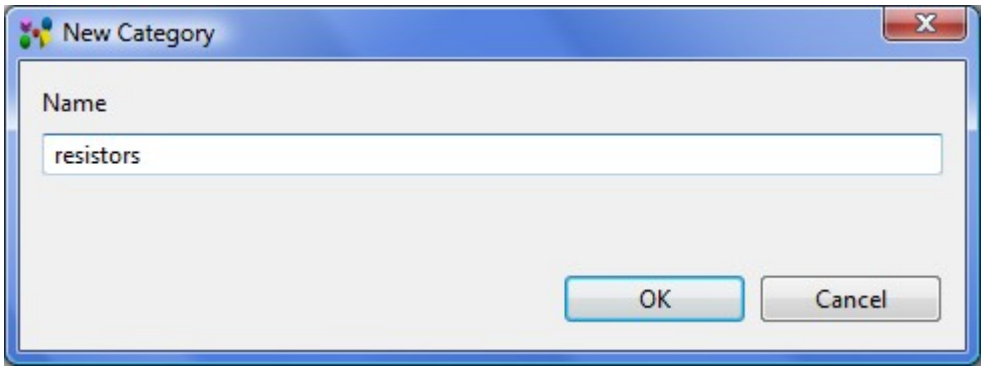

Figure 2. New Category dialogue window with specified category name

  - Enter a name, by which the category is known into the **Name** field.
  - Press the **OK** button. The created category appears in the **Parts** table.
    Note. In case the created category appears under the another category node, you can move it to the upper level - press with the mouse on the category entry, hold and drag it to the upper level.
- To *delete* a redundant or an incorrect category - select a category entry in the table list and press the **Delete** button next to the table. **Note.** In case the deleted category was not empty, related parts are moved to the top level.
- To *move parts to a category* - select a part, press with a mouse button hold and drag to an applicable category.

After all changes are made in the Manage 3D Libraries wizard page, press the **OK** button to confirm changes and close the window. **Note.** Press the **Cancel** button to close the window without saving changes.

How to add parts to the library, see Add 3D Part to Library.

To top

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6.5.2. E-CAD to M-CAD Mapping

To generate 3D shape for PCA using "real" (not simply extruded) 3D package shapes we need to have a 3D parts library (see 3D Part Libraries) and package library.

IDA-STEP present the **Manage E-CAD to M-CAD Mapping** wizard, which guides in the population of

package libraries (see Figure 1).

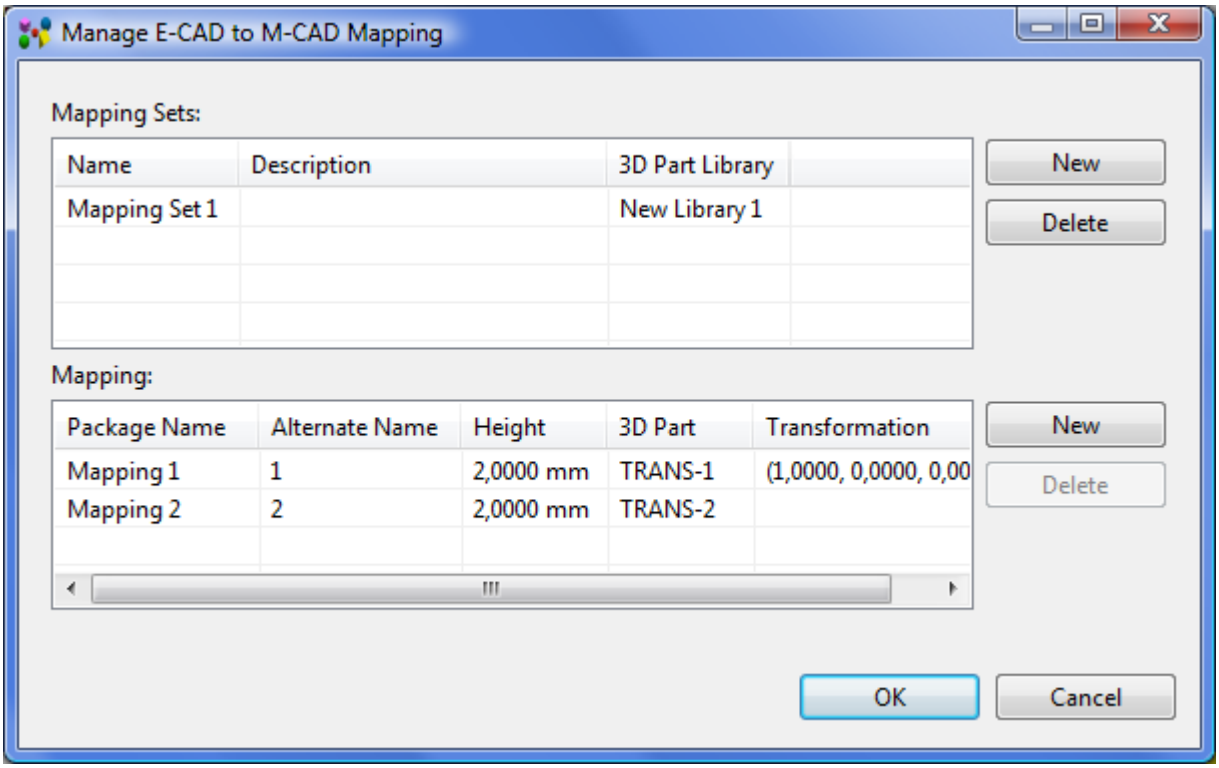

Figure 1. Manage E-CAD to M-CAD Mapping wizard page

The **Mapping Sets** table presents a list of package libraries (mapping sets), which are defined by **Name**, **Description** and associated **3D Part Library**:

- To *view a content* of a mapping set - click with a mouse button on an applicable mapping set entry in the **Mapping Sets** table. The content of the selected mapping set is displayed in the **Mapping** table.
- To *create a new package library*:
  - Press on the **New** button next to the table. A *New Mapping Set <number>* entry appears in the **Name** column. You can rename the created library name by double clicking on the **Name** column cell and entering a new text.
  - You can enter additional information about the created mapping set into the **Description** cell (double click and enter an applicable text).
  - Define 3D part library, which parts should be used for mapping:
    - Double click on the **3D Part Library** cell and press the appeared look up button.
    - Select an applicable 3D part library in the Select 3D Part Library wizard page (see Figure 2). You can filter out the particular library by entering the text into the filter field. Libraries, which names are close to the entered text, are displayed in the list. After the selection is done, press the **OK** button to confirm your choice and close the window. The selected 3D part library appears in the **3D Part Library** column cell.

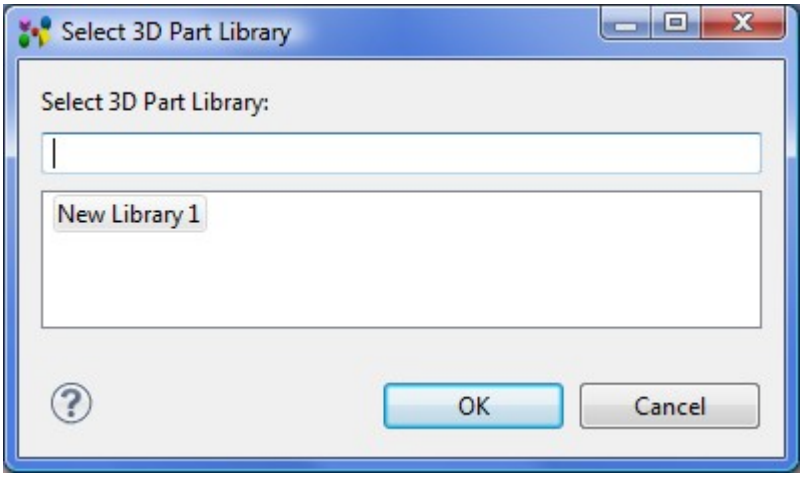

Figure 2. Select 3D Part Library wizard page

- To *remove* an incorrect or redundant mapping set - select a mapping set in the table and press the **Delete** button next to the table.

The **Mapping** table presents a list of packages (mappings), assigned for the selected package library (mapping set). The package object in package library should contain following attributes:

1. **Package name** - name, by which the package is known.
2. **Alternate name** - mandatory and unique identification within the library.
3. **Height** - the length measure with unit specifying the height to be used to generate extruded package shape, if 3D shape is not provided;
4. **3D shape** - the reference to a 3D part in 3D parts library to be used for this package in generated PCA 3D shape.
5. **Transformation** - transformation data of the 3D part.

To *create a new package object* in the selected library:

- Press the **New** button next to the **Mappings** table. The *New Mapping <number>* entry appears in the **Package Name** column. You can change the created package name by double clicking on the **Package Name** column cell and entering a new text.
- Add a text into the **Alternate Nam**e column cell to define a unique identifier for the package.
- Specify the height to be used to generate extruded package shape. Double click on the **Height** column cell. Enter the value into the text field, and select the unit in the drop down list, which appears next to the text field.
- Select a 3D part in 3D parts library to be used for this package in generated PCA 3D shape. Double click on the 3D Part column cell. Press on the look up button. The **Select 3D Part** appears on the screen (see Figure 3):
    - Expand branches (if any) and select an applicable part.
    - Press the **OK** button to confirm your selection and close the window. The selected 3D part appears in the **3D Part** column cell.

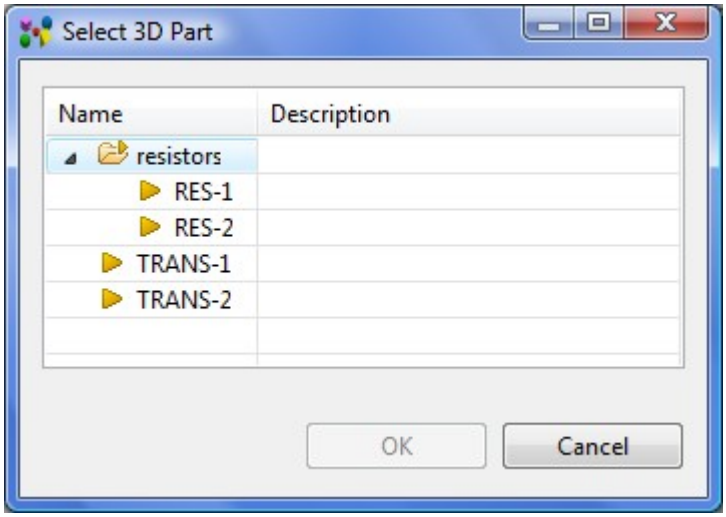

Figure 3. Select 3D Part

- Enter transformation data. Double click on the Transformation column cell. The 3D transformation editor wizard page appears on the screen (see Figure 4):
  - Enter value for the translation along X axis into **X** field;
  - Enter value for the translation along Y axis into **Y** field;
  - Enter value for the translation along Z axis into **Z** field;
  - **Units** read only field presents the measurement units, used for the translation.
  - Enter value for the rotation around X axis into **α** field;
  - Enter value for the rotation around Y axis into **β** field;
  - Enter value for the rotation around Z axis into **γ** field.
  - Press **OK** button to confirm your changes and close the window.
    Note. Units in the column are converted to the units, defined in the **Quantity and Units** preferences.

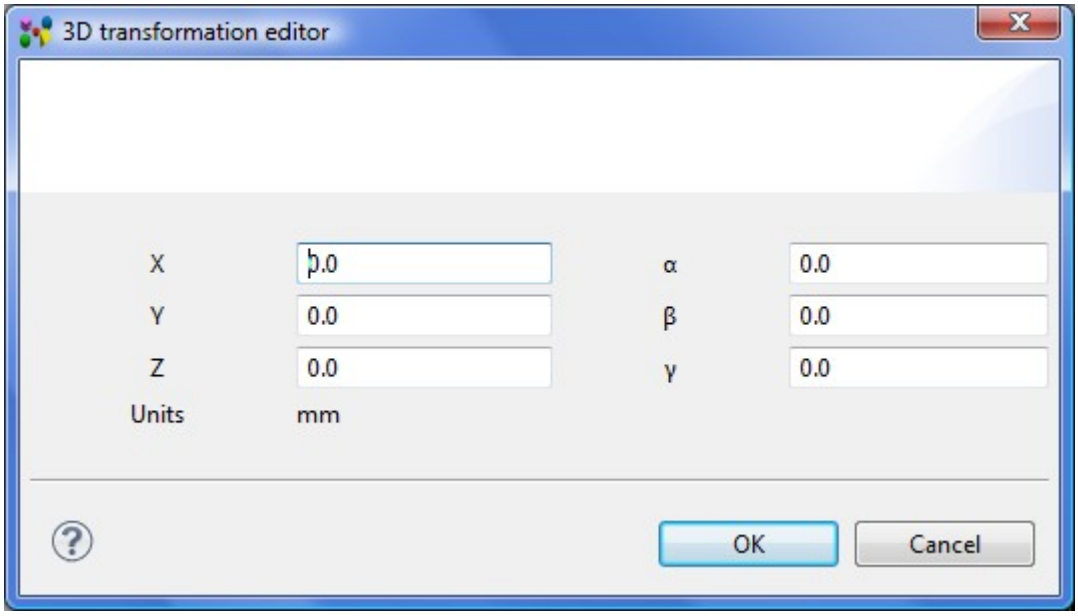

Figure 4. 3D transformation editor wizard page

- Press the **OK** button after all parameters are set for the package and close the the **Manage E-CAD to M-CAD Mapping** wizard window.
  Now user can select a package library when generating PCA 3D shape in the **Export 3D PCA/PCB Extrusion** wizard.

To *remove* an incorrect or redundant package from package library, select an applicable package in the **Mappings** table and press the **Delete** button next to the table.

To top

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6.6. Language

This feature is under construction.

There is a possibility to change the default language for data in IDA-STEP. **Note.** The default language can be selected in the preferences page (refer Clause Preferences -> General -> Language).
Follow the routine to switch the language:

- Select in the main menu bar **File -> Language**. The **Language** dialogue window appears on the screen (see Figure 1). The currently applied language is displayed next to the label **Current language**.

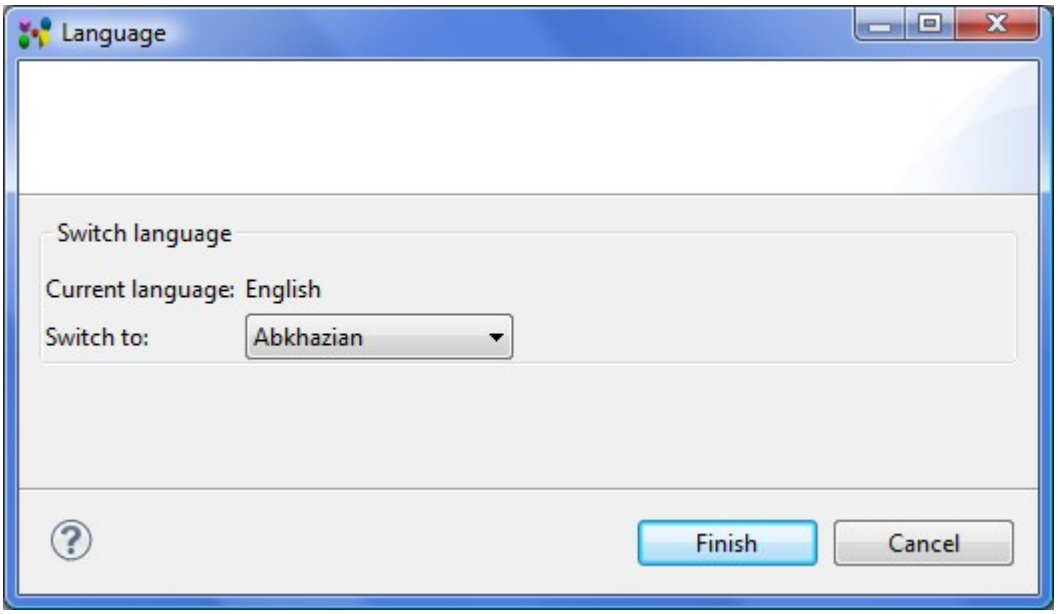

Figure 1. Language dialogue window

- Select an applicable language in the **Switch to** combo box drop down list.
- Press **Finish** button to confirm your sleection.

To top

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7. Installed Features

For general information about installed features see the Overview page. Under this clause you will find introductions of all installed features.

The following features are currently installed:

- IDA-STEP Framework
- Layer Stack Editor
- GD&T Editor

- Import CR-5000
- Viewer Basic
- Viewer Pro 3D
- Import Allegro
- Import Board Station
- Import CADIF
- Import CircuitCAM
- Import EAGLE
- Import Expedition
- Import PADS
- Electronics

To top

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7.1. Electronics

IDA-STEP Electronics focuses on ISO 10303-210 "Electronic Assembly Interconnect and packaging Design" (AP210). While the basic viewer component allows to view all aspects of AP210, the Electronics component has extended capabilities:

- Exporting to CSV File;
- Printing;
- Exporting to CircuitCAM;
- Exporting to AP203: 2D PCA/PCB data Extruded to 3D;
- Dynamic columns.

To top

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7.2. GD&T Editor

The **GD&T (Geometric Dimensioning and Tolerancing) Editor** feature allows editing of of geometric dimensions and tolerances, assembly structures and other product information.
The following data editing is available in IDA-STEP with installed **GD&T Editor** feature:

- Basic PDM editing:
  - Parts view: creating (part version, new version, classification system, classification, class attribute), editing / copying and pasting data and etc..
  - Persons and Organizations view: creating new elements (organization, person and position), copying and pasting, editing person, organization, position, address data and etc.
  - Document view: creating, editing and removing of document data.
- Product structure editing:
  - Part Details view: adding new components, editing, copying and pasting, removing components.
- Geometry Details view (geometric model details and editing):
  - Construction Geometry Models: create model, style construction geometry elements.

    - Geometric Models: style geometric model elements (visibility, colour, transparency),
    - Style/Annotation Models: create new style/annotation models, style annotations and annotation occurrences (visibility, colour),
    - Presentation Models.
- GD&T editing:
    - shape aspects;
    - dimensions;
    - datum features, targets,
    - geometric tolerances and etc.
- Exporting STEP data.
- Saving of XIM files.

To top

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7.3. IDA-STEP Framework

IDA-STEP Framework feature handles the main aspects of the Graphical User Interface (GUI) and all aspects on registration, installation and update.

To top

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7.4. Import Allegro

Cadence® Allegro® PCB Editor is a PCB layout system from Cadence Design Systems for designing Printed Circuit Boards (PCB).

The **Import Allegro** feature allows to convert Allegro board files (*.brd) into IDA-STEP. This conversion process has two steps. The first step is to run a special `extracta` command inside Allegro PCB Editor to dump out ASCII files. In the second step these files are then imported into IDA-STEP. Import feature uses the following Allegro design files from indicated Allegro data folder:

- ASCII layers file,
- ASCII pads file,
- ASCII components file,
- ASCII geometry file,
- ASCII nets file.

This feature imports the following PCB and Printed Circuit Assembly (PCA) data:

- detailed layout data of the bare PCB (layers, tracks, pads, physical nets, thickness, keep-out areas, board-outline, etc.);
- PCA layout data (assembly components with transformation, networks);
- library information (parts, packages, lands, vias, etc.);
- technology (Surface Mount Device (SMD), through hole, etc.);
- full graphical information (2D).

User can choose whether the complete design PCA and PCB is imported, or Import PCA data only.

See Importing Allegro for details.

To top

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7.5. Import BoardStation

Board Station® is a PCB-CAD System from Mentor Graphics for designing Printed Circuit Boards (PCB).

The **Import Board Station** feature allows to convert Board Station design files into IDA-STEP. This conversion process has two steps. The first step is to get PCB design directory, which is created automatically after a particular project is created in Board Station and also generate geometry ASCII file (*.txt). In the second step the PCB directory content and geometry file are then imported into IDA-STEP.

This feature imports the following PCB and Printed Circuit Assembly (PCA) data:

- detailed layout data of the bare PCB (layers, tracks, pads, physical nets, thickness, keep-out areas, board-outline, etc.);
- PCA layout data (assembly components with transformation, networks);
- library information (parts, packages, lands, vias, etc.);
- technology (Surface Mount Device (SMD), through hole, etc.);
- full graphical information (2D).

See Importing Board Station for details.

To top

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7.6. Import CADIF

CADIF (*.paf) is an ASCII neutral file, which is used for storing PCB, PCA and other information. This file is used by ECAD systems CR-5000, VISULA and CADSTAR from Zuken.

The **Import CADIF** feature allows to import design stored in CADIF file into IDA-STEP. It requires only the CADIF (*.paf) file.

This feature imports the following PCB and Printed Circuit Assembly (PCA) data (it is possible to import PCA data only):

- detailed layout data of the bare PCB (layers, tracks, pads, physical nets, thickness, keep-out areas, board-outline, etc.);
- PCA layout data (assembly components with transformation, networks);
- library information (parts, packages, lands, vias, etc.);
- technology (Surface Mount Device (SMD), through hole, etc.);
- full graphical information (2D).

User can choose whether the complete design PCA and PCB is imported, or Import PCA data only.
See Importing CADIF for details

To top

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7.7. Import CircuitCam

LPKF CircuitCAM PCB is a CAM system from LKSoft / LPKF Laser & Electronics AG, which is used for designing Printed Circuit Boards (PCB).

LPKF CircuitCAM PCB software provides the user with a sophisticated CAM station. It can control all production data, modify the design, step and repeat the complete layout, perform different design rule checks, generate ground planes, input text using TrueType fonts, and generate routing data. LPKF CircuitCAM PCB is also a very efficient data converter, such as converting AutoCAD data to Gerber data.

The **Import CircuitCAM** feature allows to convert CircuitCAM files (*.cam) into IDA-STEP. This conversion process has two steps. The first step is to make an auxiliary XML design file. In the second step these files are then imported into IDA-STEP.

This feature imports the following PCB and Printed Circuit Assembly (PCA) data:

- detailed layout data of the bare PCB (layers, tracks, pads, physical nets, thickness, keep-out areas, board-outline, etc.);
- PCA layout data (assembly components with transformation, networks);
- library information (parts, packages, lands, vias, etc.);
- technology (Surface Mount Device (SMD), through hole, etc.);
- full graphical information (2D).

See Importing CircuitCAM for more details.

To top

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7.8. Import CR-5000 Feature

CR-5000 is Zuken's printed circuit board (PCB) and advanced packaging layout suite of customizable tools aimed at the Enterprise market.
The **Import CR-5000** feature allows to import CR-5000 design into IDA-STEP. It requires the following data from CR-5000:

- **PCB ASCII File** (*PCF* extension) - the ASCII file describing the contents of the PCB Database;
- **Footprint ASCII File** (*FTF* extension) - the ASCII file describing the contents of the Footprint Library.

This feature imports the following PCB and Printed Circuit Assembly (PCA) data (it is possible to import PCA data only):

- detailed layout data of the bare PCB (layers, tracks, pads, physical nets, thickness, keep-out areas, board-outline, etc.);
- PCA layout data (assembly components with transformation, networks);
- library information (parts, packages, lands, vias, etc.);
- technology (Surface Mount Device (SMD), through hole, etc.);
- full graphical information (2D).

User can choose whether the complete design PCA and PCB is imported, or Import PCA data only.
See Importing CR-5000 for more details.

To top

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7.9. Import EAGLE

EAGLE Layout Editor is a tool from CadSoft for designing Printed Circuit Boards (PCB).
The **Import EAGLE** feature allows to convert EAGLE board files (*.brd) into IDA-STEP. This conversion process has two steps. The first step is to run a special ULP program inside EAGLE to dump out the binary board file into an ASCII file and also to generate a design rule file (*.dru). In the second step these files are then imported into IDA-STEP.

This feature imports the following PCB and Printed Circuit Assembly (PCA) details:

- detailed layout data of the bare PCB (layers, tracks, pads, physical nets, thickness, keep-out areas, board-outline, etc.);
- PCA layout data (assembly components with transformation, networks);
- library information (parts, packages, lands, vias, etc.);
- technology (Surface Mount Device (SMD), through hole, etc.);
- full graphical information (2D).

User can choose whether the complete design PCA and PCB is imported, or Import PCA data only.
The EAGLE converter also supports packages with varying height information. In order to generate 3D package shapes it is needed to have data, created according to special rules in Eagle design:

1. Extrusion to 3D uses a line width to calculate a height of a geometrical shape and it is calculated this way:
   Height = line width of geometrical shape * ratio
   The default ratio is 1000 and can be changed in the Export 3D PCA/PCB Extrusion wizard.
2. Each geometrical shape should be closed.
3. Each package may contain many geometrical shapes with different line widths.
4. Each geometrical shape will be extruded from the package placement surface to it's height.
5. Geometrical shapes may intersect or be in other shapes. But take into account that smaller shapes put into higher ones will not be visible after extrusion.
6. Shapes that should be extruded to the top side from package placement surface must be drawn on 3D top layer, which by default is Layer 57.
   Shapes that should be extruded to bottom side from package placement surface must be drawn on 3D bottom layer, which by default is Layer 58.
   The default numbers of layers that contain 3D top and bottom shapes are configurable and stored in configuration file *Eagle-AP210.main.xml*. This file is located in IDA-STEP installation folder (*.../plugins/net.jsdai.xim.ap210.eagle/config*).
   A user can change numbers of layers in the <*packShapes/*> tag:
   <packShapes>

```
    <shape layer="57" purpose="3D top"/>
    <shape layer="58" purpose="3D bottom"/>
</packShapes>
```

See Importing EAGLE for details.

To top

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7.10. Import Expedition

Expedition PCB™ is a PCB-CAD System from Mentor Graphics for designing Printed Circuit Boards (PCB).
The **Import Expedition** feature allows to import design from Expedition PCB data files into IDA-STEP. It requires the Expedition PCB data folder with the following files, which can be find in one of the the Expedition PCB project folders:

- Cell.hkp,
- JobPrefs.hkp,
- Layout.hkp,
- NetClass.hkp,
- NetProps.hkp,
- Padstack.hkp,
- PDB.hkp.

This feature imports the following PCB and Printed Circuit Assembly (PCA) data (it is possible to import PCA data only):

- detailed layout data of the bare PCB (layers, tracks, pads, physical nets, thickness, keep-out areas, board-outline, etc.);
- PCA layout data (assembly components with transformation, networks);
- library information (parts, packages, lands, vias, etc.);
- technology (Surface Mount Device (SMD), through hole, etc.);
- full graphical information (2D).

See Importing Expedition for more details.

To top

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7.11. Import PADS

PADS® is a PCB-CAD System from Mentor Graphics for designing Printed Circuit Boards (PCB).
The Import PADS feature allows to import design from PADS data files (*.asc) into IDA-STEP. It requires the PADS ASCII data files (*.asc) to import data.

This feature imports the following PCB and Printed Circuit Assembly (PCA) data:

- detailed layout data of the bare PCB (layers, tracks, pads, physical nets, thickness, keep-out areas,

board-outline, etc.);
- PCA layout data (assembly components with transformation, networks);
- library information (parts, packages, lands, vias, etc.);
- technology (Surface Mount Device (SMD), through hole, etc.);
- full graphical information (2D).

See Importing PADS for more details.

To top

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7.12. Layer Stack Editor

The PCB Layer Stack Editor is a tool for the specification of PCB technologies independent of particular PCB designs. The tool supports a wide range of technologies, including the ones defined in **IPC-2226** *Sectional Design Standard for High Density Interconnect (HDI) Printed Boards*. The main advantages of using IDA-STEP PCB Layer Stack Editor are:

- Efficient and successful coordination among PCB designer, fabricator and the intermediate procurement process
- Precise, formal and standardized information avoids communication errors
- Enables formal checking of the given PCB technology information for completeness and consistency

In addition the results of the PCB Layer Stack Editor can be used to complete the output of PCB-CAD systems, either by associating it with the delivered GERBER files or by directly linking it with native design data.

Useful perspectives

PCB Layer Library and PCB Layer Stack.

Available views

Design Stack Models, Stratums, Passages, Stratums cross-section, Stratum Technologies, Passage Technologies.

To top

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7.13. Viewer Basic

The **IDA-SEP Viewer Basic** is a free feature of IDA-STEP providing the core viewing functionality for all kinds of STEP files (*.stp). This feature alone makes IDA-STEP a fully usable tool for viewing PDM, CAD and other data, available as STEP (internation standard ISO 10303) files. STEP file format is supported by the major PDM, mechanical CAD and other systems.
The **Viewer Basic** feature includes the following functionality:

- opening STEP, XIM files;
- viewing STEP file content. Available views:
    - Datasets,
    - Header,
    - Log View,
    - 2D View,
    - 3D View,
    - Display Settings,
    - Part Details,
    - Design Stack Models,
    - Documents,
    - Parts,
    - Passage Technologies,
    - Passages,
    - Persons and Organizations,
    - Stratum Technologies,
    - Stratums,
    - Stratums Cross-section,
    - Tasks,
    - Zonal Breakdown.
- manipulating with perspectives: opening, creating, switching, customizing.

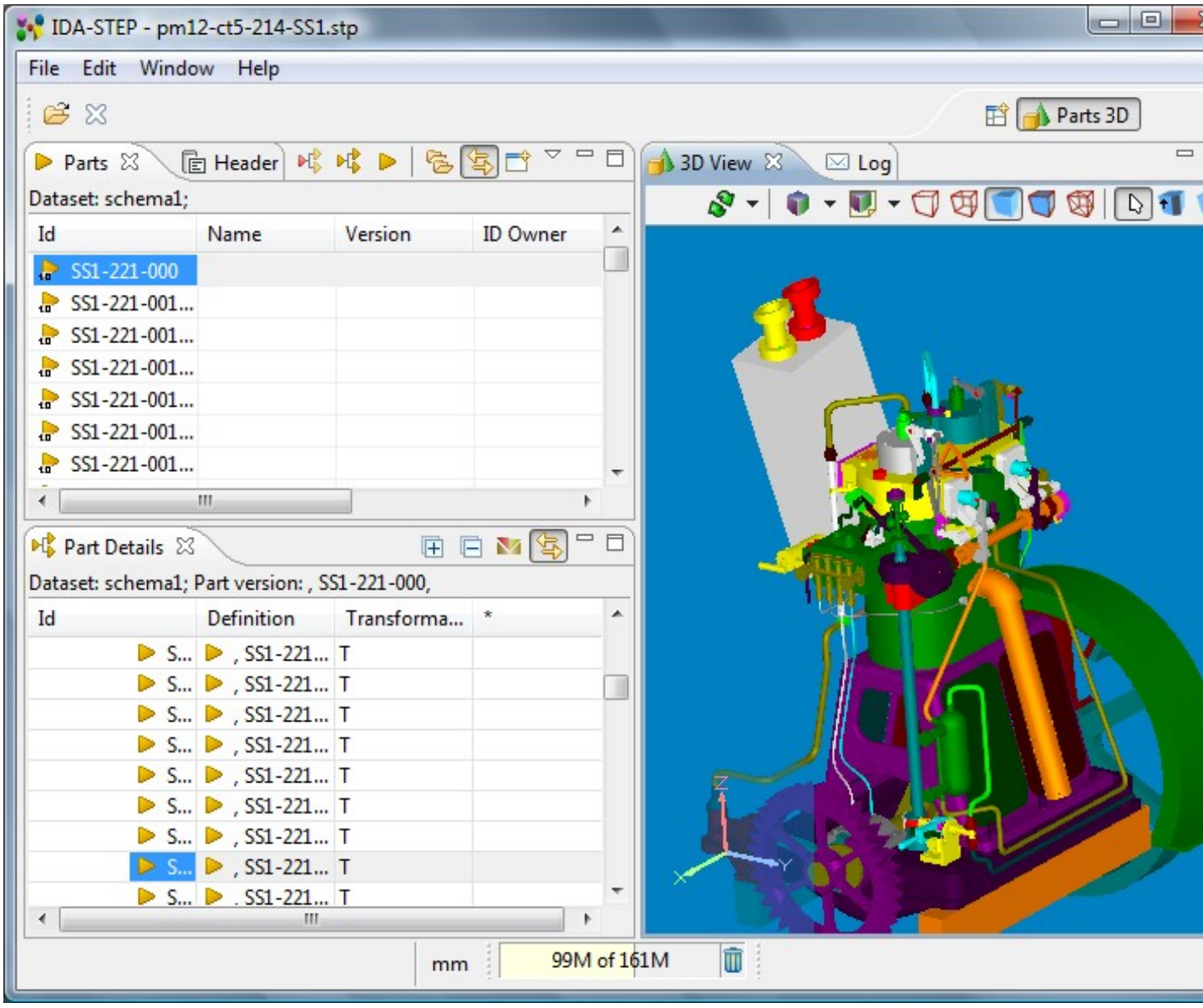

Figure 1. IDA-STEP Viewer Basic feature views

To top

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7.14. Viewer Pro 3D

The **Viewer Pro 3D** provides extended viewing capabilities in a 3D area:

- dynamic columns for customization of data displaying tables,
- Geometry Details view extension (hide/show components, change colour, change transparency, editing styling model),
- GD&T for managing GD&T elements: tolerances, dimensions, annotations and etc.;
  Note. It is not possible to save the created data.
- measurement support in the 3D view;
- printing the contents of the active view;
- exporting of 3D model data to VRML file, STL file.

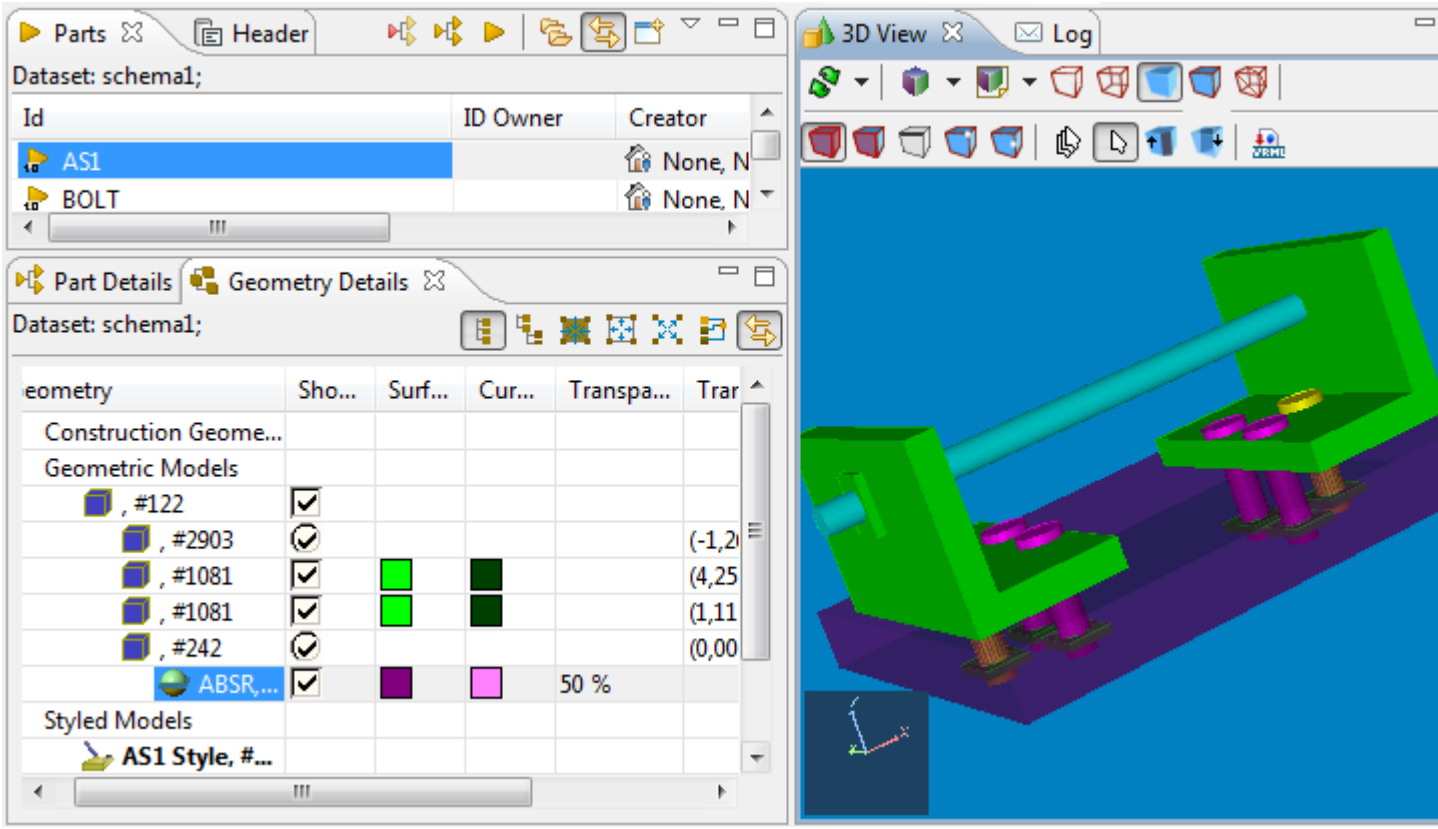

Figure 1. IDA-STEP Viewer Pro 3D functionality presentation - transparency, changed colour

To top

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8. Pre-defined Perspectives

IDA-STEP predefined perspectives are:

- GDT,
- Parts 3D,
- PCB/PCA,
- PCB Layer Library,
- PCB Layer Stack,
- Tasks.

See also:

- Perspectives Preferences
- Perspectives

To top

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8.1. GDT

The **GDT** perspective (see Figure 1) is optimised for reviewing and creating, resp. modifying GD&T data (Geometrically Dimensioning and Tolerancing). For this purpose it is important to have the full height for the **GD&T** view available. The perspective consist of the following views:

- **GD&T**,
- **Geometry Details**,
- **Parts**,
- **Header**,
- **3D View**,
- **Log**.

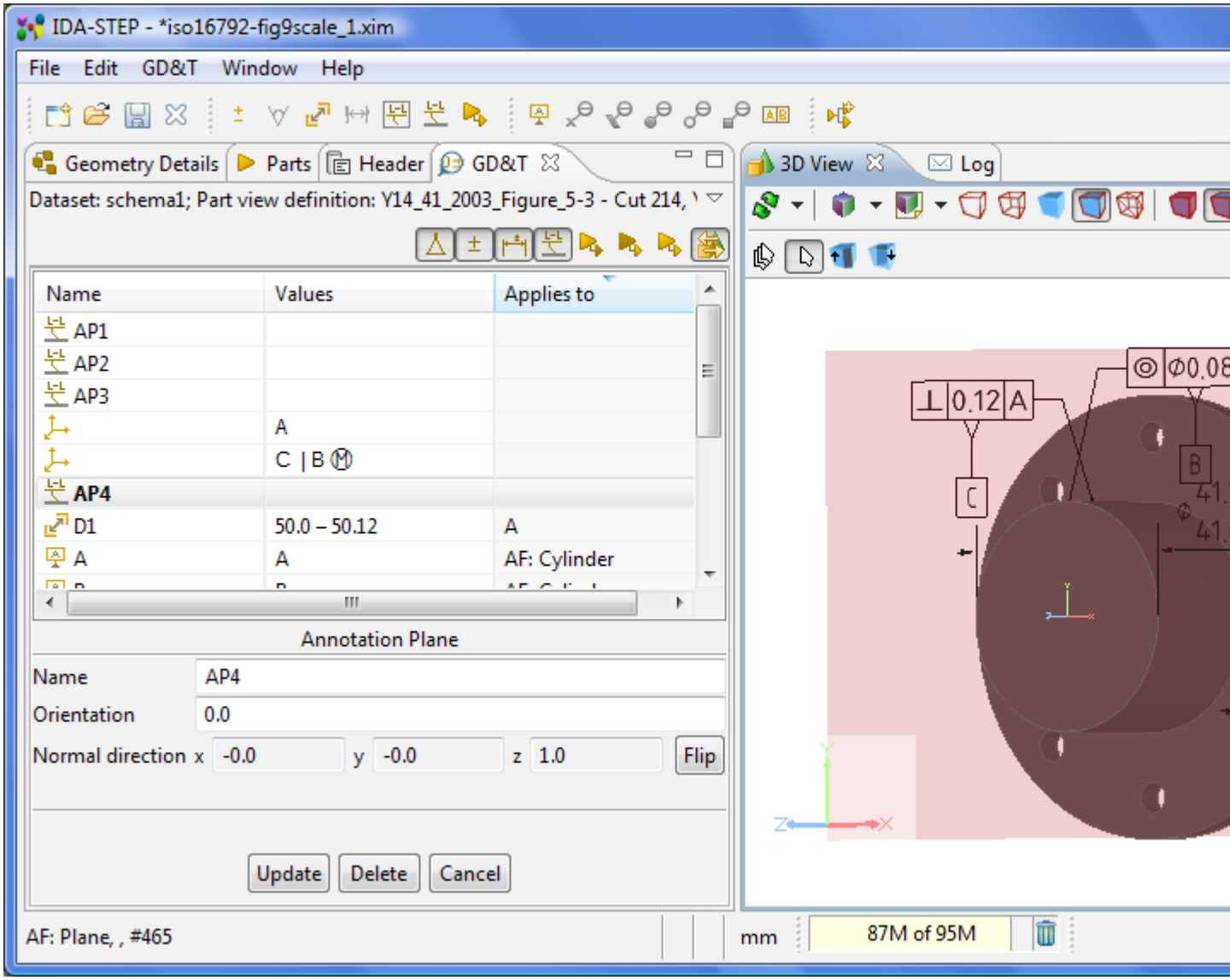

Figure 1. IDA-STEP GDT perspective view

By default, the **GDT** perspective contains the **GD&T** menu in the main menu bar, and the **GD&T** and **Datum** tool bars in the main tool bar line. For more information how to change a layout of the perspective, refer to the clause Perspectives -> Customize Perspective.

The **GDT** perspective has an auto switching policy - automatically open this perspective upon loading a file with GD&T data.
Another useful functionality for GD&T users: a user will be asked if he wants to switch to **GDT** perspective when he creates some GD&T element if all of the following preconditions are met:

1. He has created geometric tolerance, dimension, or datum (indirectly by creating datum target or datum feature).
2. Current perspective is not a **GDT** perspective.
3. The preferences are configured, so that user gets prompted for this (default preferences are set to **Prompt**).
   Note. If preferences are set to **Always open**, then IDA-STEP will automatically switch perspective, if (1) and (2) are met. If preferences are set to **Never open**, then IDA-STEP will not switch to **GDT** automatically on the creation of GD&T elements.

To confirm the perspective switch (see dialogue in Figure 2):

- Press **Yes** button to switch to the **GDT** perspective.

Or

- Press **No** or **Cancel** button to stay in the currently opened perspective.
  Note. You can make an application to remember your selection by selecting the **Remember my decision** option.

Figure 2. Confirm perspective switch dialogue window

The **GD&T** view in the **GDT** perspective displays the GD&T data of the part, selected in **Parts** view. The GD&T elements, selected in the **GD&T** view are highlighted in the **3D View** (in case the button is pressed) and in the **Geometry Details** view.

See also:
Preferences -> General -> Perspectives
Perspectives -> Customize Perspective

To top

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8.2. Parts 3D

The **Parts 3D** perspective is designed to make a work with 3D objects more convenient. The perspective consists of the following views (see Figure 1):

- **Parts**,
- **Part Details**,
- **3D View**,
- **Log**,
- **Header**.

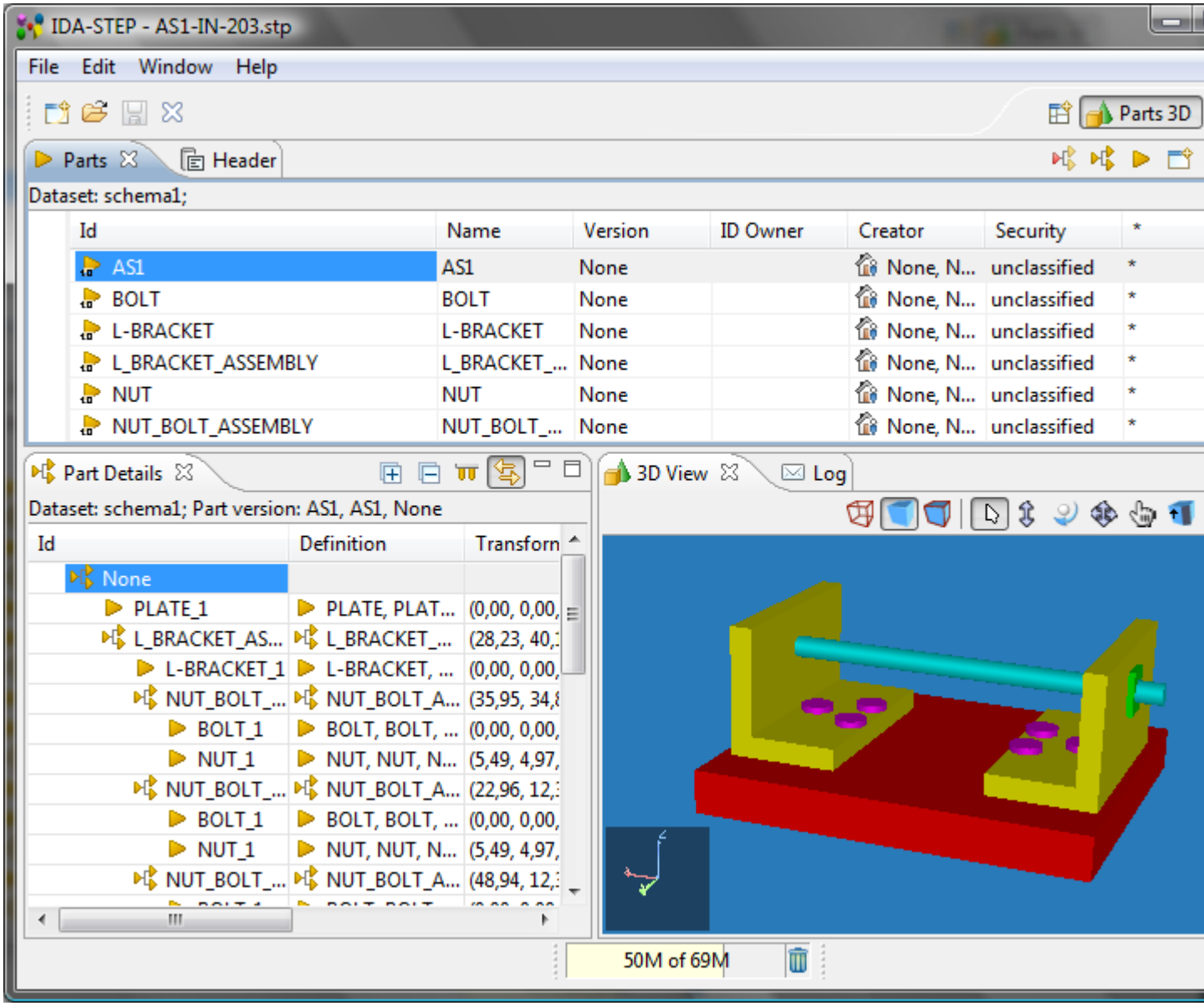

Figure 1. 3D View perspective view

The **Parts** view presents the list of part versions, available in the current dataset. The **Part Details** view presents a hierarchy of part version definitions which are geometrically presented in the **3D View**. The views in this perspective are connected in the following way:

- Selecting a part version in the **Parts** view, the **Part Details** view (also the **Geometry Details** view, which is available in the **IDA-STEP Viewer Pro** and editors) presents the hierarchy of part version definitions and the selected part version (an assembly or a single part) is geometrically presented in the **3D View** (see Figure 2).

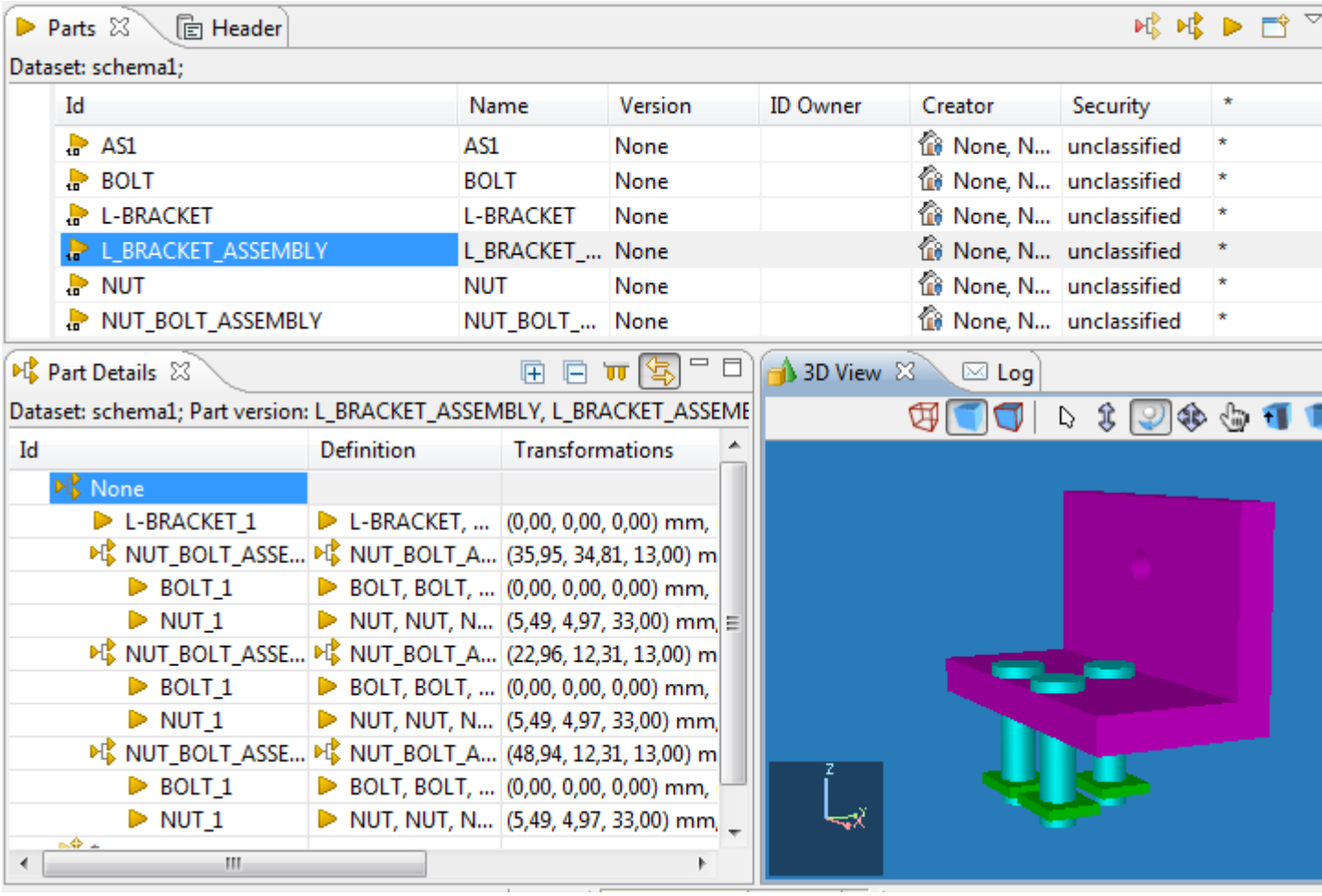

Figure 2. Part assembly version ('L_BRACKET_ASSEMBLY') in the Parts view, the hierarchy of part version definitions in the Part Details view and 3D geometrical representation in the 3D View

- Selecting an applicable component node in the **Part Details** view, highlights the particular component in the **3D View** (see Figure 3). Also an applicable component can be selected in the **3D View**. After the selection, the particular part version definition is highlighted in the **Part Details** view.

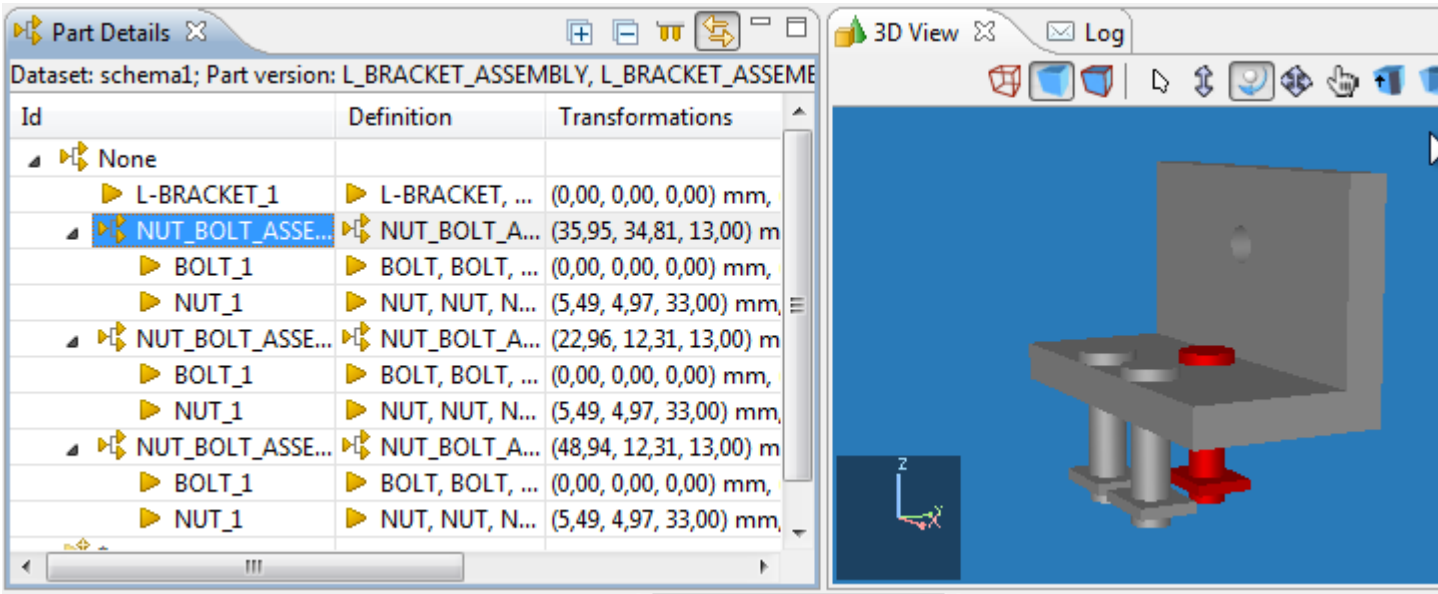

Figure 3. Selected component (bracket) is highlighted in 3D View

Note. The functionality, described below, is available only in **IDA-STEP Viewer Pro**.

- Selecting/Unselecting the **Show/Hide** check box next to an applicable component in the

Geometry Details view, shows/hides the particular component in the **3D View** (see Figure 4).
Note. The **Geometry Details** view can be opened by selecting **Window -> Show view -> Other -> Geometry -> Geometry Details**.

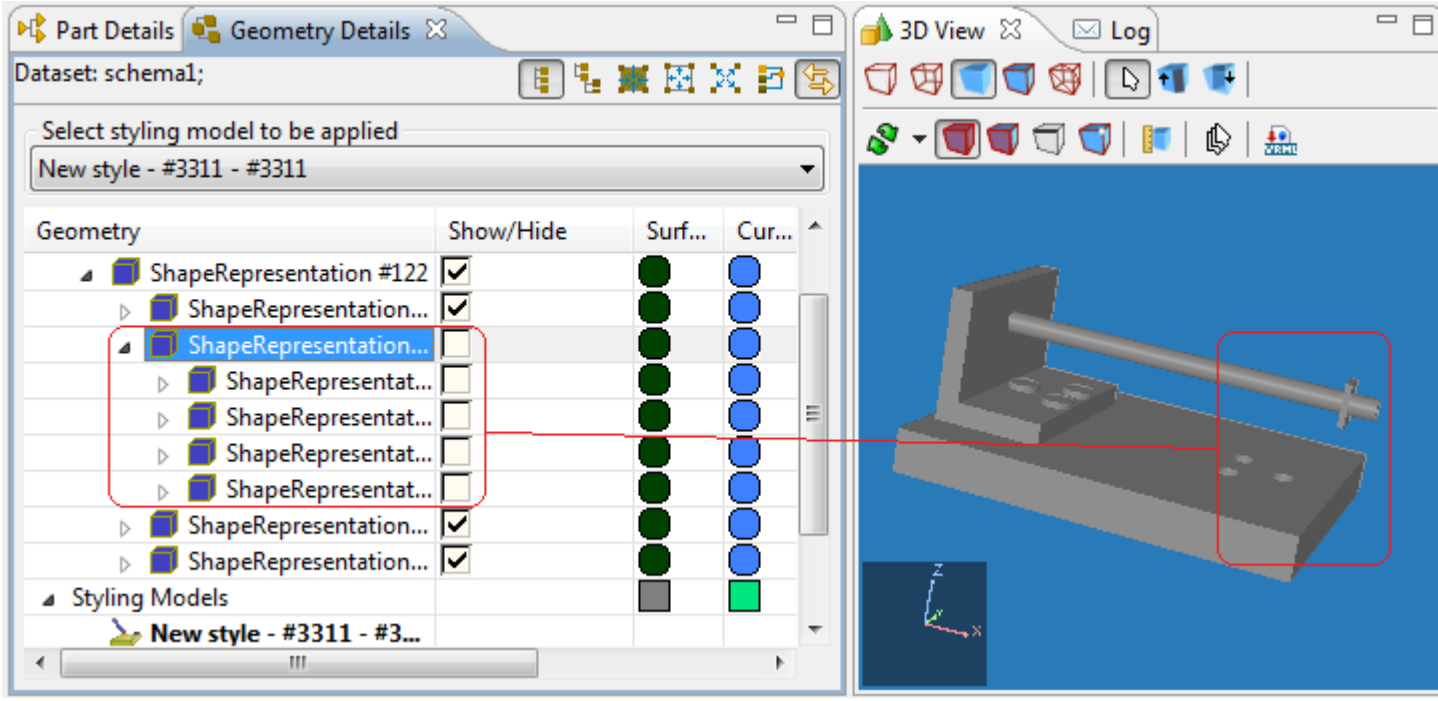

Figure 4. Selected component is hidden in 3D View together with all related components (unselected check boxes in Geometry Details view)

- Changing representation properties (transparency, surface or curve color) in the **Geometry Details** view (see Figure 5), the changes are reflected in the **3D View**.

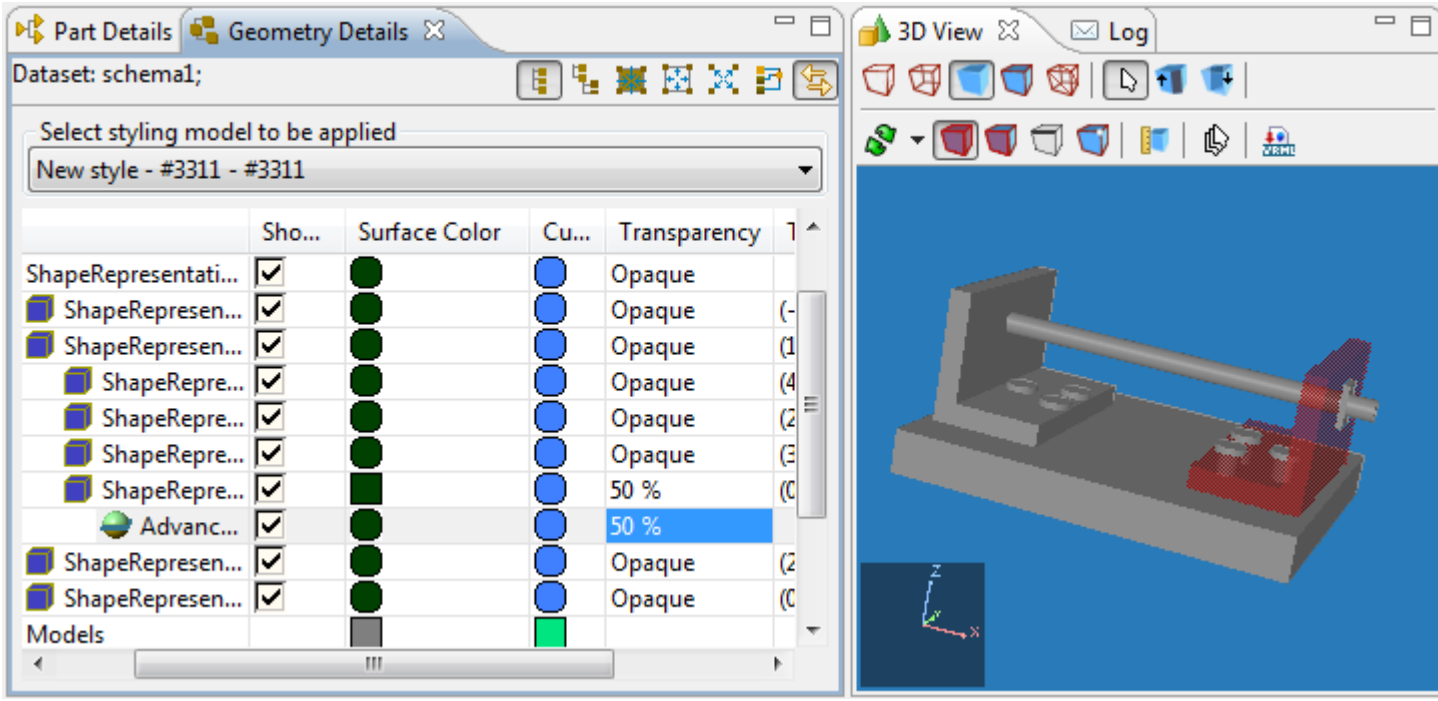

Figure 5. Changed transparency level value in Transparency column (Geometry Details view) and transparent object in 3D View

- Clicking on an applicable component in the **3D View** (being in smart or picking mode), highlights the particular component node in the **Geometry Details** view (see Figure 6).

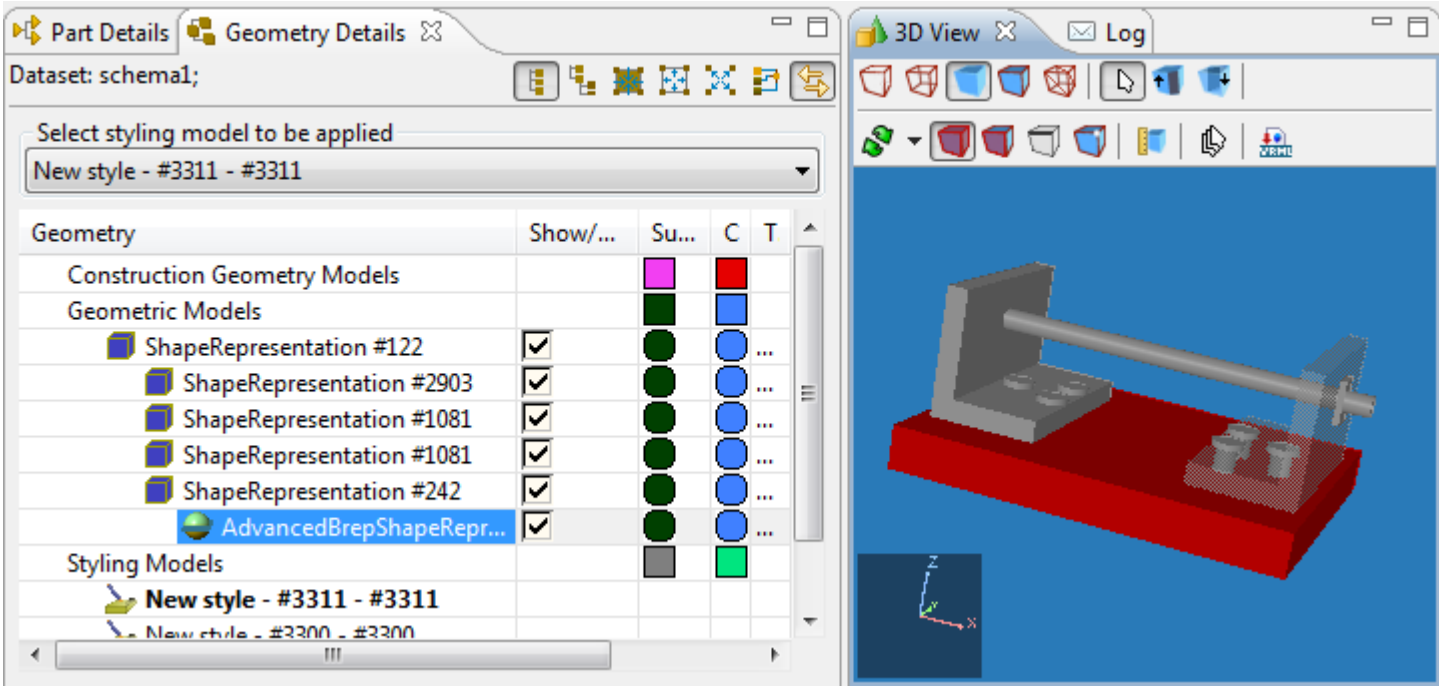

Figure 6. Picked component in 3D View is highlighted in Geometry Details view hierarchy

To top

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8.3. PCB Layer Library

The **PCB Layer Library** perspective is optimized for managing stratum and passage technologies. The perspective consist of the following views (see Figure 1):

- **Datasets** - presents available datasets for the current repository,
- **Log**,
- **Stratum Technologies** - presents available stratum technologies with the defined properties,
- **Passage Technologies** - presents available passage technologies with the defined properties.

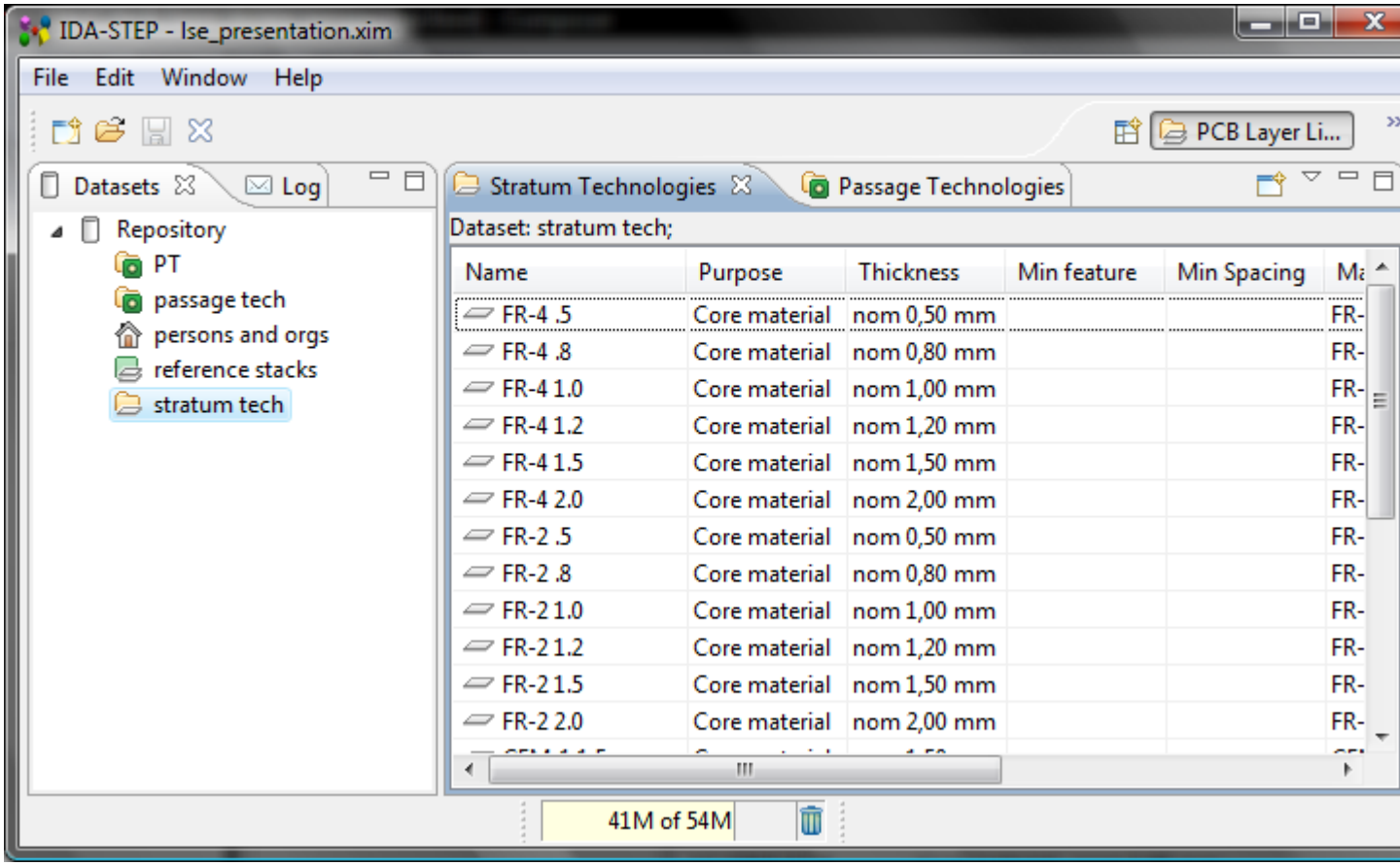

Figure 1. PCB Layer Library perspective view

The views in this perspective are connected in the following way:

- Selecting an applicable dataset of type *stratum technology* in the **Datasets** view activates the **Stratum technologies** view (see Figure 1).
- Selecting an applicable dataset of type *passage technology* in the **Datasets** view activates the **Passage technologies** view (see Figure 2).

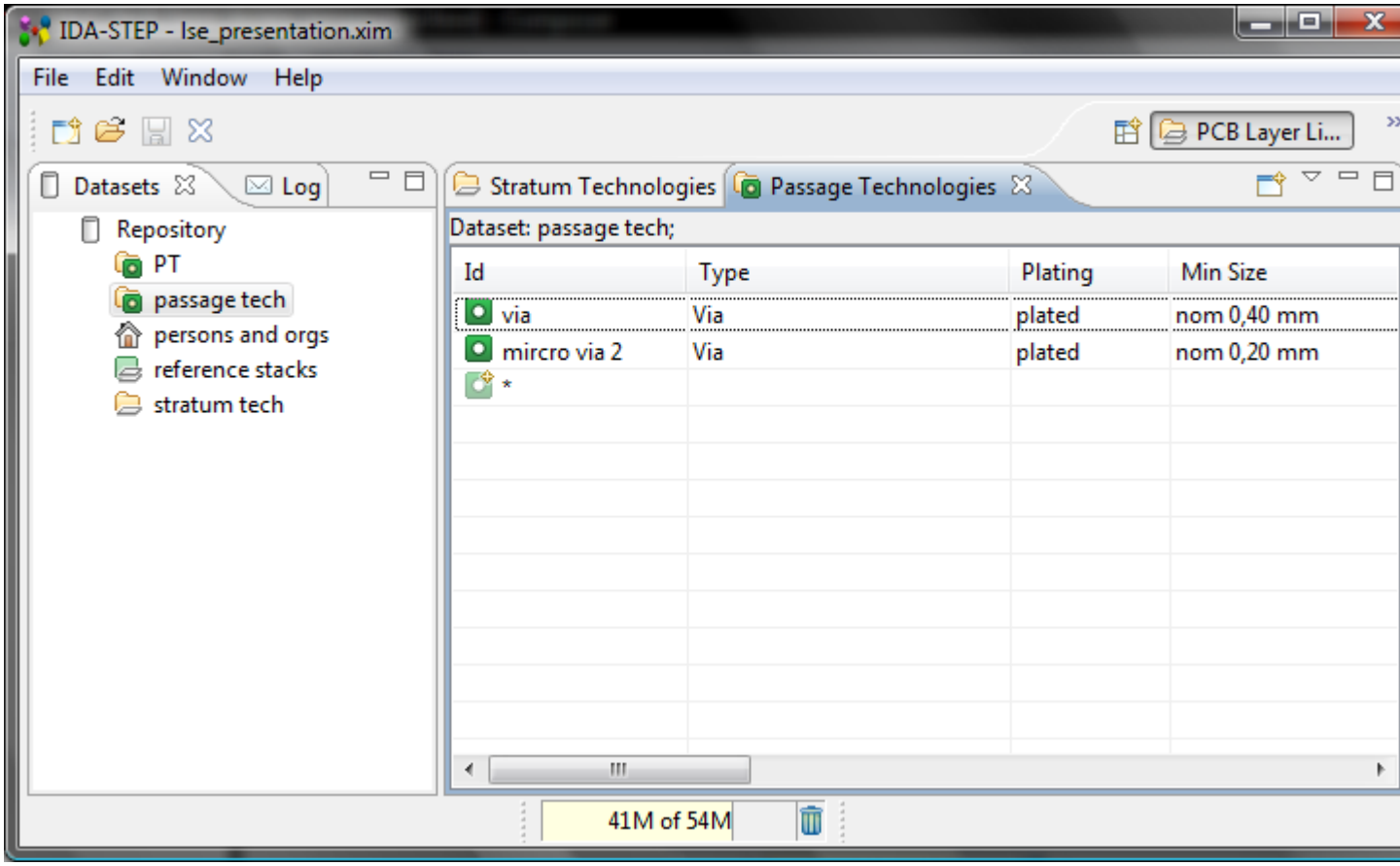

Figure 2. Selected dataset of type passage technology and activated Passage Technologies view

To top

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8.4. PCB Layer Stack

The **PCB Layer Stack** perspective is optimized for managing layers and other stratums, passages, design stack models and other PCB data. The perspective consist of the following views:

- **Datasets** - list of available datasets for the current repository,
- **Design Stack Models** - presents a list of design stack models for the selected dataset,
- **Log**,
- **Stratums** - presents a tree table with a hierarchy of layers/stratums, referenced by a particular design stack model,
- **Passages** - presents a tree table with a hierarchy of passages, referenced by a particular design stack model,
- **Stratums Cross-section** - presents the visual 2D presentation of objects, defined in **Stratums** and **Passages** view.

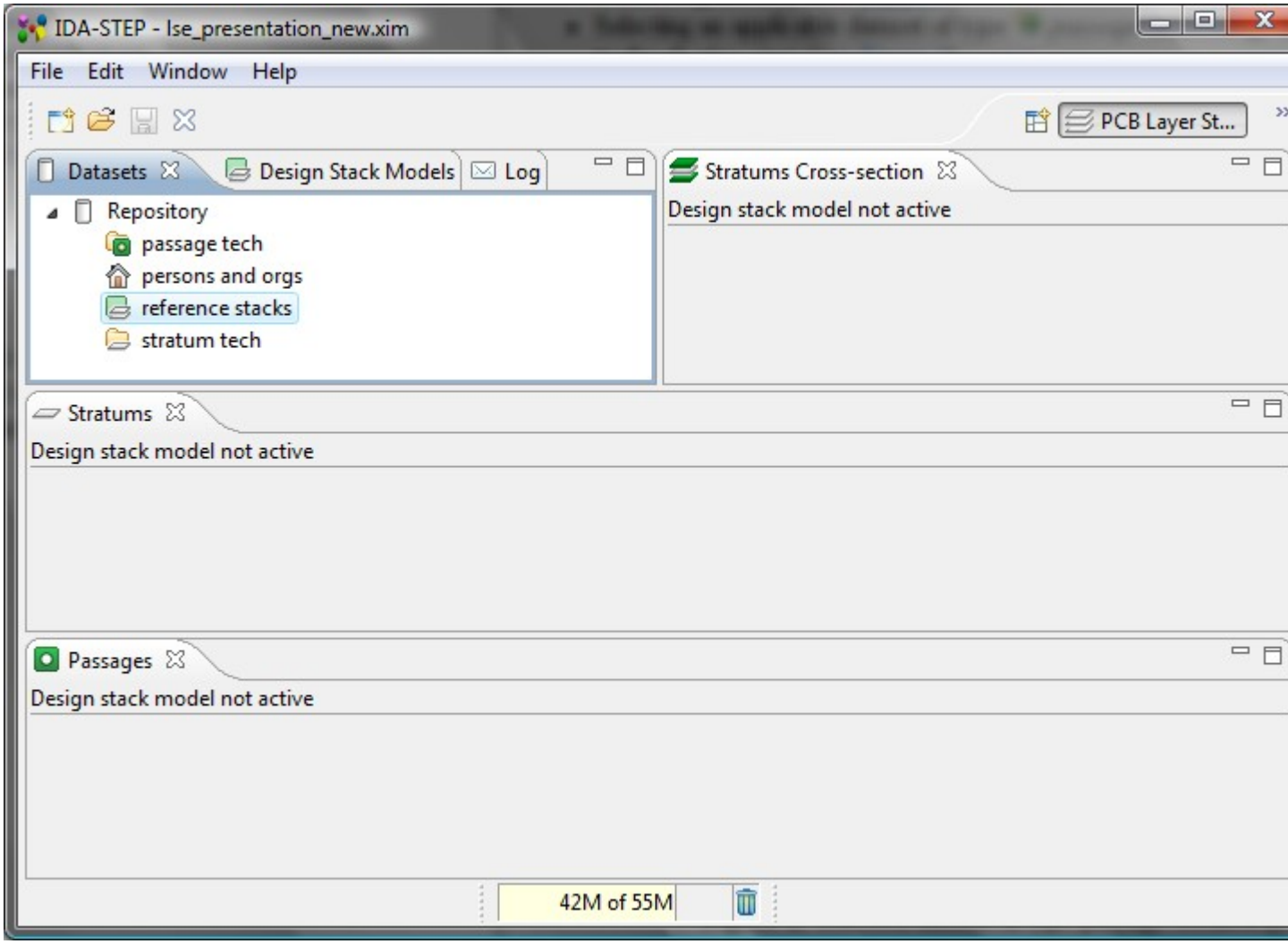

Figure 1. PCB Layer Stack perspective view

The views in this perspective are connected in the following way:

- Selecting a dataset of type *design stack model* in the **Datasets** view activates the **Design Stack Models** view. Other views are filled with available data for the first design stack model element (see Figure 2).

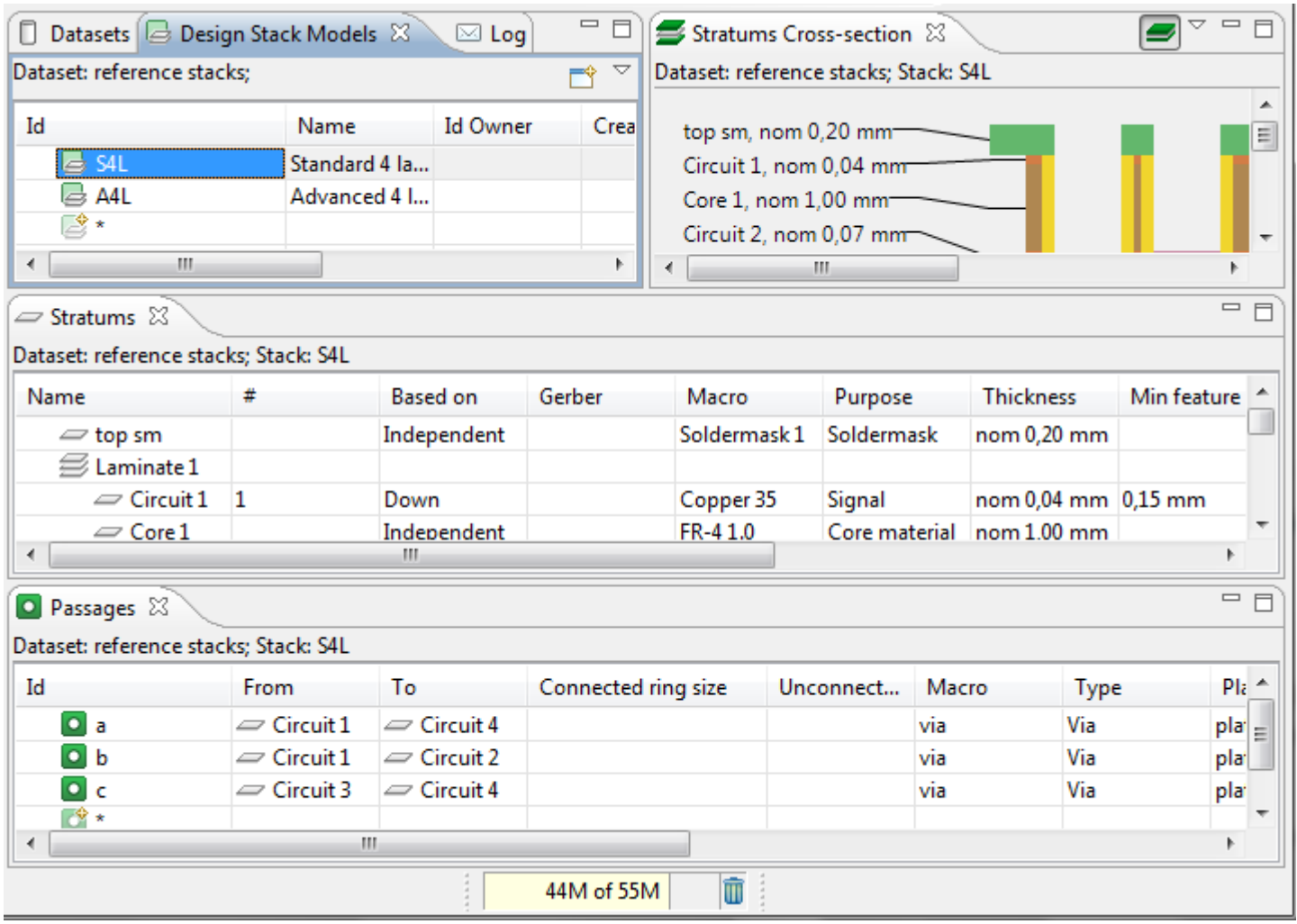

Figure 2. Activated Design Stack Model view and other views are shown with data

- Selecting a particular element in the **Design Stack Model** view shows the referred data in the **Stratums,** **Passages** and **Stratums Cross-section** views.
- Selecting an element in the **Stratums** view shows the selected element highlighted in the **Stratums Cross-section** view (see Figure 3).

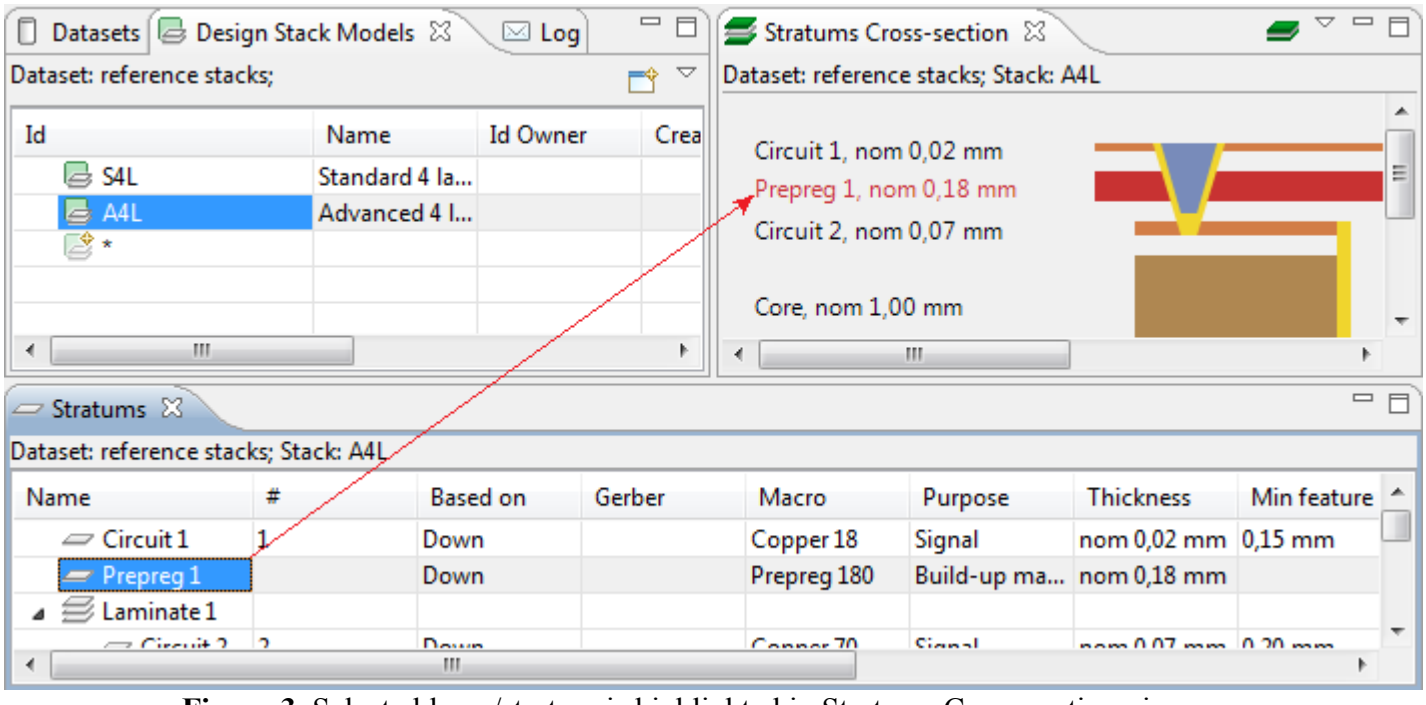

Figure 3. Selected layer/stratum is highlighted in Stratums Cross-section view

- Selecting an element in the **Passages** view shows the selected element highlighted in the

Stratums Cross-section view (see Figure 4).

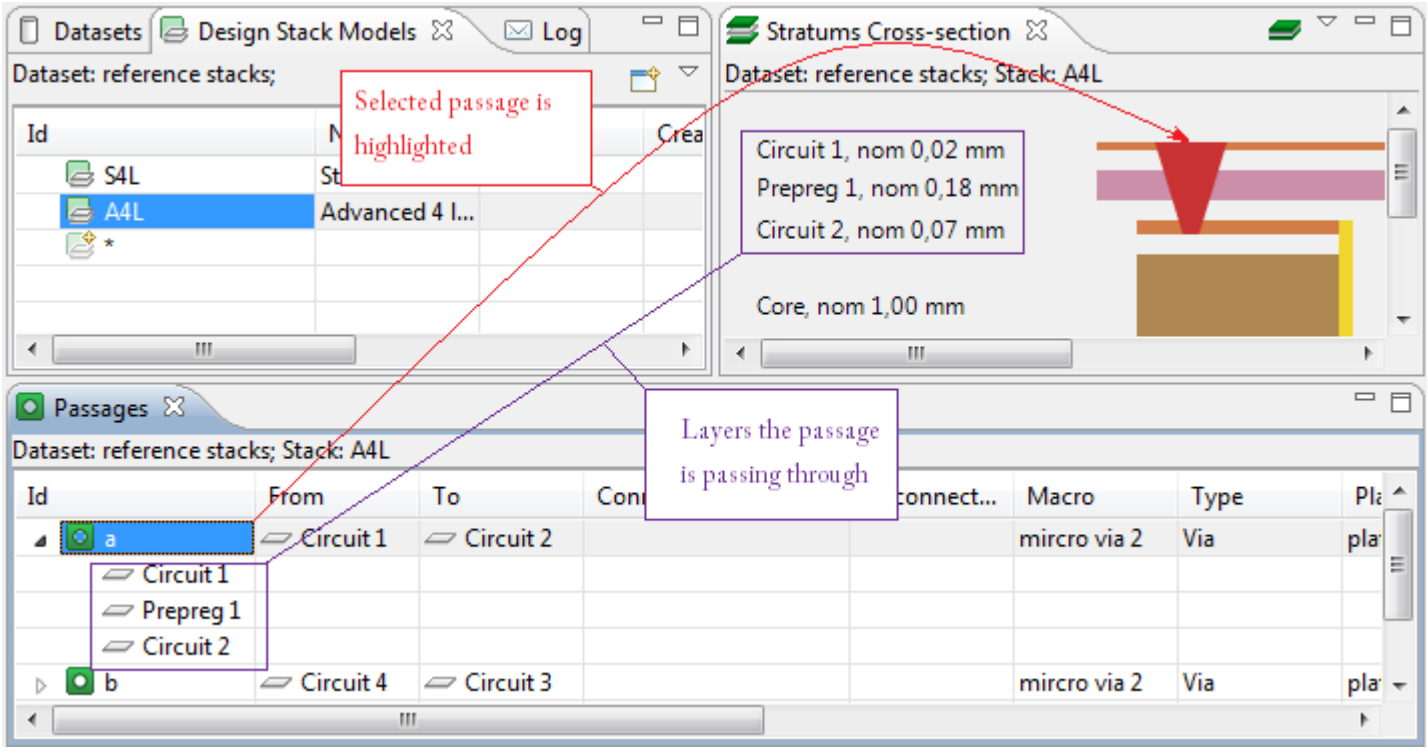

Figure 4. Selected passage is highlighted in Stratums Cross-section view

To top

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8.5. PCB/PCA

The **PCB/PCA** perspective is optimized for viewing PCB/PCA data, it consists of the following views (see Figure 1):

- **Parts**,
- **Header**,
- **Part Details,**
- **Display settings**,
- **2D View**,
- **Log**.

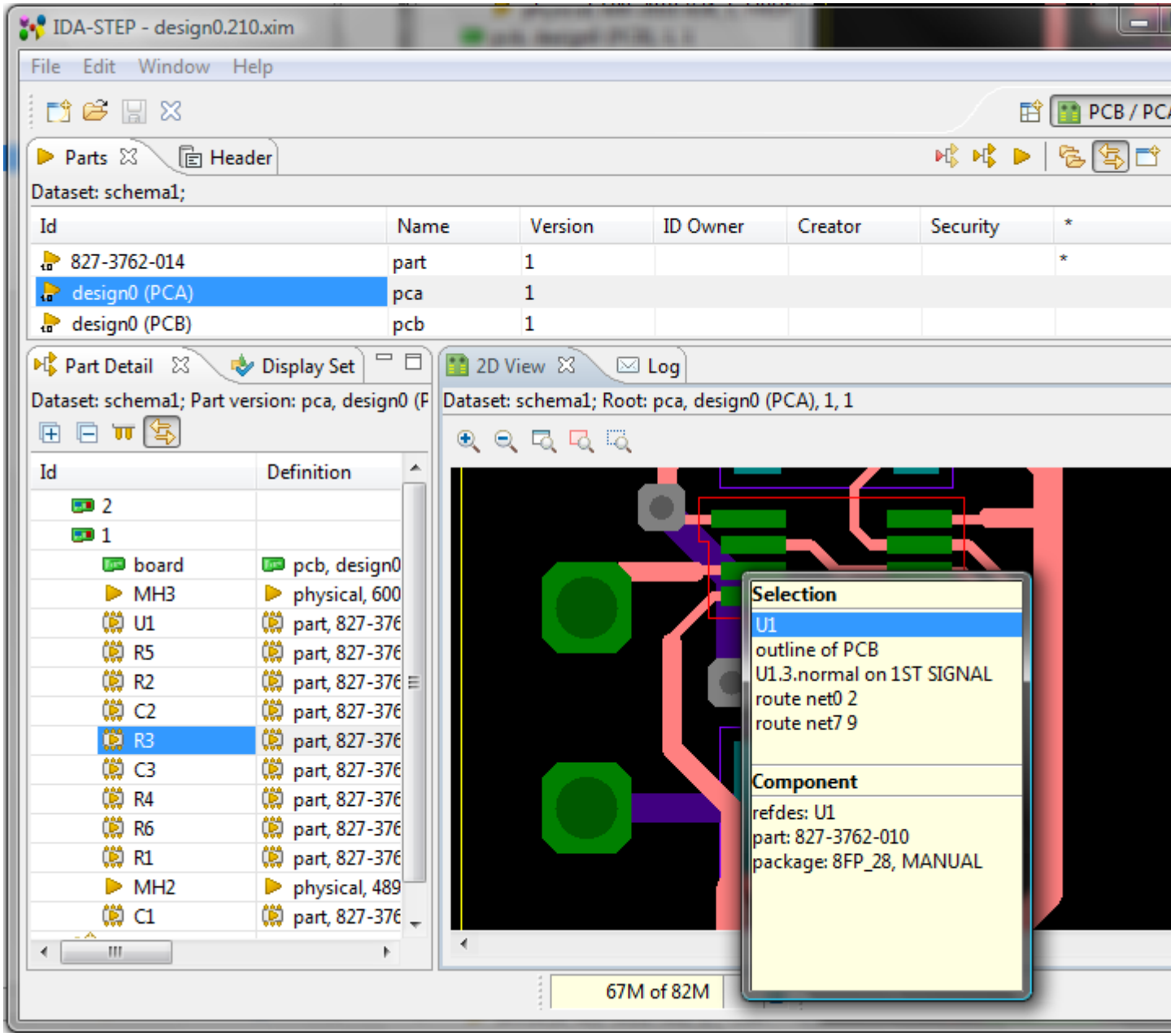

Figure 1. PCB/PCA perspective view

The **Parts** view contains the list of part versions, available in the opened repository. The **Part Details** view displays the assembly structure of PCA and its information. The content of **Part Details** view depends on selection in the **Parts** view. The **2D View** provides convenient tools to examine the PCB layout and retrieve information on selected parts. The views in this perspective are connected in the following way:

- Selecting a PCA component in the **Part Details** view, highlights a particular component in **2D View** (see Figure 2).

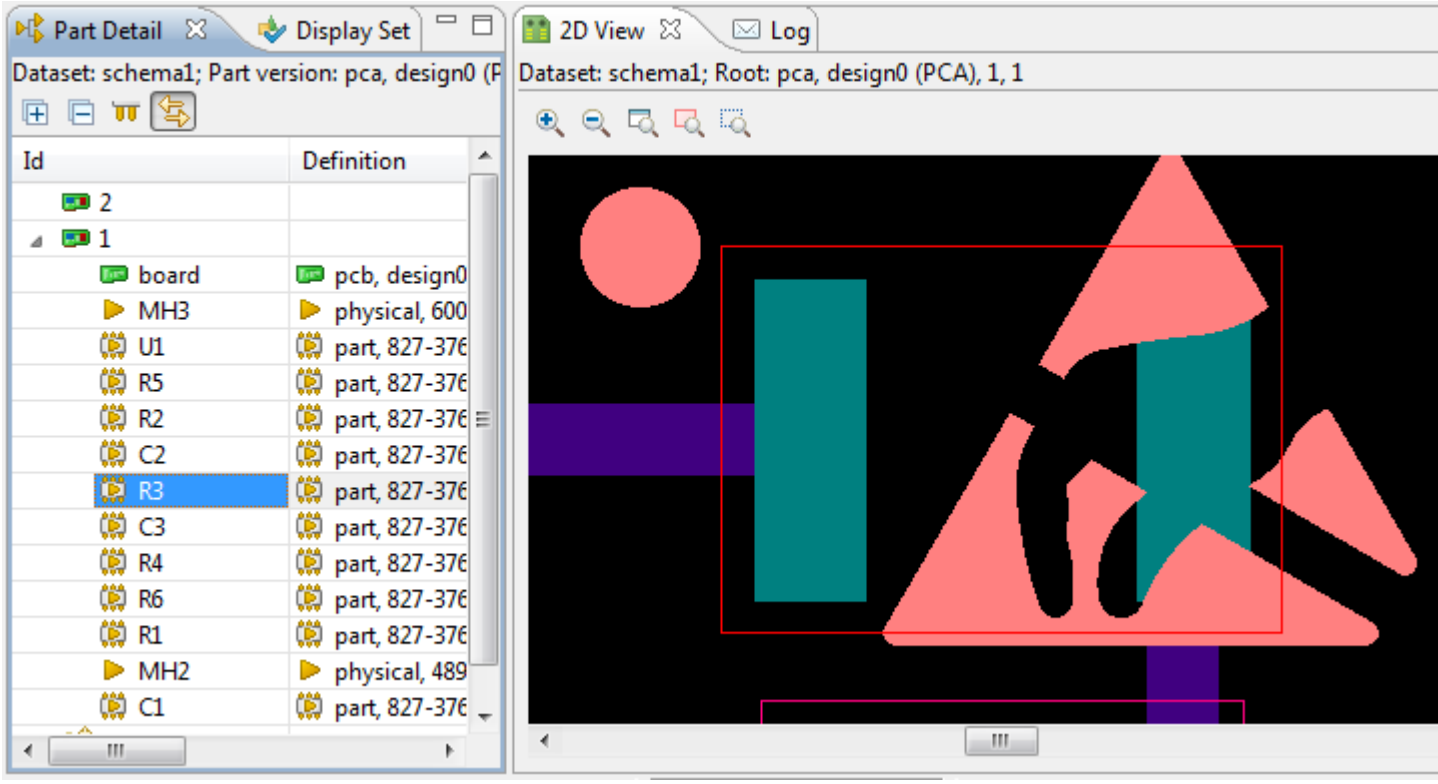

Figure 2. Selected part is highlighted in 2D View (red color)

- Changing settings in the **Display settings** view, the **2D View** presents, how changes are applied to the design components (see Figure 3).

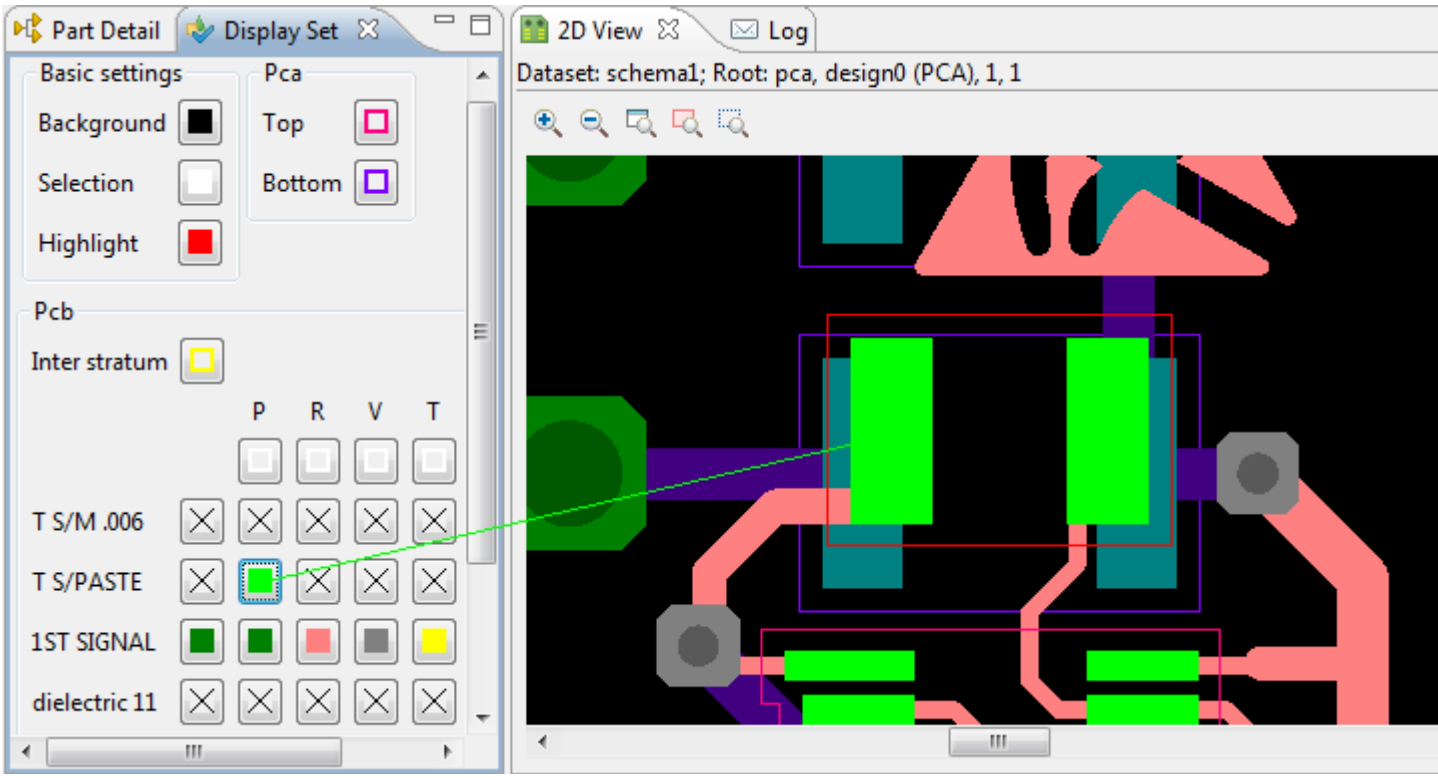

Figure 3. Changed color in Display settings view is applied to component in 2D View

To top

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8.6. Tasks

The **Tasks** perspective is optimized for maintaining existing PDM data in the STEP files or to create a training system by indicating tasks and the corresponding information. The perspective consist of the following views (see Figure 1):

- **Datasets**,
- **Log,**
- **Tasks**,
- **Zonal Breakdowns**,
- **Part Details**,
- **Persons and Organizations**,
- **Parts**,
- **Documents**.

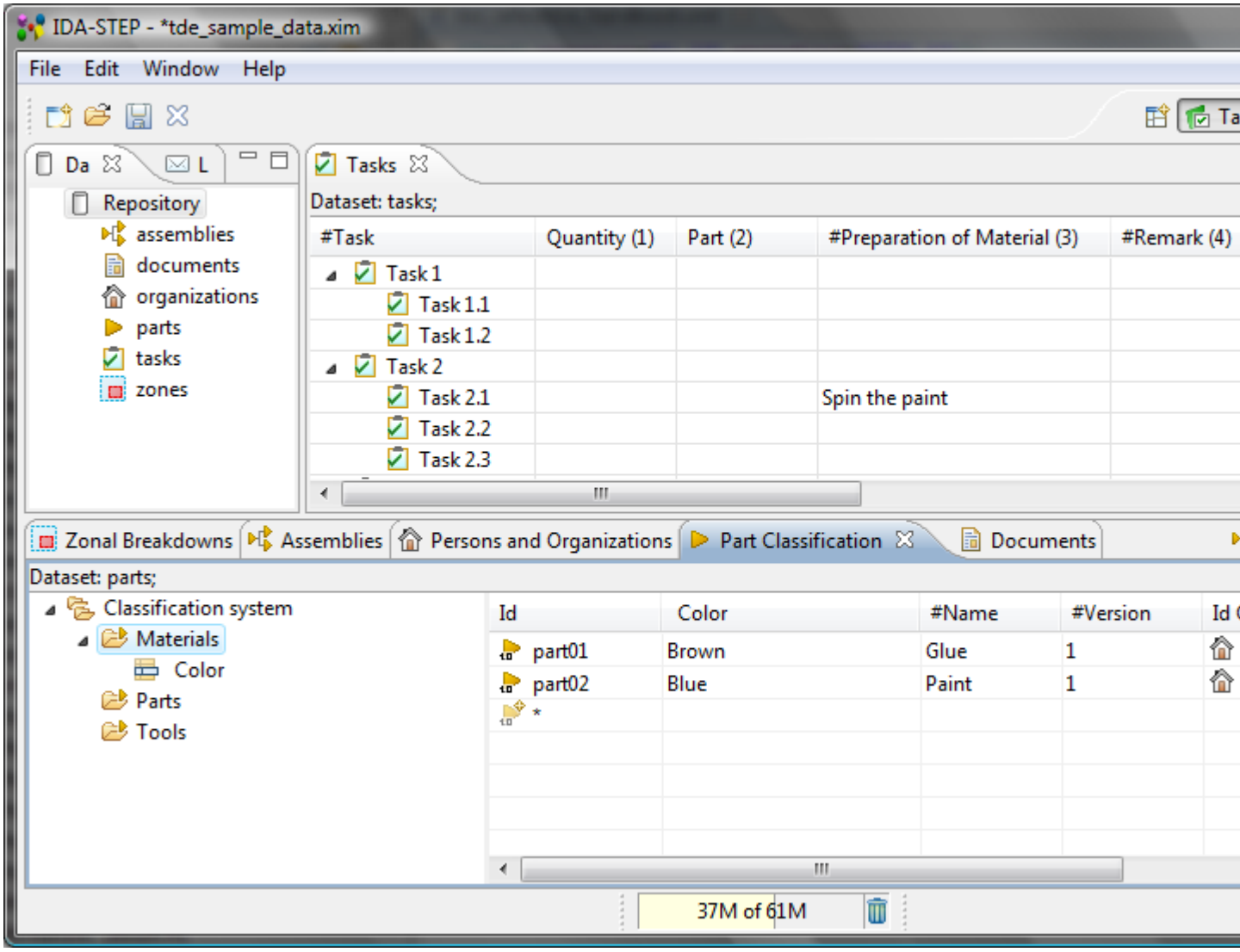

Figure 1. Tasks perspective view

The views in this perspective are connected in the following way:

- Double clicking on a dataset in the **Datasets** view activates a particular view, the selected dataset is assigned to, on the screen.

To top

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9.1.1. Opening 2D Data View

Follow the routine to open 2D data view in the **2D View** page:

- Open the **Part Details** view and **2D View** in the workbench by using main menu **Window -> Show View ->...** .
- Open a particular STEP file, containing 2D data.

Or

- Select the **PCA/PCB** perspective by selecting in the main menu **Window -> Open Perspective -> Other -> PCA/PCB**.

- Click on the assembly node in the **Part Details** view to display it in the **2D View** page. If the child element of the root assembly is selected in the **Part Details** view, it is highlighted in the **2D View**.

To top

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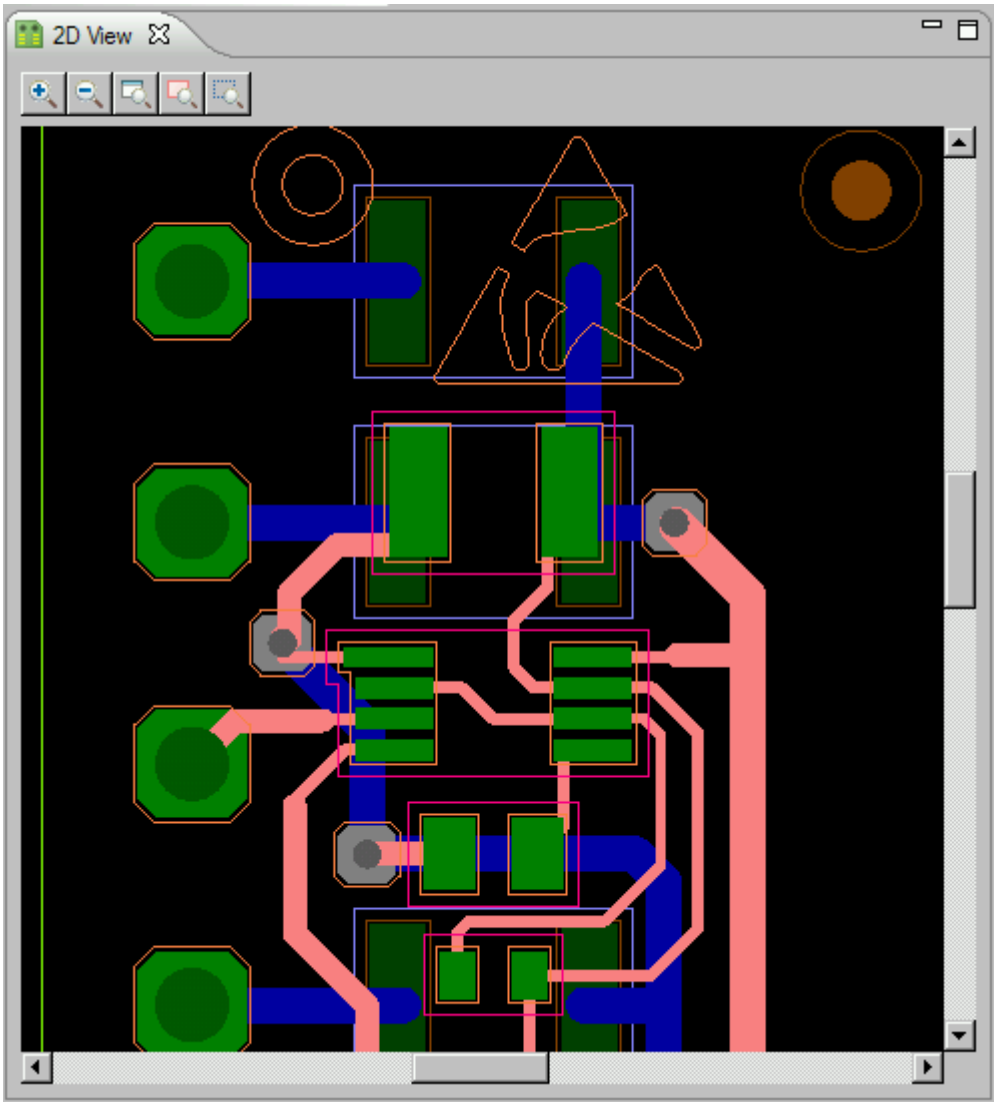
9.1.2. 2D View

The **2D View** provides a two-dimensional geometry representation. Currently it includes zooming and scrolling, coloring and highlighting components.



The **2D View** has it's own toolbar:

Button icon	Description
	Zoom in
	Zoom out
	Zoom to fit the window
	Zoom to highlighted object
	Zoom window

To top

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9.1.3. Zoom In

To increase a graphic view in the **2D View**, click button. The graphic view is increased according to your clicking on the button.

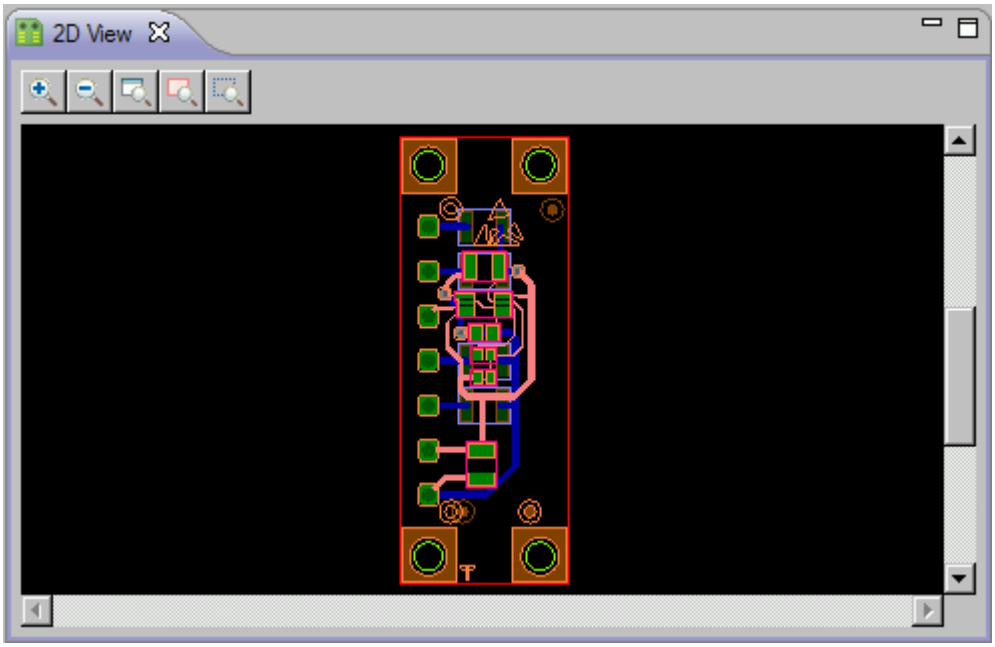

Figure 1. Initial view

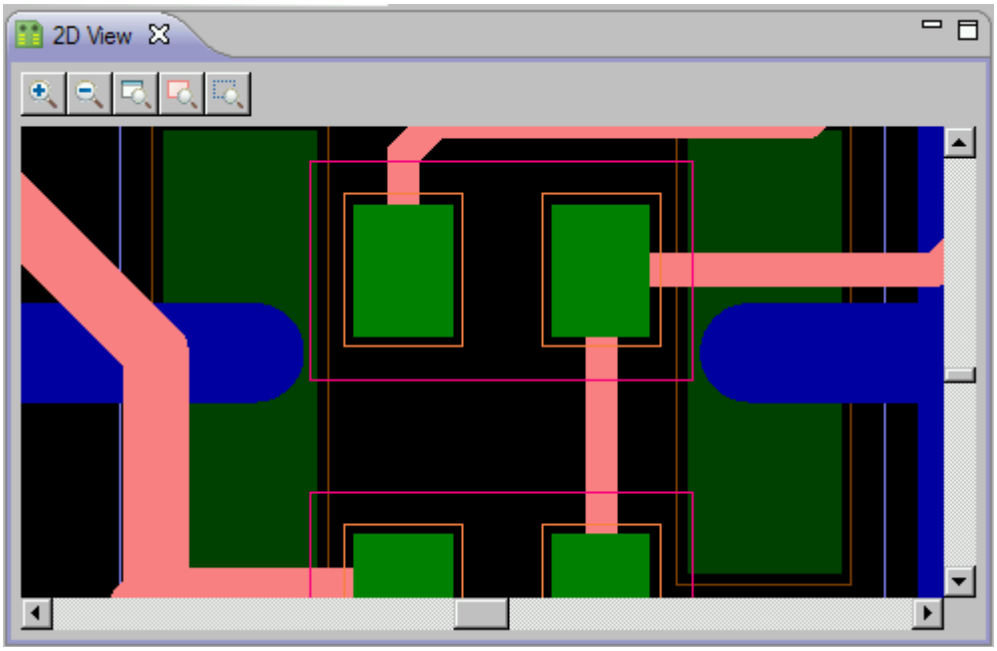

Figure 2. Zoomed in view

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9.1.4. Zoom Out

To reduce a graphic view in the **2D View**, click button. The graphic view is reduced according to your clicking on the button.

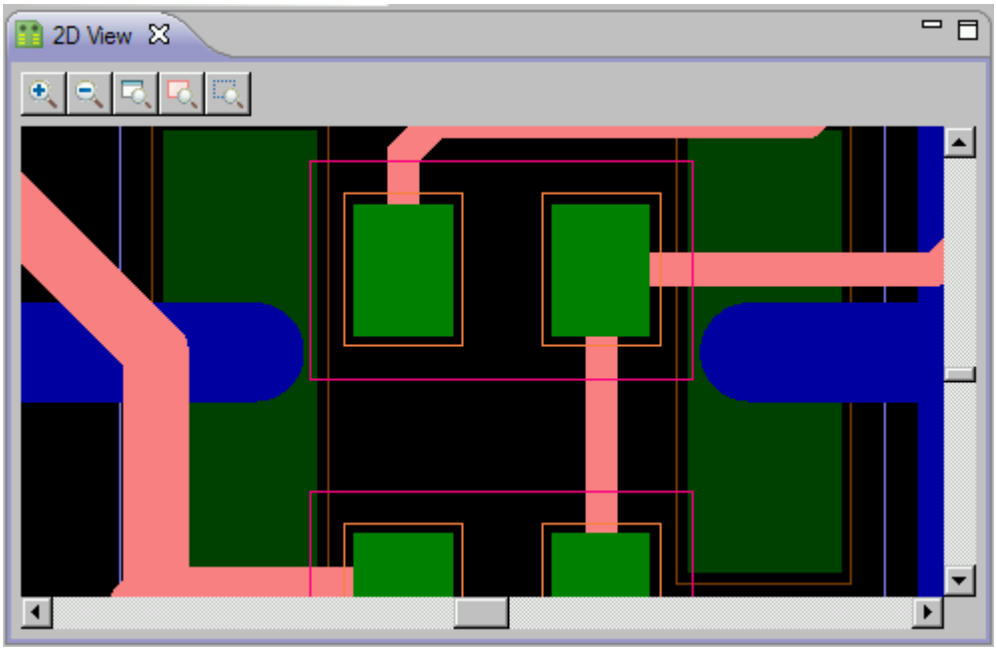

Figure 1. Initial view

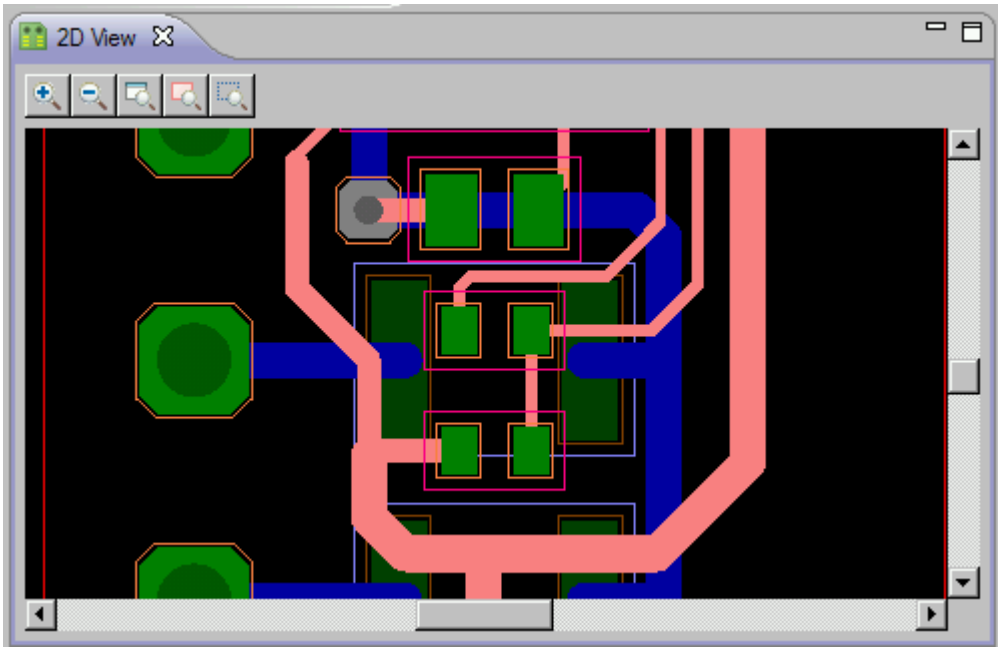

Figure 2. Zoomed out view

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9.1.5. Zoom to Fit

To apply a graphic view to the **2D View** window, click button.

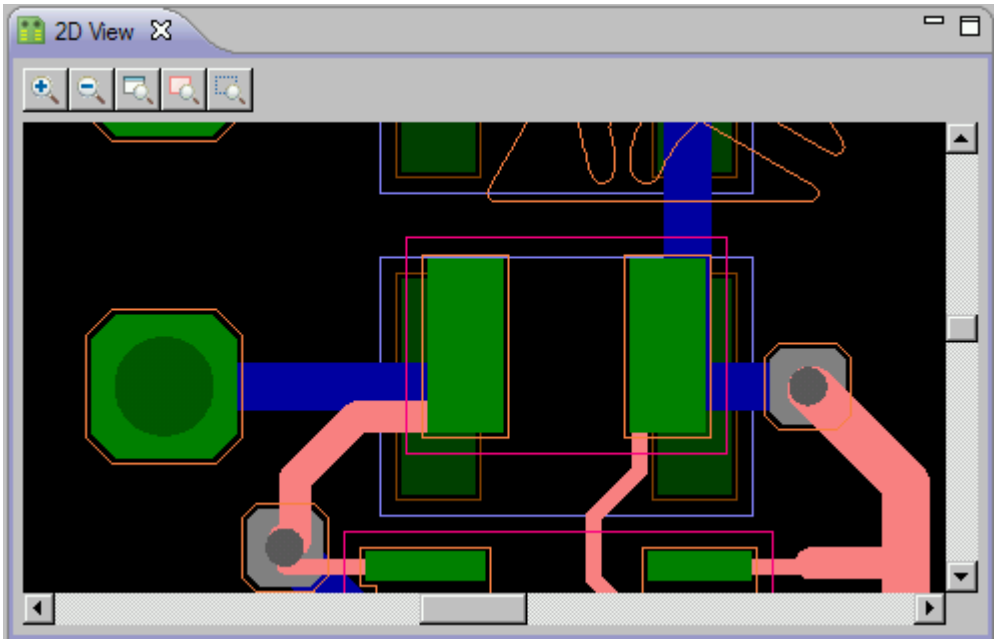

Figure 1. Initial view

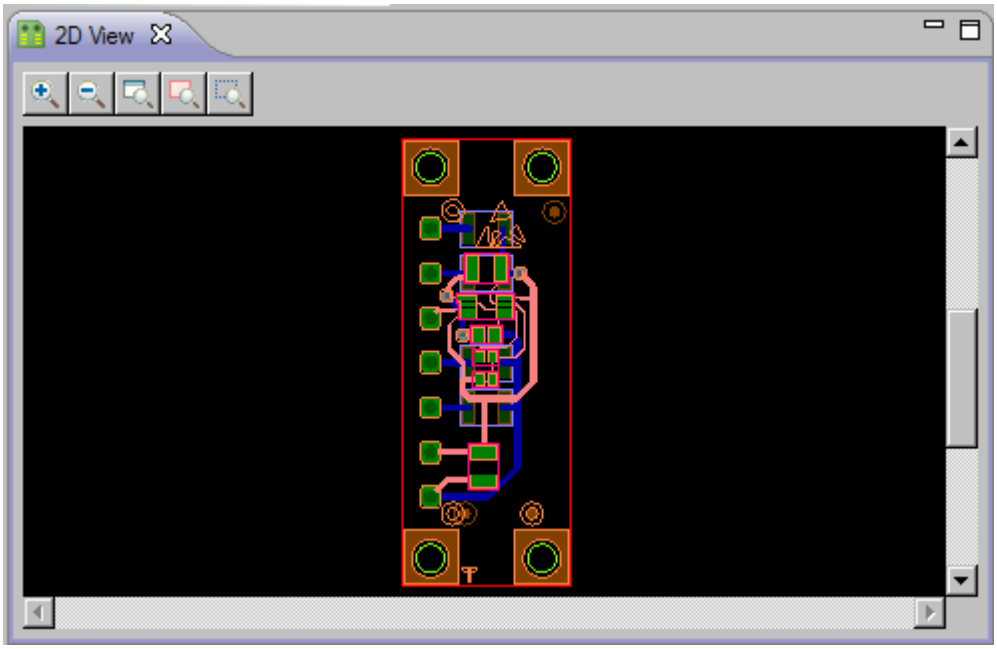

Figure 2. Fit to window view

To top

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9.1.6. Zoom to Highlighted

To zoom and center the highlighted graphic object in the **2D View** (see Figure 1), click button. The highlighted object is enlarged and centered in the **2D View** (see Figure 2).

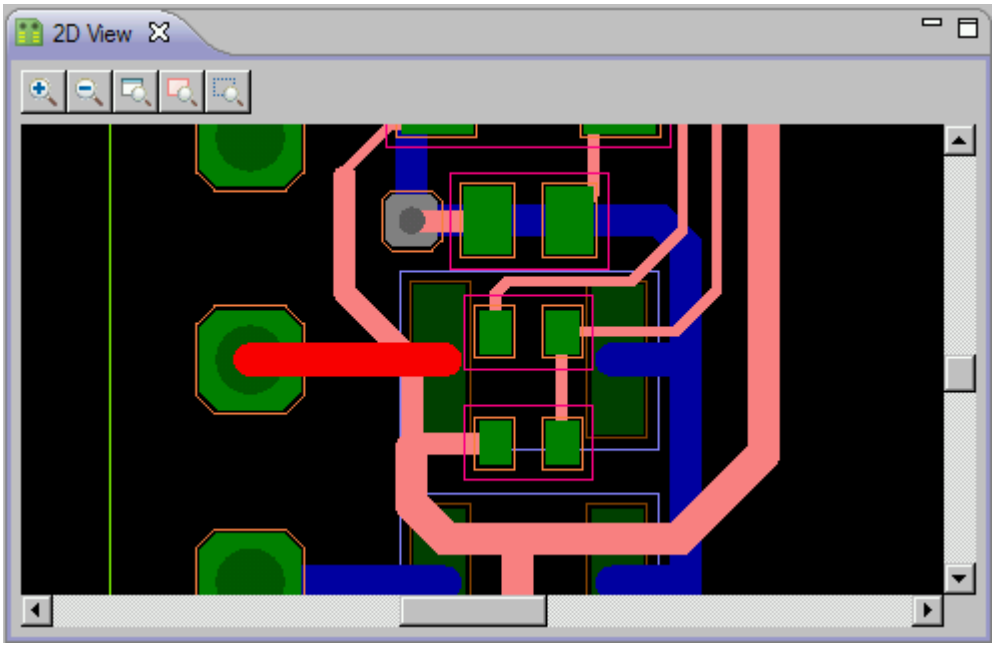

Figure 1. 2D View with highlighted component

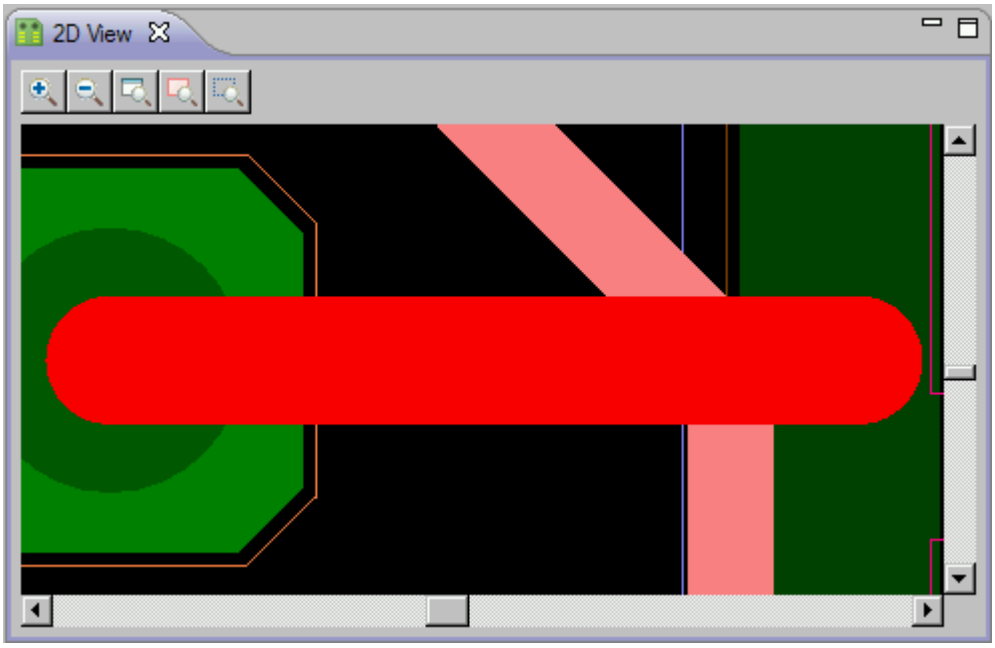

Figure 2. Zoomed to highlighted view

To top

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9.1.7. Zoom Window

To zoom a part of an object in the **2D view**, click button to switch on the *zoom window*. Point and drag with a mouse on the 2D View area to define the area to be zoomed in (white colour rectangle) (see Figure 1). Release the mouse mutton to view the zoomed area (see Figure 2).

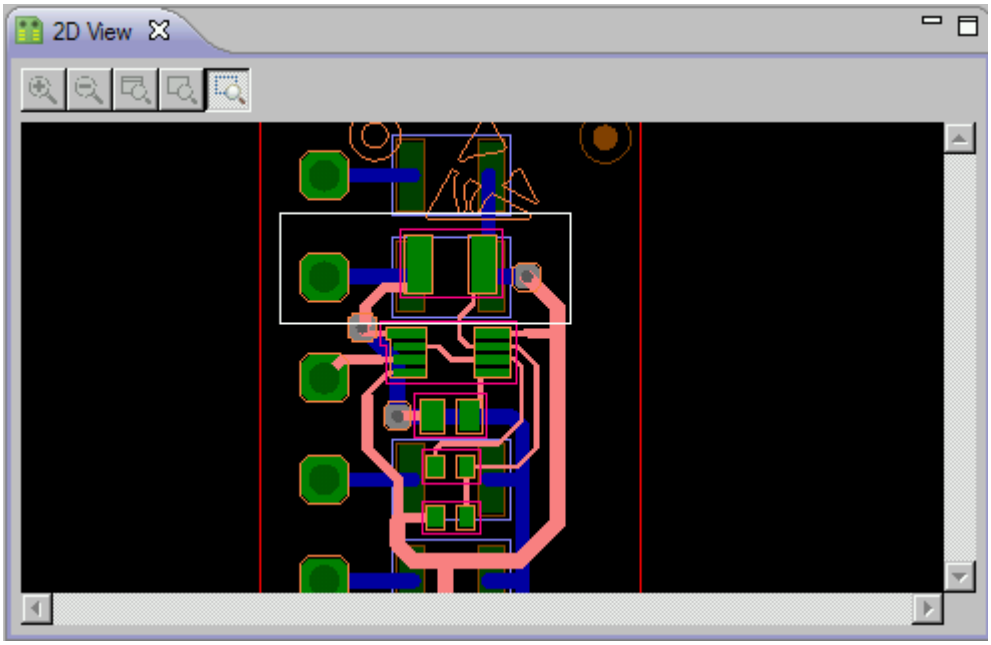

Figure 1. Selected area (left side) is zoomed in (right side)

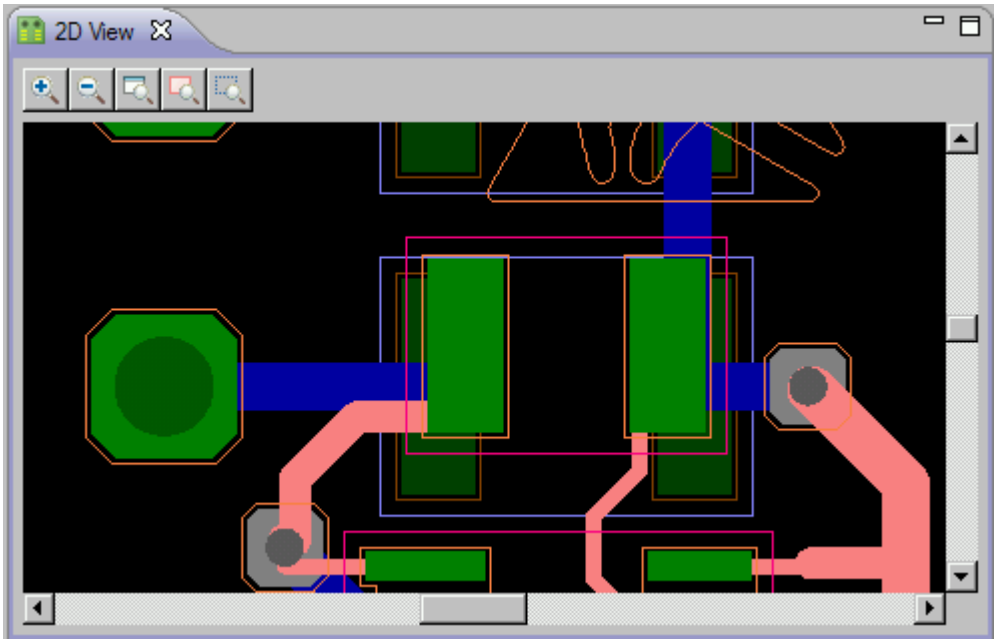

Figure 2. Zoomed area view

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9.1.8. Highlight Object

To highlight an object in the **2D View**:

- Click on a graphic object in the **2D View** area.
  Note: there can be several objects in the pointed area, therefore a context sensitive menu appears to select a particular object you want to highlight.
- Select an object from the context sensitive menu. The selected object is highlighted (by default, red colour) (see Figure 1). You can change highlighting colour in the **Display settings** view.

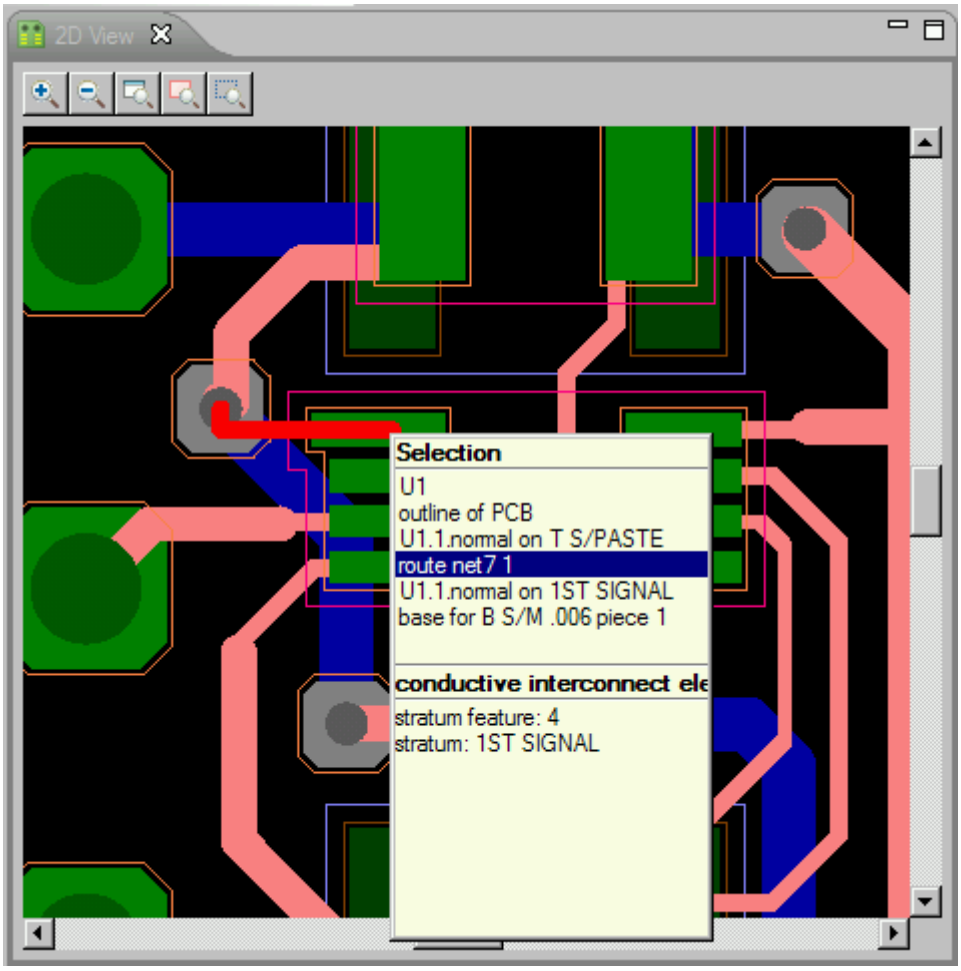

Figure 1. The selected object is highlighted in 2D View directly

To top

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9.1.9. Display Settings

The **Display settings** view provides a possibility to customize colours for the graphical representation in the **2D View** (see Figure 1).

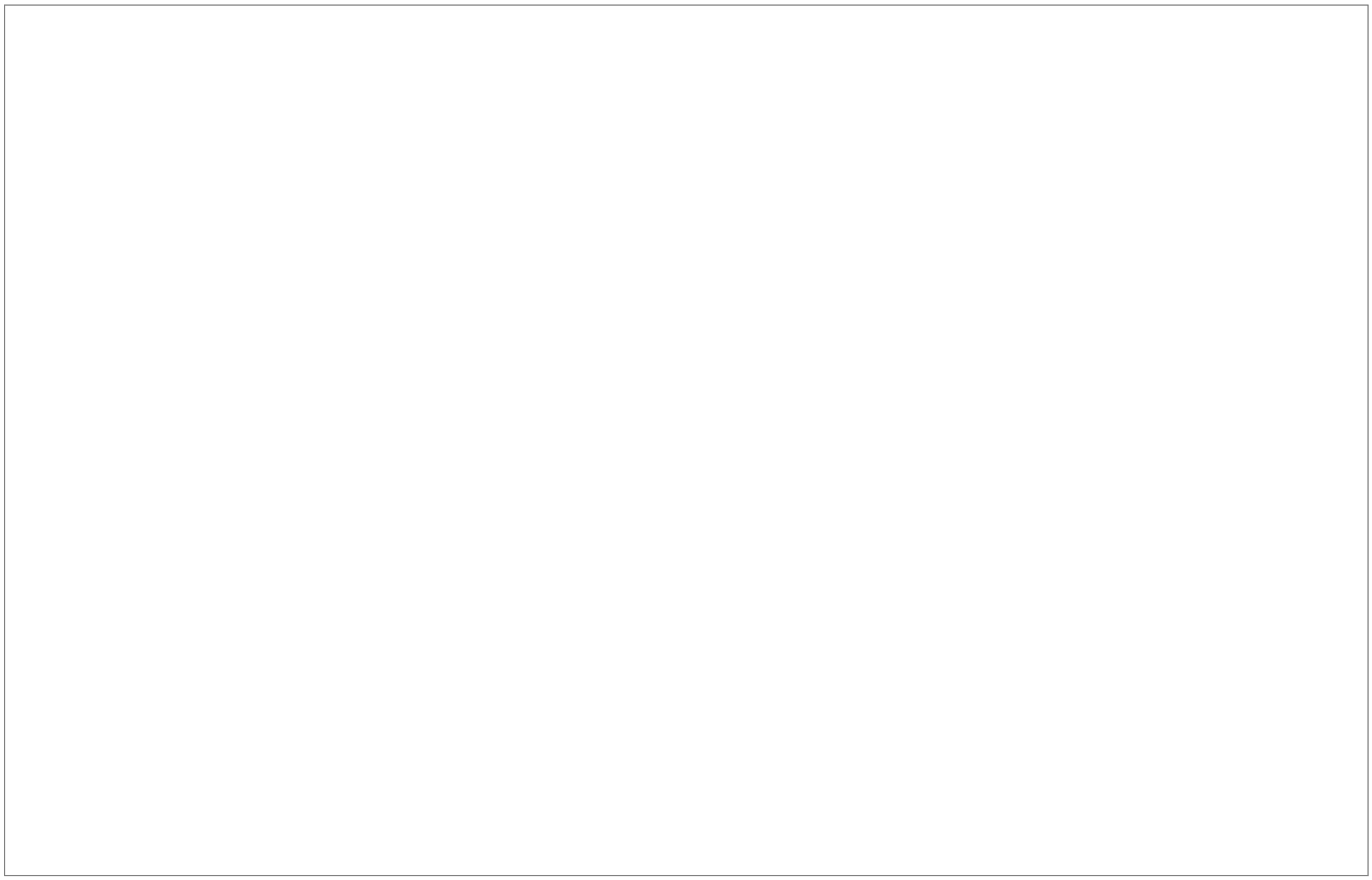

Figure 1. Display settings view and 2D View

- **Basic settings** layout: change the colour of the *Background*, *Selection* (*zoom window* rectangle) and *Highlighting*.
- **Pca** (Printed Circuit Board) layout: change the colour, filling visibility modes of the *Top* and *Bottom* side of the PCA.
- **Pcb** (Printed Circuit Board) layout: change the colour, filling visibility modes of the *Inter stratum*, PCB components in a particular stratum (layer).
  The PCB components and layers are presented in the table:
  - Component name (column header): *P* (pads), *R* (Traces), *V* (vias), *T* (text), *C* (copper), *F* (fiducial).
  - Stratum name (row header): *T S/M .006*, *T S/PASTE*, *1ST SIGNAL*, *dielectric 11*, *bot*, *B S/M .006*, *B S/PASTE*.

Changing colour

To change colour for a particular component:

- Click the first mouse button on a button with a colour square next to an applicable component. The **Color** dialog window appears (see Figure 2).

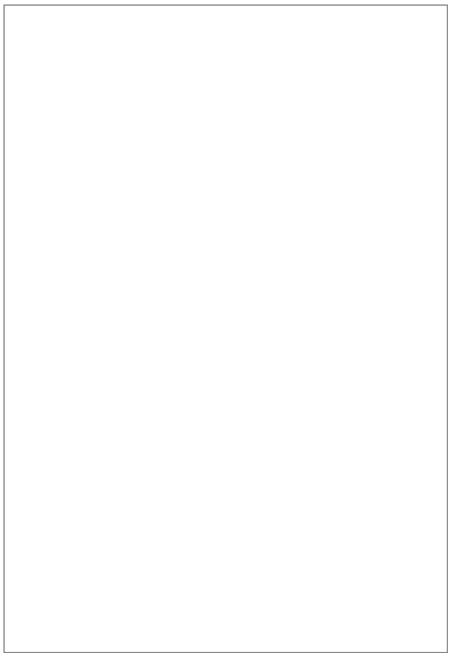

Figure 2. Color dialog window

- Click on an applicable colour in the **Basic color** or **Custom colors** layout.
  Or
- Define your custom colour by clicking on the **Define Custom Colors** button. The additional part of the dialog window appears (see Figure 3).

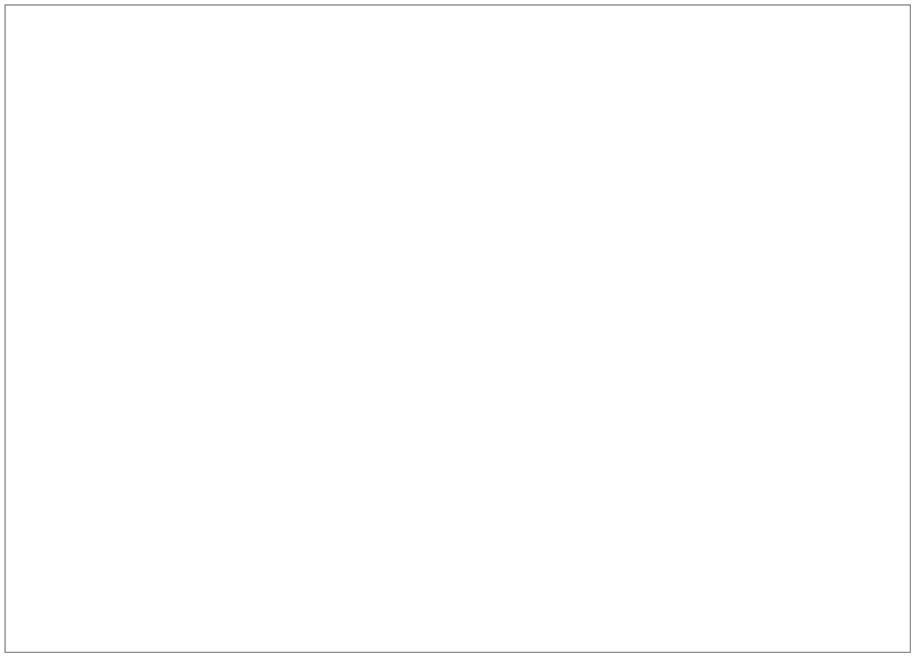

Figure 3. Color dialog window to add custom colour

  - Select a colour and click **Add to Custom Colors** button.
  - A square with a chosen colour appears in the **Custom colours** layout (on the left).
  - Click **OK** button.
- The colour of the selected component is changed.

The **Pcb layout** contains a table with a possibility to make the same colour for the component in all stratums or for all components in one stratum:

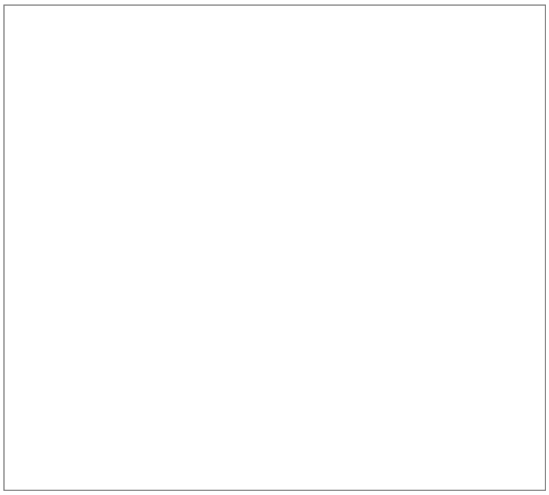

Figure 4. Table view

- Click the first mouse button on the button with the white square icon below the component (*P* (pads), *R* (Traces), *V* (vias), *T* (text), *C* (copper), *F* (fiducial)) to change a colour for a particular component in all stratums.
- Click the first mouse button on the button with the white square icon next to the stratum name (*T S/M .006*, *T S/PASTE*, *1ST SIGNAL*, *dielectric 11*, *bot*, *B S/M .006*, *B S/PASTE*) to change a colour for all components in a particular stratum.

Changing filling mode

A button icon next to a particular component indicates, whether the components is filled or outlined:

- - filled;
- - outlined.

To change a filling mode of a particular component click the second mouse button on a button with a colour square next to an applicable component.
You can change a filling mode for the component in all stratums or for all components in one stratum:

- Click the second mouse button on the button with the white square icon below the component (*P* (pads), *R* (Traces), *V* (vias), *T* (text), *C* (copper), *F* (fiducial)) to change a filling for a particular component in all stratums.
- Click the second mouse button on the button with the white square icon next to the stratum name (*T S/M .006*, *T S/PASTE*, *1ST SIGNAL*, *dielectric 11*, *bot*, *B S/M .006*, *B S/PASTE*) to change a filling mode for all components in a particular stratum.

Changing visibility

The icon button indicates, that the current component is not visible in the **2D View**.
To change visibility of a particular component click the third mouse button on a button next to an applicable component.

To top

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9.2.1. Displaying Geometry in 3D View

The **3D View** is driven by the **Parts** view. As long as no part is selected in the **Parts** view, nothing is displayed in the **3D View**. Also nothing is displayed if no 3D geometry is defined for the selected part. IDA-STEP is configured in such a way, that a part from the **Parts** view is automatically selected and displayed in the **3D View**.

STEP data is defined by different kinds of curves (line, circle, spline, etc.) and surfaces (plane, sphere, toroidal, etc.). To be able to display this data by computer, the curves have to be interpolated by short line segments and the surfaces have to be emulated by small triangles. This process,called triangulation or tessellation, is time consuming. During this process a dialog with a process bar shows up (see Figure 1). To avoid repeated tessellations the result is stored together with it's originating STEP data in the XIM file. That is a reason why the Viewer Pro feature, when opening a STEP file is asking on whether or not to create a corresponding XIM file. The Viewer Basic alone will do the tessellation always anew.

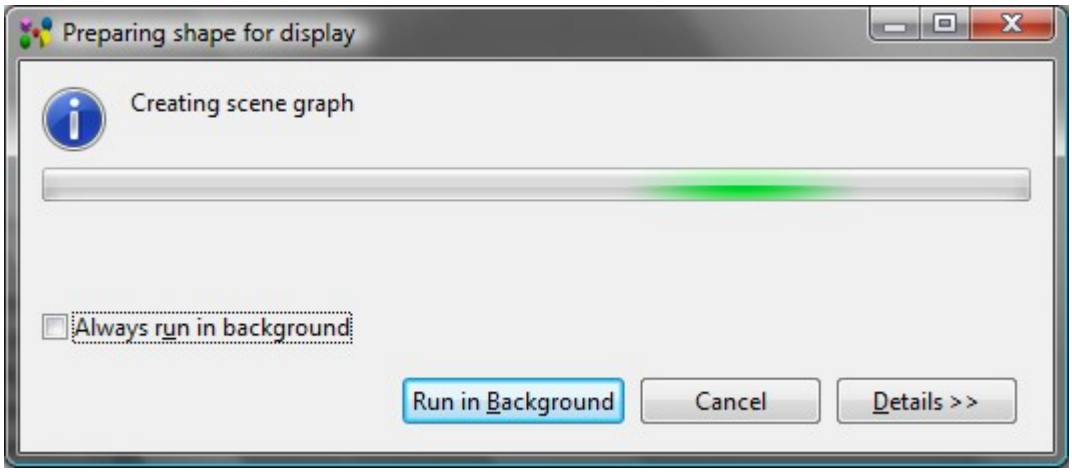

Figure 1. Preparing shape for display window with a process bar

Follow the routine to open 3D geometry in the **3D View** page:

- Open the **Parts** view and **3D View** in the workbench by using main menu **Window -> Show View ->...** .
- Open a particular STEP file, containing 3D data.

Or

- Select the **Parts 3D** perspective by selecting in the main menu **Window -> Open Perspective -> Other -> Parts 3D**.

- Click on an applicable part version element in the **Parts** view to display it in the **3D View** page (see Figure 2).

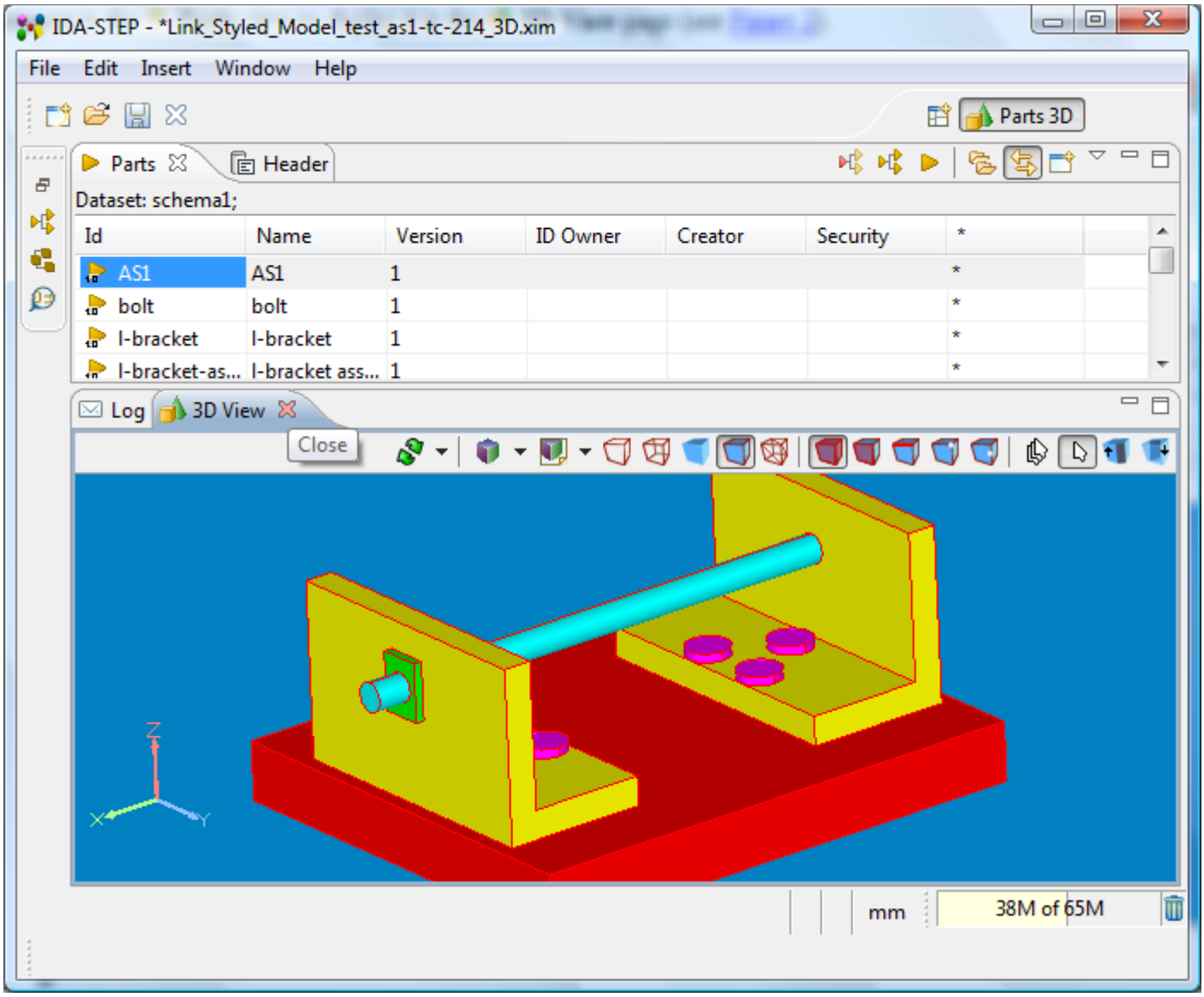

Figure 2. Selected part version in Parts view is displayed in 3D view

To top

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9.2.2. Overview of 3D View

The **3D View** is able to display 3D geometry contained in geometry models, styling models and construction geometry. It provides controls for:

- zooming, rotation and panning;
- predefined orientations;
- predefined styling: wireframe, solid, solid with wireframe, triangle style;
- front and back clipping;
- selecting and highlighting of points, edge/curves, surfaces and components;
- merging surfaces (available with Viewer Pro);
- dynamic selection and measurement of geometric objects (available with Viewer Pro).

By default the **3D View** synchronizes it's selected elements with **Geometry Details**, **Part Details** and **GD&T** view.

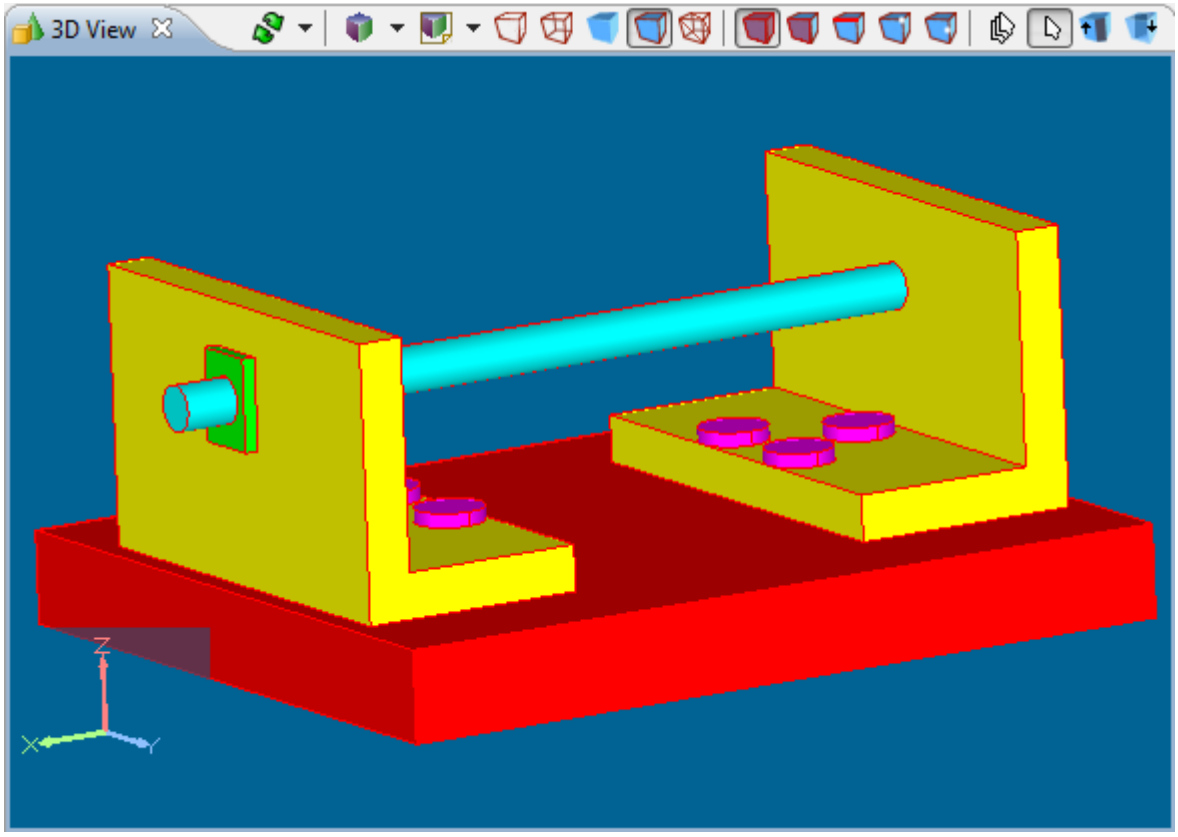


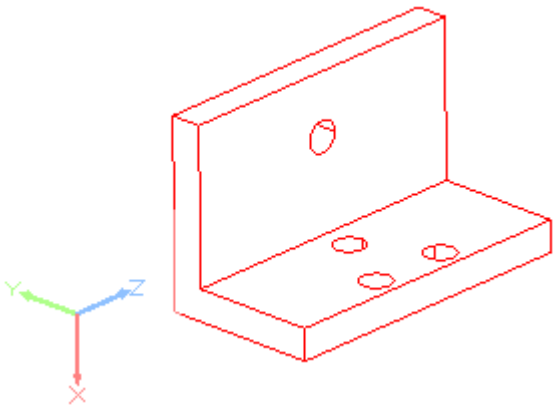
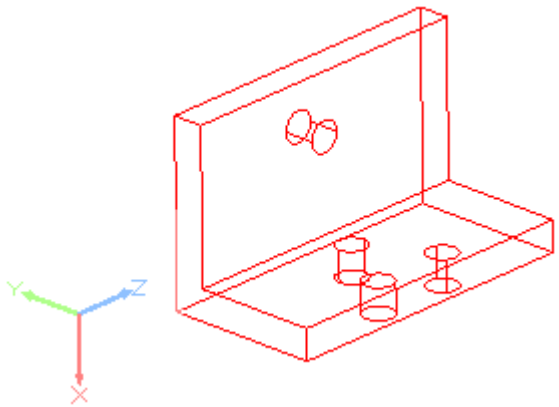
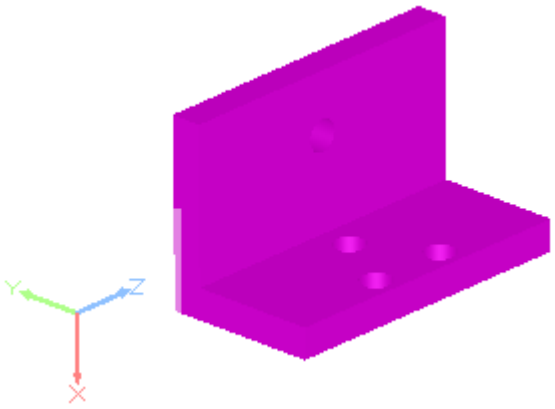
Figure 1. 3D viewer with a 3D geometry

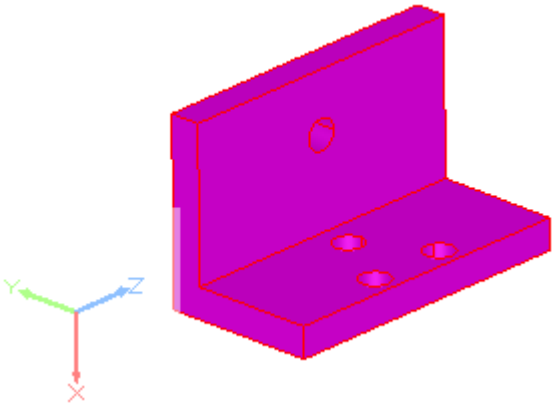
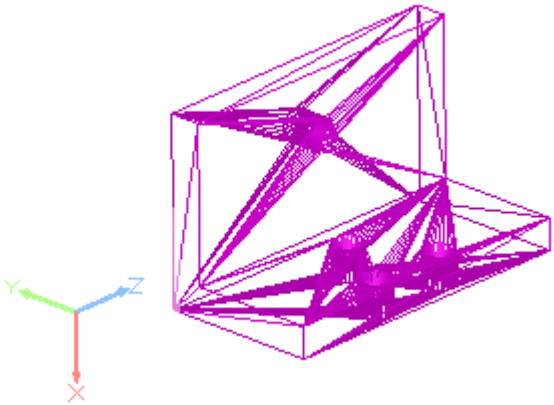
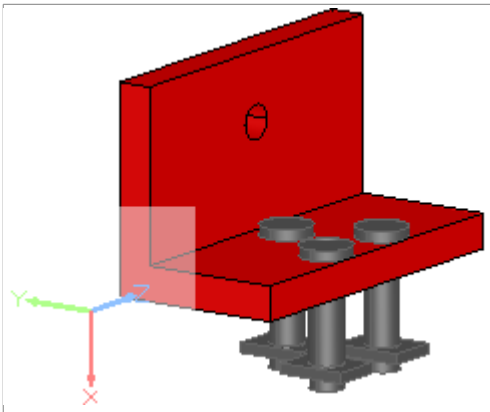
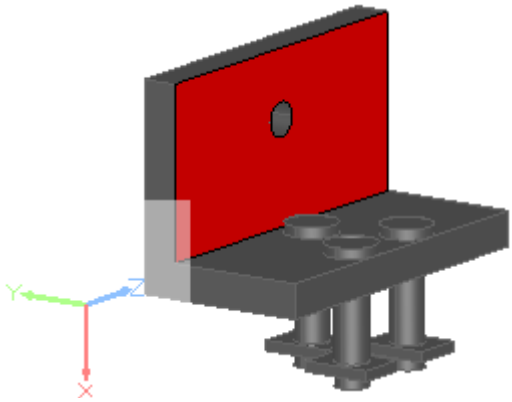
The **3D View** has it's own tool bar and context sensitive menu.

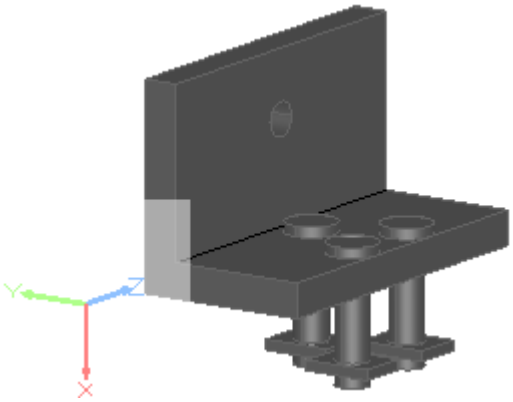
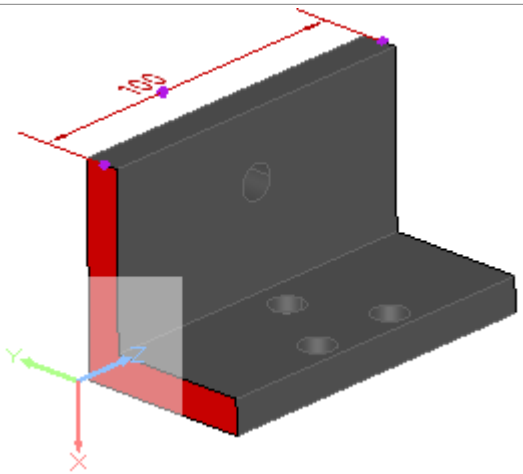
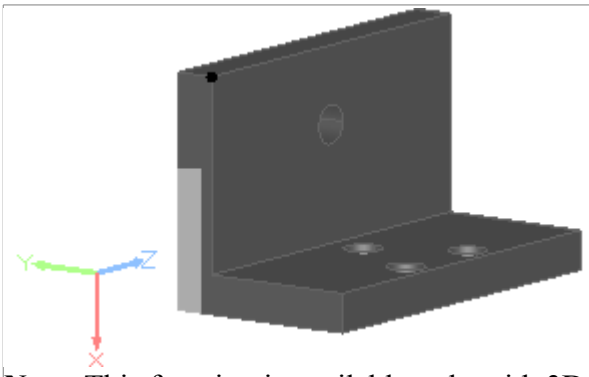
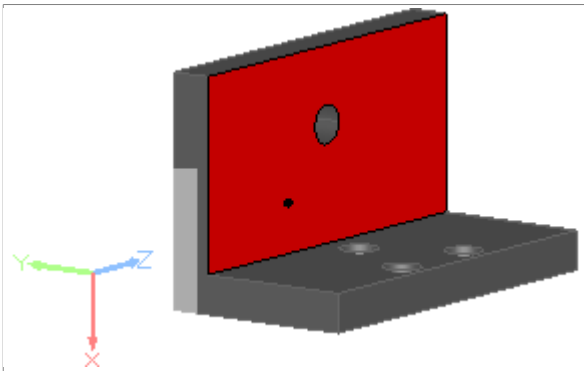
Toolbar

Tool bar buttons are provided for changing appearance of 3D geometry, selection of modes, styling, moving, shifting, picking, clipping and etc.:

Button icon	Description	
	Predefined Orientation (Front): • Overview • Top • Front • Left • Right • Back • Bottom • SW Isometric • SE Isometric • NE Isometric • NW Isometric	
	Style/Annotation Model	• *Unselect active style/annotation model* - press the button; • *Activate style/annotation model/saved view* - select model/saved view in the drop down list; • *Ignore styling* - select the **Ignore styling** item in the list;

		• *Save current 3D view* - select the **Save current 3D view** item; • *Change active and bring to front annotation plane* - select an annotation plane item in the drop down list to make it active and bring to front the underlying annotation plane so that it is aligned in the right reading position to the actual camera position. **Note.** This function is available only with 3D Viewer Pro feature.
	Styling Preference	Select default styling schema in the drop down list. See also Preferences - > 3D View -> Display Styles
	Wireframe with Hidden Line Removal Style	
	Wireframe Style	
	Solid Style	

	Solid with Wireframe Style	
	Triangle Style	
	Component/Model Selection Mode	
	Face/Surface Selection Mode	 **Note.** This function is available only with 3D Viewer Pro feature.

	Edge/Curve Selection Mode	 **Note.** This function is available only with 3D Viewer Pro feature.
	Annotation Selection Mode	 **Note.** This function is available only with GD&T Editor feature.
	Vertex/Point Selection Mode	 **Note.** This function is available only with 3D Viewer Pro feature.
	Point on Edge or Face Selection Mode	 **Note.** This function is available only with 3D Viewer Pro feature.
	Dynamic Selection Mode	• highlighting - flying by with a mouse pointer • rotating - mouse drag, • zooming - mouse drag up/down with **Ctrl** key pressed or rolling

		forward/backward with mouse wheel, • panning - mouse drag with **Alt** key or mouse wheel pressed. **Note.** This function is available only with 3D Viewer Pro feature.
	Normal Selection Mode	• select/highlight - mouse click, • rotating - mouse drag, • zooming - mouse drag up/down with **Ctrl** key pressed or rolling forward/backward with mouse wheel, • panning - mouse drag with **Alt** key or mouse wheel pressed.
	Clip front mode	Mouse drag up/down
	Clip back mode	Mouse drag up/down

Context sensitive menu

The context sensitive menu groups functions, which are available in the **3D View** (see Figure 2):

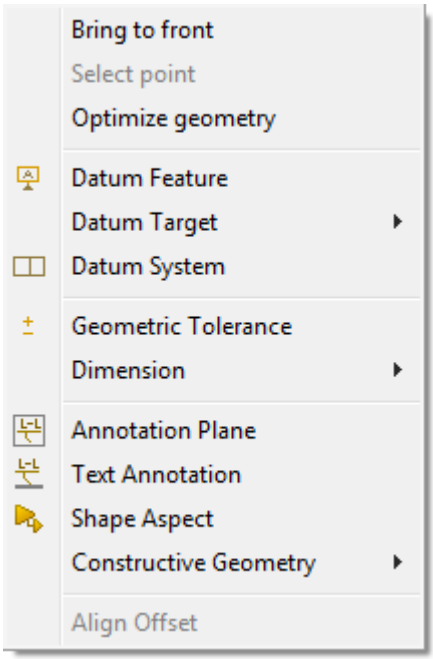


Figure 2. Context sensitive menu in 3D View

Functions, available with 3D Viewer Pro feature:

- **Bring to front** - to change orientation of the geometry object according the selected surface.
- **Select point** - to set a point on the geometry object.
- **Optimize geometry** - combine neighbouring faces (cylinders, cones, spheres) when their underlying surface is the same or equivalent.

Functions, available with GD&T Editor feature:

- **Datum Feature** - to create an actual surface component to establish a datum.
- **Datum Target** - to create a specified point, line or area on a part that is used in establishing a datum. This menu item contains a sub menu, where available datum target types are listed: **Point Datum Target**, **Line Datum Target**, **Circle Datum Target**, **Circular Curve**

Datum Target and **Rectangle Datum Target** (see Figure 3).

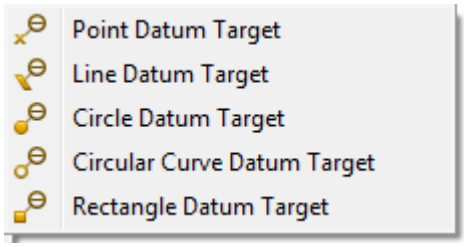


Figure 3. Datum Target sub menu items

- **Datum System** - to create a group of one or more separate datums used as a combined reference for a toleranced feature.
- **Geometric Tolerance** - to create a geometrical tolerance for selected geometry component.
- **Dimension** - to add measurement annotation to a geometry object. This menu contains a sub menu, where available dimensions are listed (see Figure 4):
  - **Angle** - to create an angular dimension annotation.
  - **Size** - to create a radius, a diameter or a length dimension annotation.
  - **Distance** - to create a distance dimension annotation.

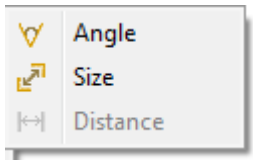


Figure 4. Dimension sub menu items

- **Annotation Plane** - to create a conceptual plane, which contains annotations.
- **Text Annotation** - to create a 3D annotation with a free text, one or several line.
- **Shape Aspect** - to create an element of the shape of a product.
- **Supplemental Geometry** - to create supplemental geometry elements. This menu contains a sub menu, where available supplemental geometry elements are listed: **Apex**, **Symmetry Axis**, **Symmetry Plane**, **Symmetry Point**, **Common Line**, **Common Plane**, **Intersection**, **Offset**, **Perpendicular Line**, **Perpendicular Plane**, **Extension**, **Tangent Plane**, **Line** and **Plane** (see Figure 5). **Note.** There are enabled only those menu items, which represent functions available for the current selection.

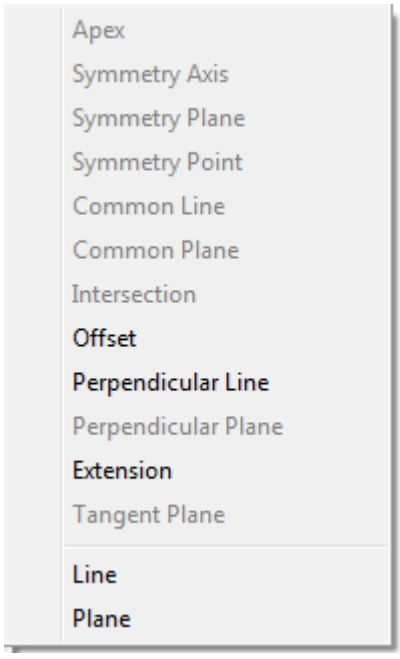


Figure 5. Supplemental Geometry sub menu items

- **Align Offset** - to align the offset of annotations on the same annotation plan.

To top

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9.2.3. Predefined Orientations

The predefined orientations provide an easy way to set the orientation for the displayed geometry to precise orientation, panning and zooming values. The predefined orthographic and isometric orientations are listed in the drop down list next to the **Reset to Front** button in the **3D View**, they are:

- **Overview**: switch back from a zoomed-in state to an overview without changing the orientation.
- Orthographic orientations: **Top** (see Figure 1), **Front**, **Left**, **Right**, **Back**, and **Bottom**.
- Isometric orientations: **SW** (southwest) Isometric, **SE** (southeast) Isometric (see Figure 2), **NE** (northeast) Isometric, and **NW** (northwest) Isometric.
- Relative 90 degrees rotation around axes:
  - **RX +90** - 90 degrees clockwise rotation around X axis,
  - **RX -90** - 90 degrees counter-clockwise rotation around X axis,
  - **RY +90** - 90 degrees clockwise rotation around Y axis,
  - **RY -90** - 90 degrees counter-clockwise rotation around Y axis,
  - **RZ +90** - 90 degrees clockwise rotation around Z axis,
  - **RZ -90** - 90 degrees counter-clockwise rotation around Z axis,

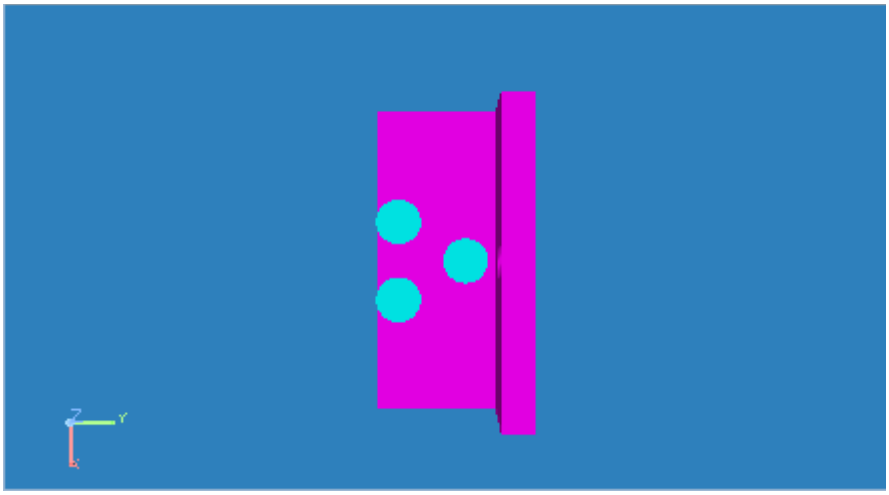

Figure 1. Top orientated object view

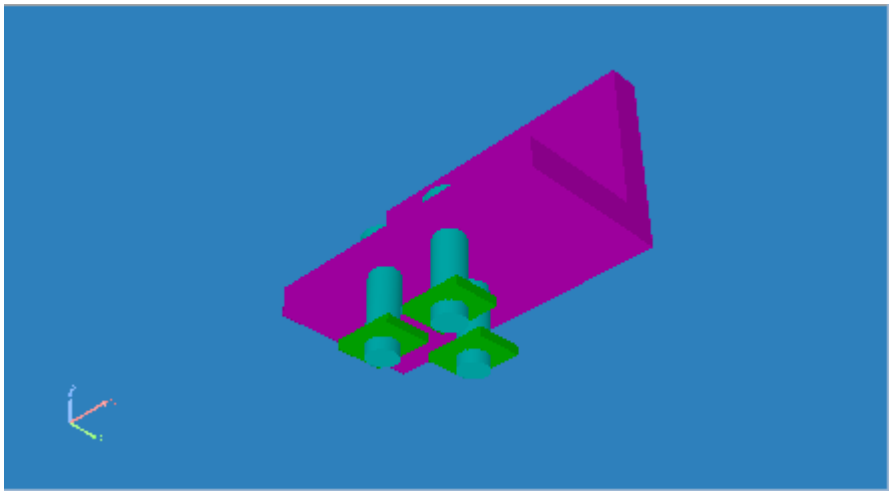

Figure 2. SE (southeast) isometric oriented object view

Click **Reset to Front** button to reset default settings of the current 3D object orientation, size and position in the **3D View**, if rotation, zooming, panning or clipping was performed.

See also Preferences -> 3D View -> Front side orientation.

To top

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9.2.4. Save Current 3D View

The rotated and magnified geometry model view, currently active in the current **3D View**, can be saved and used for reviewing in the future.
Follow the routine to save the current 3D view:

- Activate the **3D View**.
- Click on the arrow next to the **Styled Model** button.
- Select **Save current 3D view...** item in the appeared drop down list (see Figure 1).

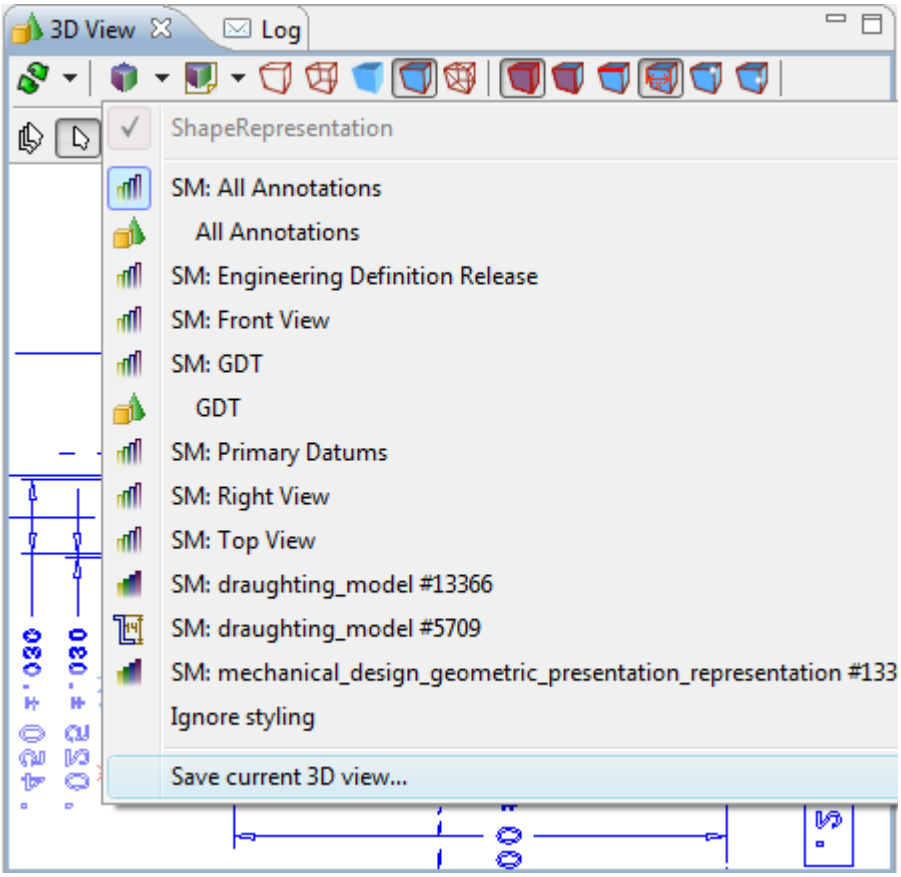

Figure 1. 3D View page with Styled model drop down list, selected Save current 3D view... menu item

- The **Creation of the saved view** dialogue window appears (see Figure 2):

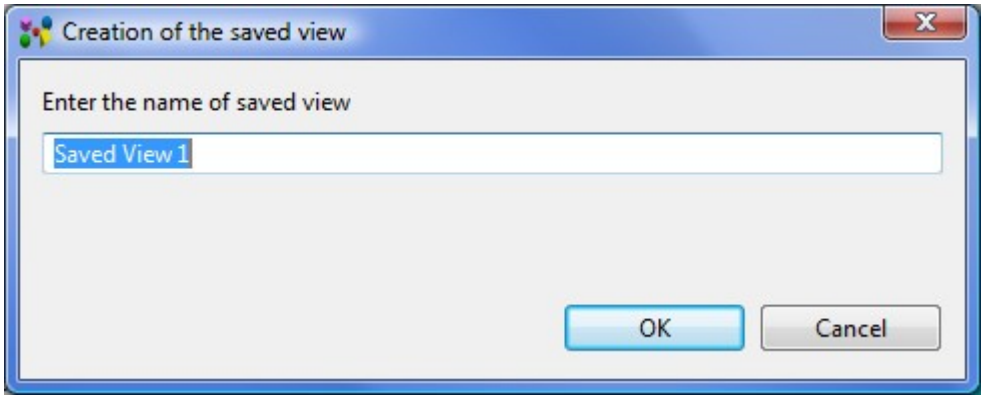

Figure 2. Creation of the saved view dialogue window

- Enter the name of the saved view into the text field.
  The prompted name is *Saved View 1*. You can edit this text or enter a new one. If there already exist the saved view with the same name in the parent styled model, a view is overwritten with the last one.
- Click **OK** button.

- The saved view is located under the currently active styling model (see Figure 3). **Note.** The parent styled model is activated together with the saved view (both of them are highlighted in the context sensitive menu).

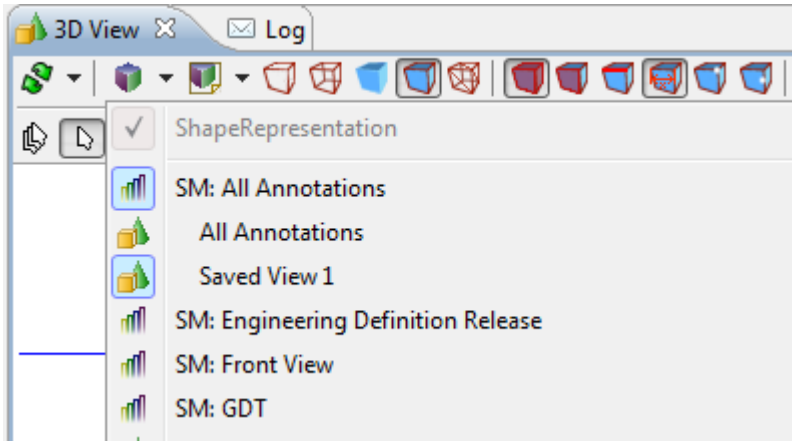

Figure 3. Created Saved View 1 in the drop down list

To activate a saved view, select an applicable item in the **Styled Model** drop down list. The selected saved view is displayed with the check mark or switched *On* icon in front of the saved view name.
Note. The parent styled model is activated together with the selected saved view.
If the rotation or zooming of the geometry model is changed in the **3D View**, the check mark in front of the previously selected saved view in the **Styled Model** drop down list disappears or the switched *On* icon becomes the normal state icon.

How to rename and remove an applicable saved view, see Saved Views (if **Geometry Details** view is available).

To top

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9.2.5. Appearance Style

Adjust the 3D geometry representation appearance in the **3D View**. Use one of the button in the **3D View** tool bar (see Figure 1).

Figure 1. Tool bar buttons to change appearance of the 3D geometry representation

In case all appearance mode buttons are *Off*, the **3D View** area is empty, unless some Styling Model is previously selected, then the styled items are displayed.

Note. When starting IDA-STEP anew the default **Solid With Wireframe Syle** is applied.

Wireframe with Hidden Line Removal

- Click button on the tool bar to view a wireframed 3D object, where lines (or segments of lines) covered by surfaces are not drawn (see Figure 2).

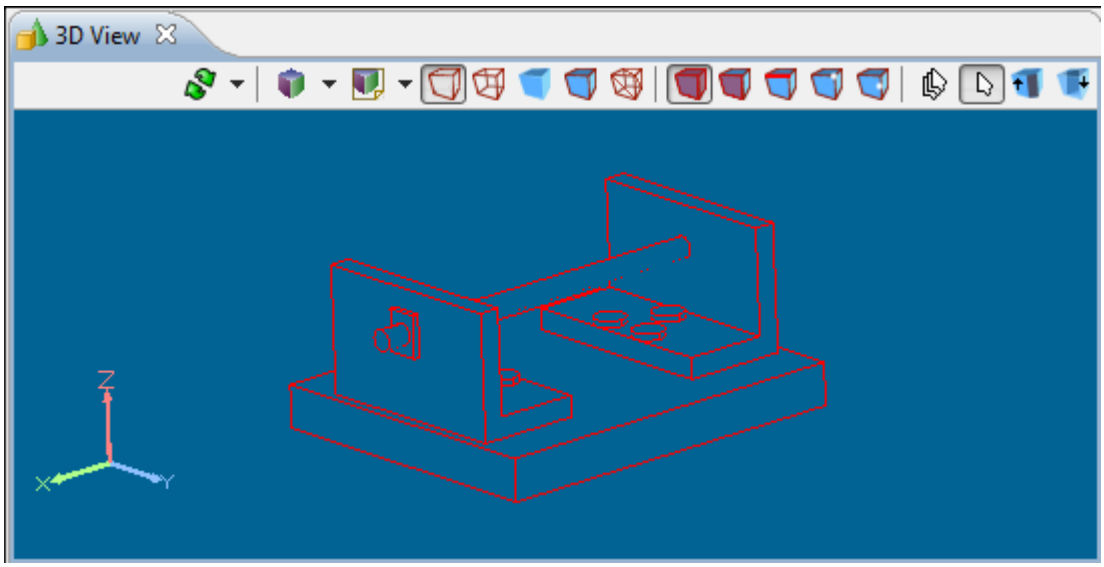

Figure 2. Wire frame without hidden lines view

To top

Wireframe

- Click button on the tool bar to view wire-framed 3D object view.
  The "**Wireframe**" is a graphic view, where 3D body is displayed as a "red" wire frame, consisting of curves only (see Figure 3).

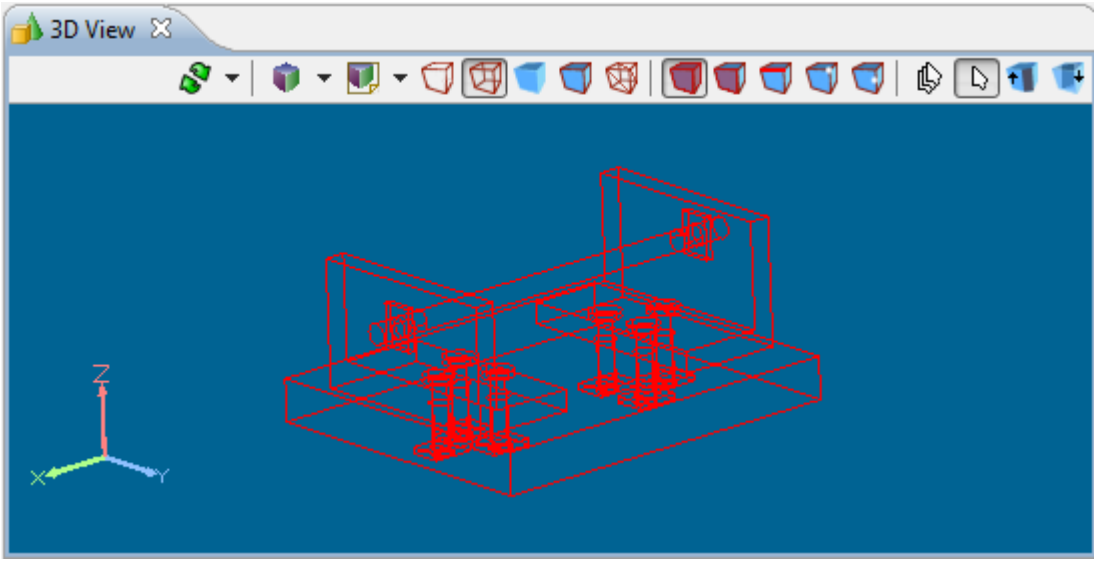

Figure 3. Wireframe view of 3D object

To top

Solid

- Click button to view a solid 3D object view.
  The "**Solid**" is a graphic view, where 3D body is displayed as a coloured solid (see Figure 4).

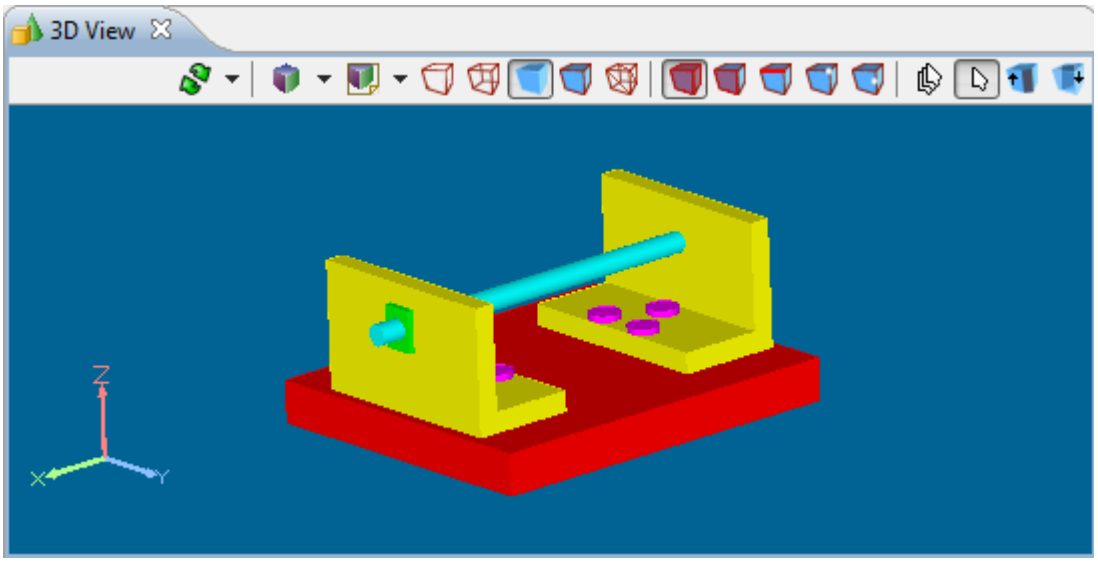

Figure 4. Solid view of 3D object

To top

Solid with Wireframe

- Click button to view a solid 3D object view with wire frame lines.
  "**Solid with Wireframe**" is a combined graphic view, where 3D body is displayed as both solid and wire frame (see Figure 5).

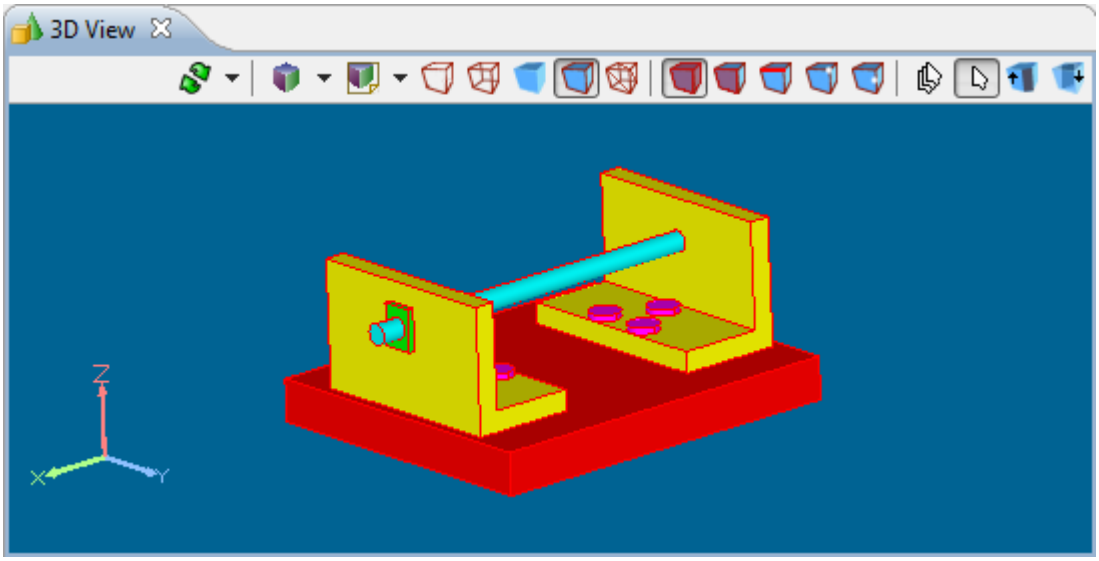

Figure 5. Solid and wireframe view of 3D object

To top

Triangle

- Click button to view a 3D object, where surfaces are displayed as wire frame triangles (see Figure 6).

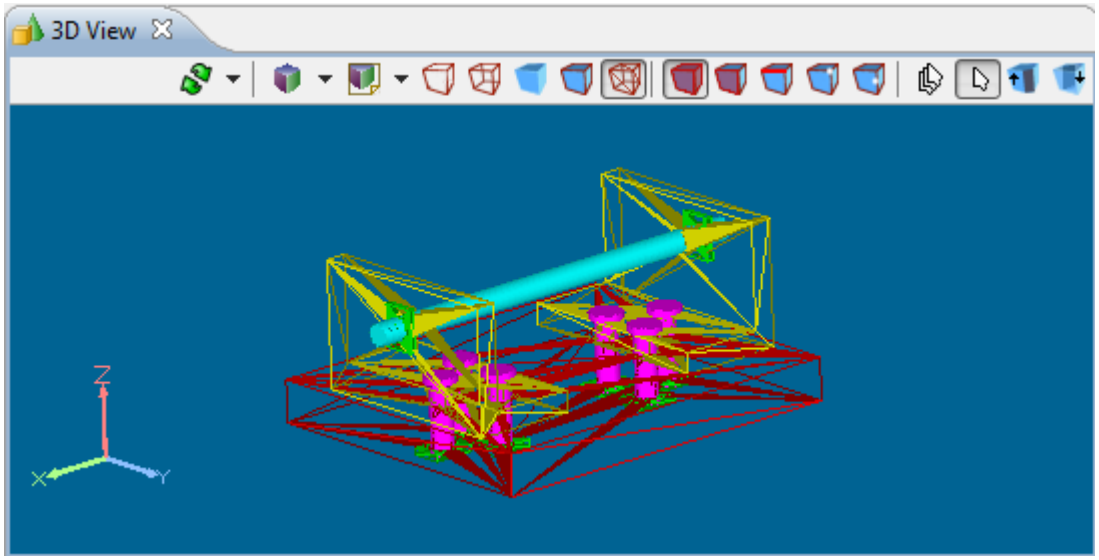

Figure 6. 3D object where surfaces shown as triangles

To top

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9.2.6. Style/Annotation Model

By default a geometry model in the **3D View** is displayed by using default styling schema, which is defined in the 3D View -> Display Styles preferences and currently selected in the **Styling Preference** list.

Style/Annotation Model

There can be several styling models created in STEP file data. There are two different kind of information

these models can provide:

- styling (e.g. assigning colors),
- defining annotations, typically draughting annotations.

To change the styling model for the 3D geometry model in the **3D View**, use the **Styled Model** drop down list. The table below presents the icons in the drop down list, tool tips of the icon button, when particular model is selected and the content of the styling model.

Menu icon	Button Tool tip	Content
	Style Model: <name>	style only
	Annotation Model: <name>	annotations only
	Style + Annotation Model: <name>	style and annotations
		no style and/or annotations
	-no-	the styling is disabled

The Annotation or Style + Annotation Model has associated annotation planes. The **Styled Model** drop down list displays the annotation planes, which are available for the selected Annotation or Style + Annotation Model (see Figure 1). The active annotation plane is highlighted. Please find below the description, how to change active style/annotation model and annotation plane.

Change active Style/Annotation Model

Follow the routine to change an active styling model of a shape representation in the **3D View**:

- Activate the **3D View** and press an arrow next to the icon button.
- Select an applicable associated styling model in the appeared drop down list (see Figure 1).
  Note. There is **ShapeRepresentation** menu item in the drop down list always checked and disabled for selection. This styling model has to be present.
  The number of styling models in the drop down list is limited to 10 items. If more styling models are present, the **More** menu item appears at the end of the list (see Figure 1). The **More** menu item opens the **Styling Models** window with a tree of styling models together with saved views (second level nodes) (see Figure 2). Double click on the node (or single click + click on the button **Select**) activates respective styling model or saved view.
  Note. The list of 10 styling models (in the drop down list) always includes the currently active styling model.

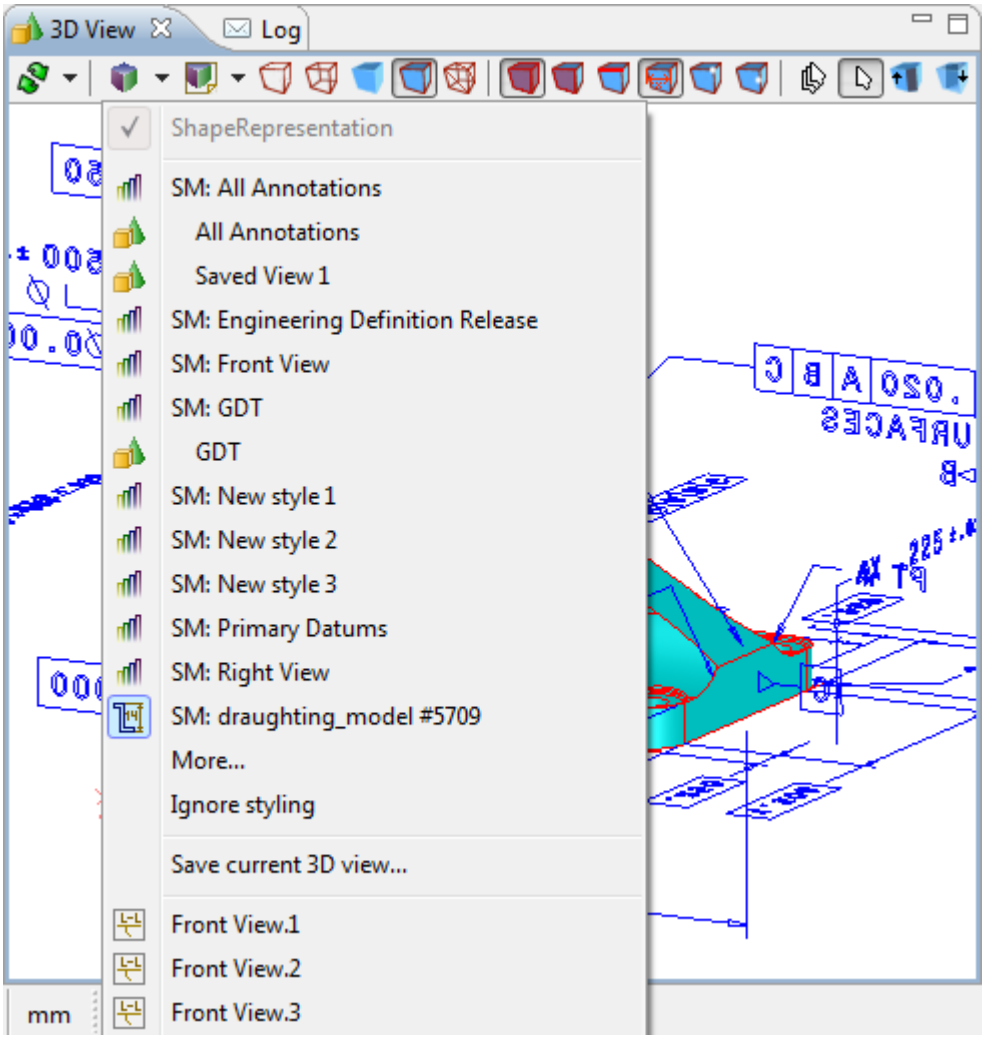

Figure 1. 3D View, list of associated models (active style is highlighted (icon) in the list), Ignore styling item, Save current 3D view item and a list of available annotation planes ('Front View.1' and etc.) for the currently active style and annotation model.

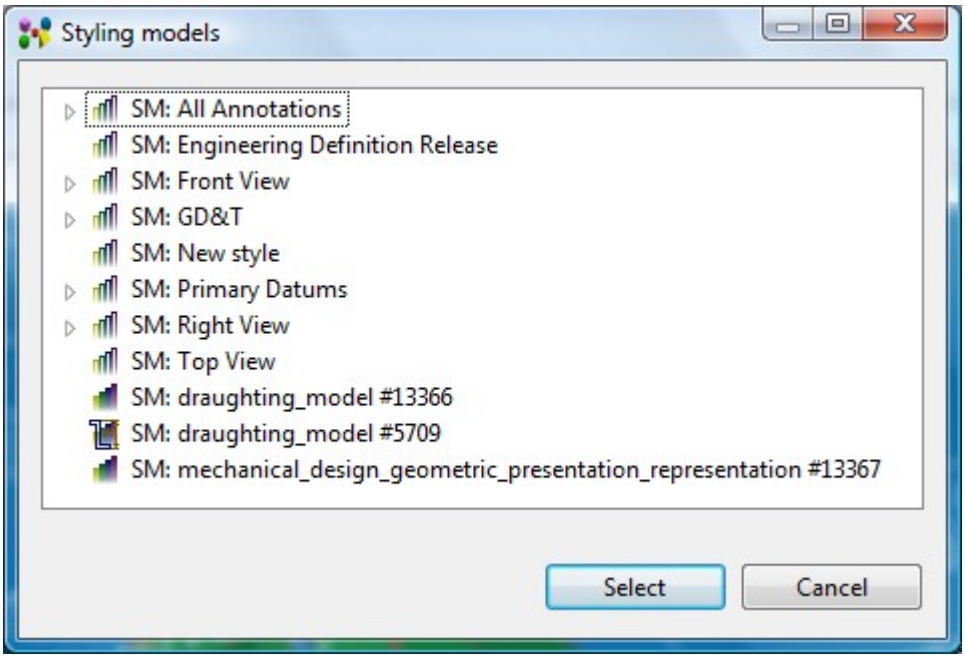

Figure 2. Styling models window, tree of styling models and saved views

The selected styling model is made active for the 3D geometry in the **3D View**.

After some styling model is selected, the icon drop down list is extended with one more menu item, i.e. **Ignore styling**. After this menu item is selected, the style information is ignored and now the **3D View** displays geometry model and annotations (if any) without style. The default styling schema is used,

which is currently active (selected in the **Styling Preference** list).

To disable the active styling model, press icon button.

Change active and bring to front Annotation Plane

The annotations (if any) of the current 3D view are listed at the end of the menu. Any press on an annotation plane make it always both *Active* and *Bring it to front* (bring to front the underlying annotation plane so that it is aligned in the right reading position to the actual camera position).

Follow the routine to change active annotation plane nd bring to front in the **3D View**:

- Activate the **3D View** and press an arrow next to the icon button.
- Select an applicable annotation plane in the appeared drop down list (end of the list). **Note.** The active Style/Annotation Model should have associated annotation planes.
  Note. Any press on an annotation plane make it always both active and bring it to front. Pressing the second time on already active annotation plane, just bring it to front.

In case a new annotation plane is created in the **3D View**, it becomes the currently active annotation plane.

Styling Preferences

The default active styling schema for the geometry model is set in the preferences 3D View -> Display Styles page. The **Styling Preference** in the **3D View** contains a drop down list with available styling schemas. A user can select an applicable styling schema and make it active directly in the **3D View**, without calling preference page. The active schema is marked with a check mark in the drop down list.
Note. After a styling schema is selected, it is made as an active schema in preferences 3D View -> Display Styles page.

To top

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9.2.7. Selection Modes

The *Selection Mode* determines which scene elements can be selected and manipulated at a given time. There are some object selection modes available in IDA-STEP **3D View**: normal, dynamic, component, face, edge and vertex selection mode. An applicable selection mode can be switched *On* by pressing a particular button in the **3D View** tool bar (see Figure 1).

Figure 1. Buttons to switch selection mode

When starting IDA-STEP anew and displaying STEP file in **3D View** the default selection modes are active:

- **Component/Model Selection Mode** for assemblies and
- **Face/Surface Selection Mode** for a single piece part.

The user has a possibility to switch *On* several selection modes to be available at once. For example: **Face/Surface** + **Edge/Curve** + **Annotation** + **Vertex/Point** - user can select everything on the fly with a kind of snapping for vertexes, edges and faces.
Note. To switch *Off* a selection mode - unpress the particular button.

Component/Model

The **Component/Model** selection mode lets you select a certain component of an object.

To top

Face/Surface

The **Face/Surface** selection mode lets you select the individual faces (also known as polygons) which comprise an object.

To top

Edge/Curve

The **Edge/Curve** selection mode lets you select individual edges/curves. Edge/curves are used to define boundaries between neighbouring surfaces or for annotation purposes.

To top

Annotation

The **Annotation** selection mode lets you select all types of annotations.

To top

Vertex/Point

The **Vertex/Point** selection mode lets you select the individual vertices (also known as points or control points) which define the faces and edges of a model.

To top

Point on Face or Edge

The **Point on Face or Edge** selection mode lets you select a point on a face or edge of a geometry:

- to select a point on a face, the **Face /Surface** selection mode should to be tuned *On*;
- to select a point on an edge, the **Edge/Curve** selection mode should be turned *On*.

  Note. The above functionality can be achieved if the **Component/Model** selection mode is turned *On*, but more convenient is to use the appropriate selection modes (face or edge).

If the **Point on Face or Edge** selection mode is *On* together with one of selection modes mentioned

above - both things (component, face, edge + point) are shown and highlighted.
The selection of a point can be executed by:

- Left mouse single click, or
- Right click and selecting **Select point** menu item in the appeared context sensitive menu.

After pointing with a mouse on a solid geometry object (face, annotation plane, filled terminator symbol, etc.), the **Bring to front** context sensitive menu item appears. By selecting this menu item, the geometry object will be oriented according the selected surface.

To top

Normal

The **Normal** selection mode lets you select an object component by clicking on it with a mouse button.

Cross-highlighting

When selecting some object in the **3D View**, the associated data are highlighted in the **Geometry Details** and **GD&T** views.
Lets' examine, how the **Geometry Details** and **GD&T** views are affected after the **Annotation Selection Mode** is switched *On* and some annotation is selected in the **3D View** (see Figure 2):

1. Ensure the **Normal** and **Annotation Selection Mode** are switched *On*.
2. Pick with a mouse on an applicable annotation in the **3D View**. The particular object is highlighted together with the associated shape feature.
3. The node, which represents the selected object, is highlighted in the **Geometry Details** view.
4. The GD&T objects, which are related to the selected annotation, are highlighted in the **GD&T** view.

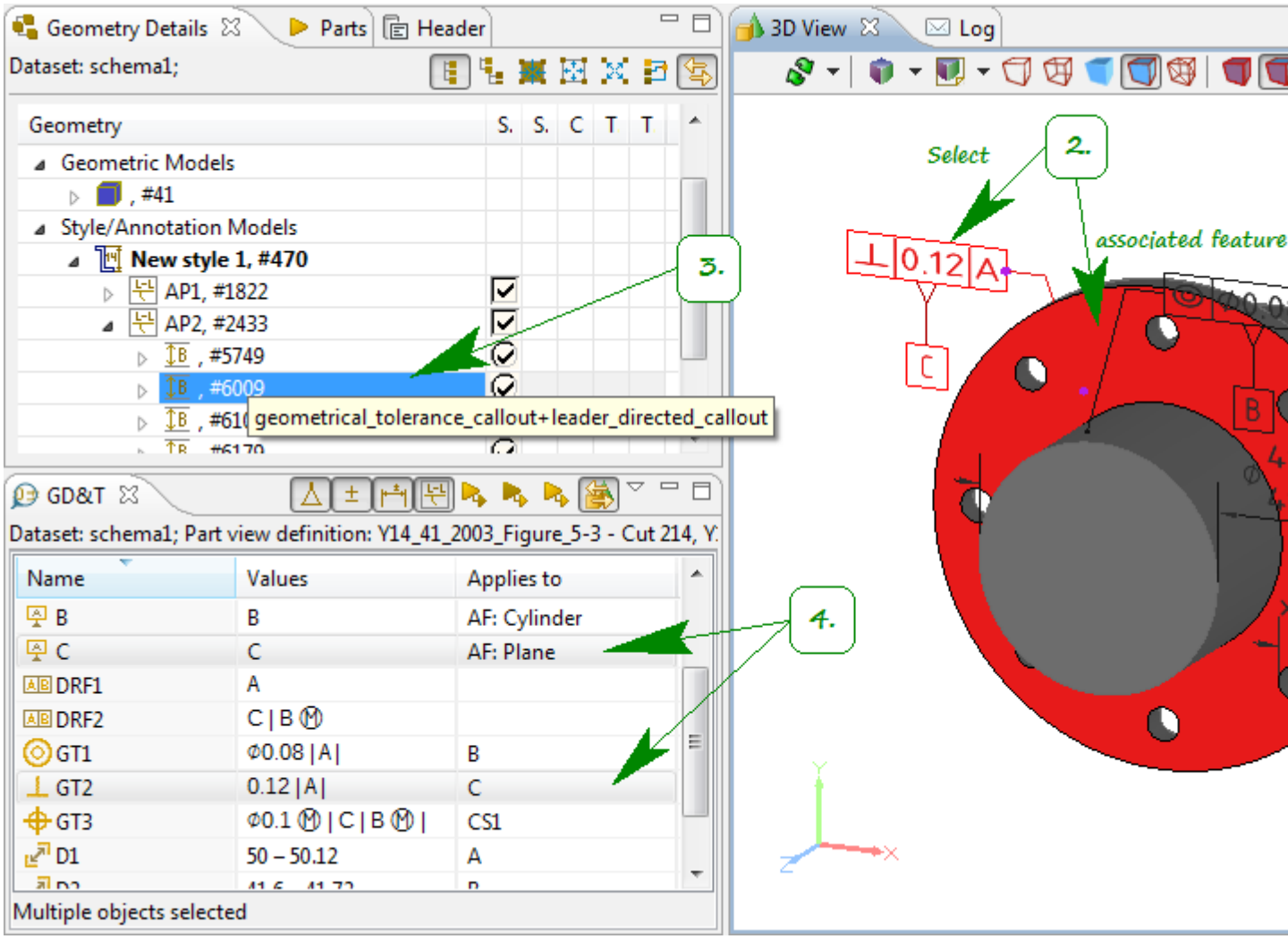

Figure 2. Selected annotation in 3D View with cross-highlighted data in 3D View (associated feature), Geometry Details and GD&T view

To top

Dynamic

The **Dynamic** selection mode lets you to select an object without clicking on it. A particular object is selected while dragging with a released mouse button.
Note. In case an annotation is selected in the **3D View**, the associated shape feature is highlighted at the same time.

To top

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9.2.8. View Control

You can select/highlight, pan, zoom in/out and rotate a 3D object in the **3D View**. To control a 3D object you have to switch *On* the **Normal Selection** or **Dynamic Selection** modes by pressing a particular button in the **3D View** tool bar.

Select/Highlight

To select/highlight an object in the **3D View**, click on an applicable object. The object is highlighted (see Figure 1). Which scene elements can be selected and manipulated at a given time determines the Selection Mode.

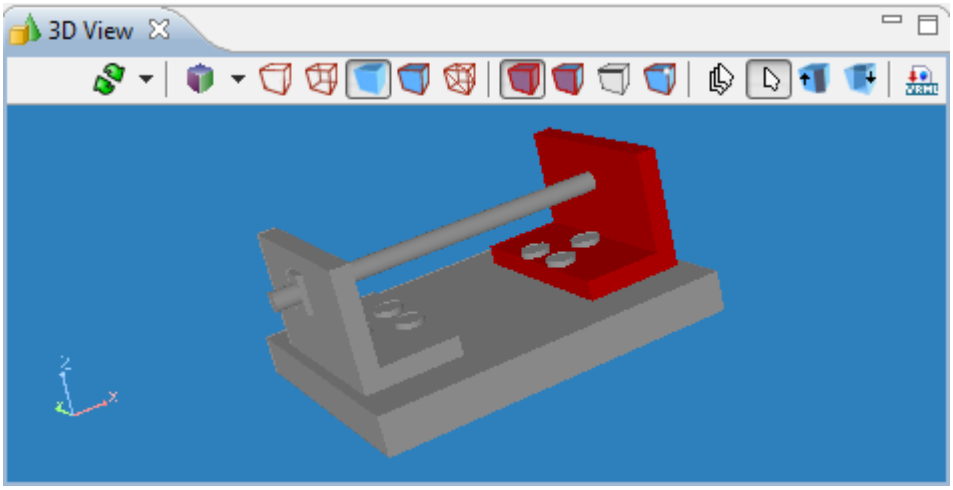

Figure 1. Selected/highlighted object (red coloured)

There is another way to highlight an applicable object in the **3D View**:

- Select an applicable object in the **Part Details** view or in the **Geometry Details** view (if available).
- The selected component is highlighted in the **3D View** (see Figure 2).

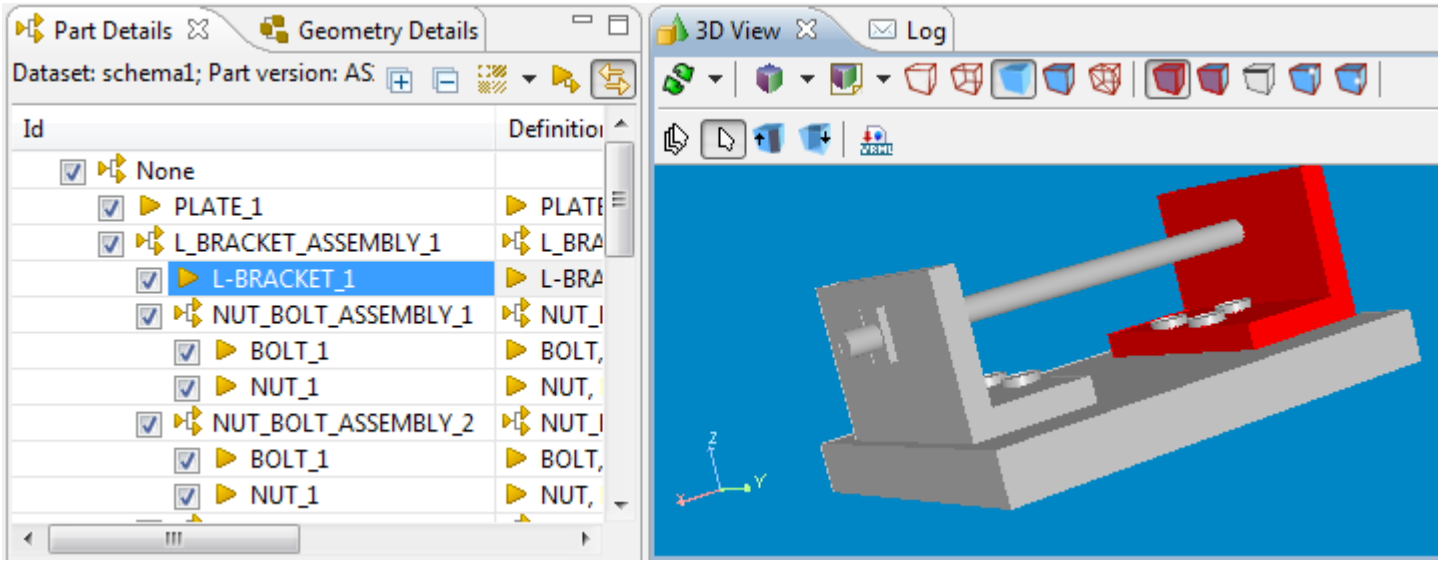

Figure 2. Selected object (in Part Details view) is highlighted in 3D view and vice versa

Note. To highlight annotation planes, annotation occurrences, construction geometry planes in the **3D View**, use **Edge/Curve Selection Mode**.

To top

Rotate

To move a 3D object into a circular motion around a particular axis:

- Click and hold with any mouse button on the **3D View** area.

- Drag to any direction to rotate an object. The object rotates according to the mouse movement.

To top

Zoom In/Out

To enlarge or reduce a 3D object in the **3D View**:

- Move the mouse cursor over the feature you are going to zoom in/out.
- Rotate the mouse wheel: backward - to enlarge (zoom in) the object; forward - to reduce (zoom out) the object.

Or

- Press **CTRL** key and click on the the **3D View** area.
  - Hold and drag upwards to reduce (zoom out) the object.
    Or
  - Hold and drag downwards to enlarge (zoom in) the object.

The object in the **3D View** reduces or increases according to the mouse wheel/mouse movement. The center of the zooming area is the area, over which the mouse cursor was set.
Note. You can change zooming direction in Preferences -> 3D View.

To top

Pan

To pan an object in the **3D View**:

- Press the mouse wheel or **ALT** key + click on the 3D object.
- Hold and drag the 3D object to any direction. The object moves according to the mouse movement.

To top

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9.2.9. Front and Back Clipping Plane

Diving into / Clipping allows studying some details without being disturbed by front and back objects.

- Click button to turn *On* **Clip front Mode**. The north - south cursor type appears (see Figure 1).

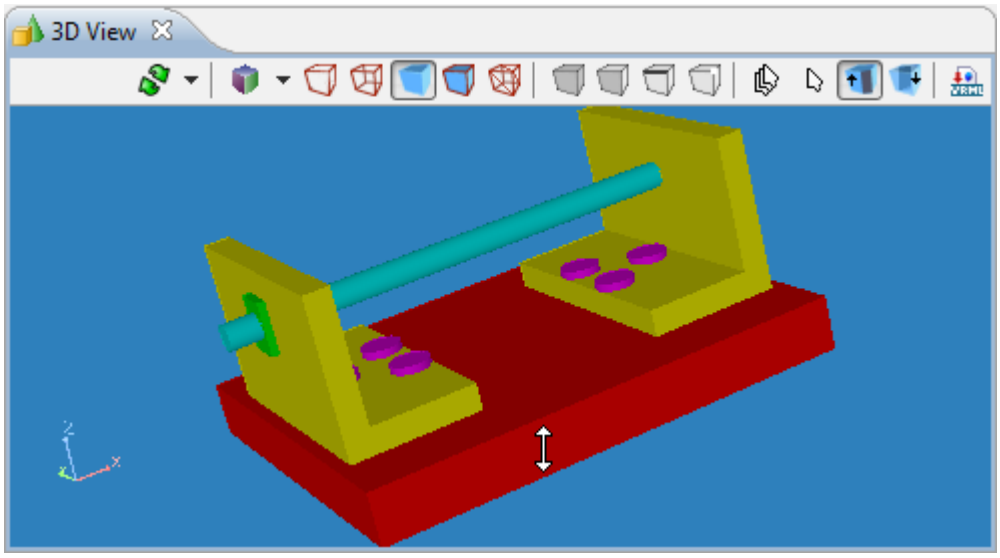

Figure 1. Clip front mode turned *On*, north-south cursor type

- Click and drag the mouse upwards to dive into a front clipping plane of the 3D object (see Figure 2).
- Click and drag the mouse downwards to restore an initial shape of a 3D object.

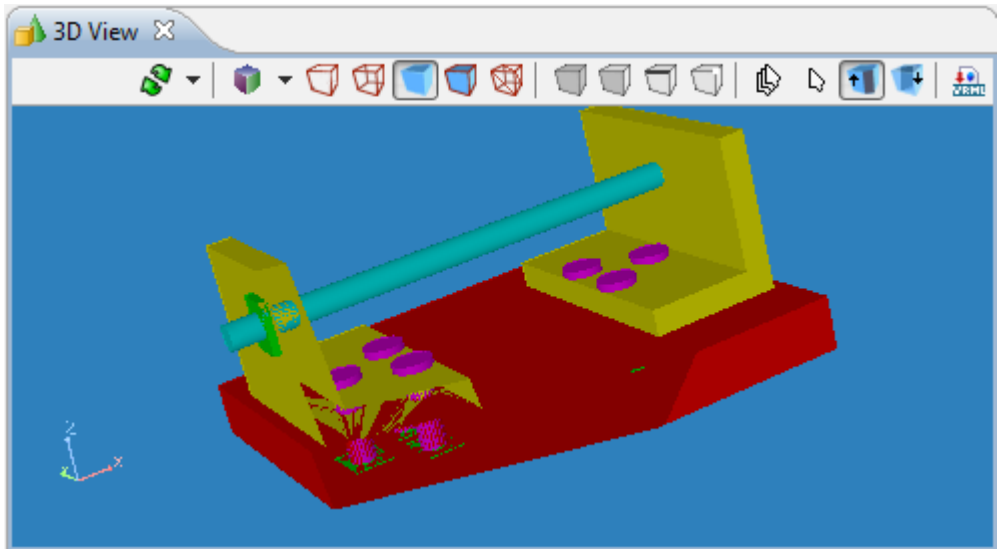

Figure 2. Clipped front plane

- Click button to turn *On* **Clip back** mode. he north - south cursor type appears.
- Click and drag a mouse downwards to dive into a back clipping plane of the 3D object (see Figure 3).
- Click and drag a mouse upwards to restore an initial shape of a 3D object.

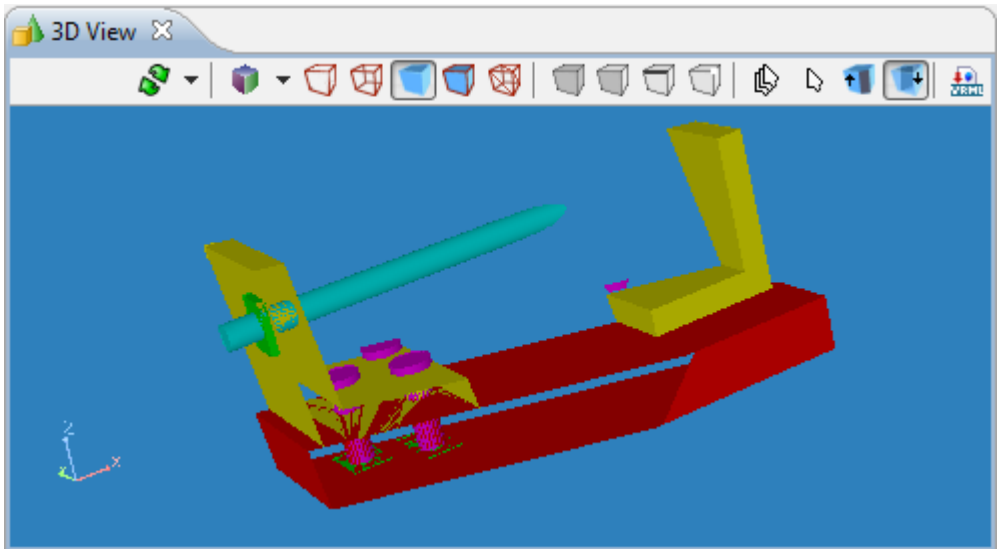

Figure 3. Clipped front and back planes

To top

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9.2.10. Inspecting Model

IDA-STEP presents a dynamic inspection of a model in the **3D View**. When a user is pointing with a mouse on a 3D object in the **3D View**, particular information is displayed in the IDA-STEP status bar. This information can be copied to the clipboard and then pasted to any text editor.
Note. **Dynamic selection** mode have to be switched *On*.

The following information can be displayed in the status bar:

- Component reference designator, when a particular component is selected (see Figure 1);

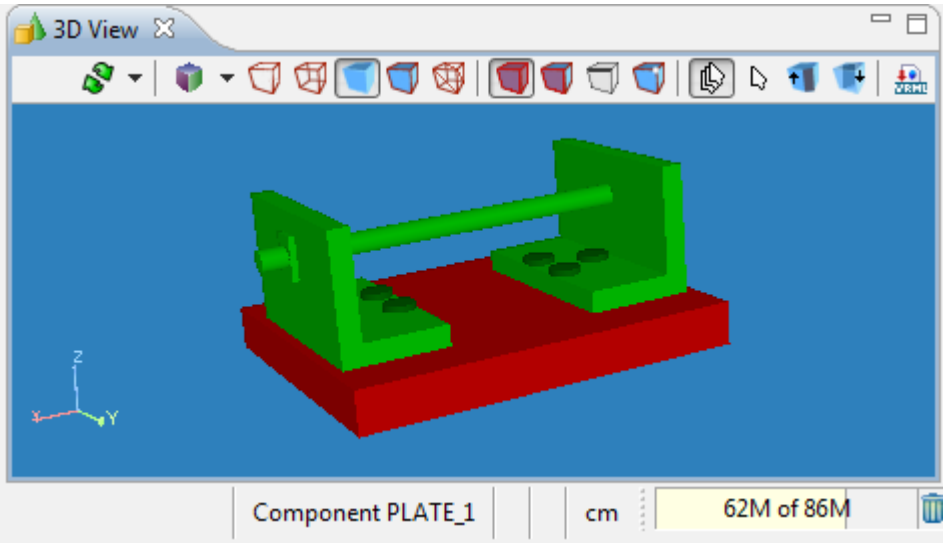

Figure 1. Status bar information, when component is dynamically selected (i.e. 'PLATE_1' - highlighted in red colour)

- Type of underlying surface (plane, cylinder with radius and etc.), an identifier of entity instance (#number), measurement units (see Figure 2);

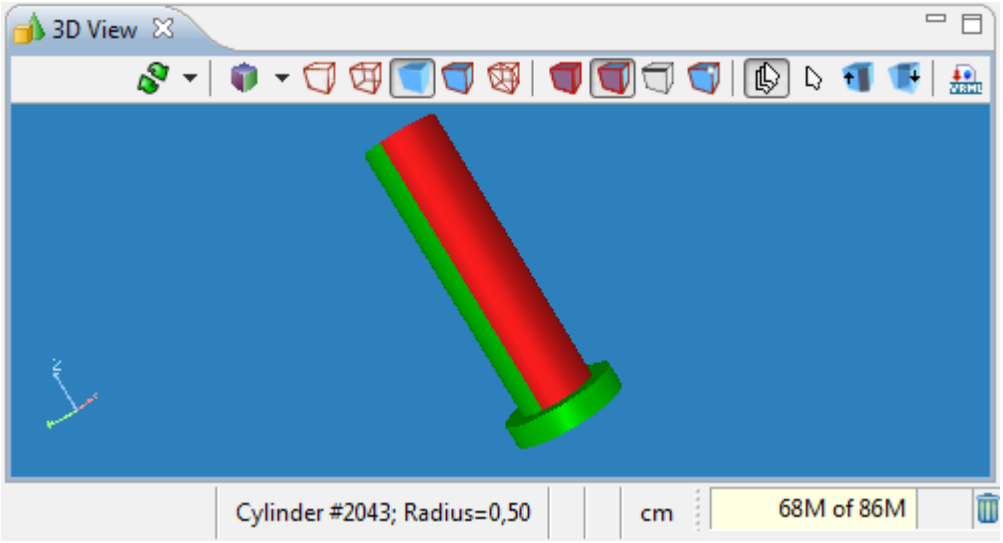

Figure 2. Status bar information: type of underlying surface ('Cylinder'), entity instance identifier ('#2043'), radius ('Radius=0,50') and measurement units (cm)

- Type of underlying curve with an absolute coordinates of the Start and End points, measurement units (see Figure 3);

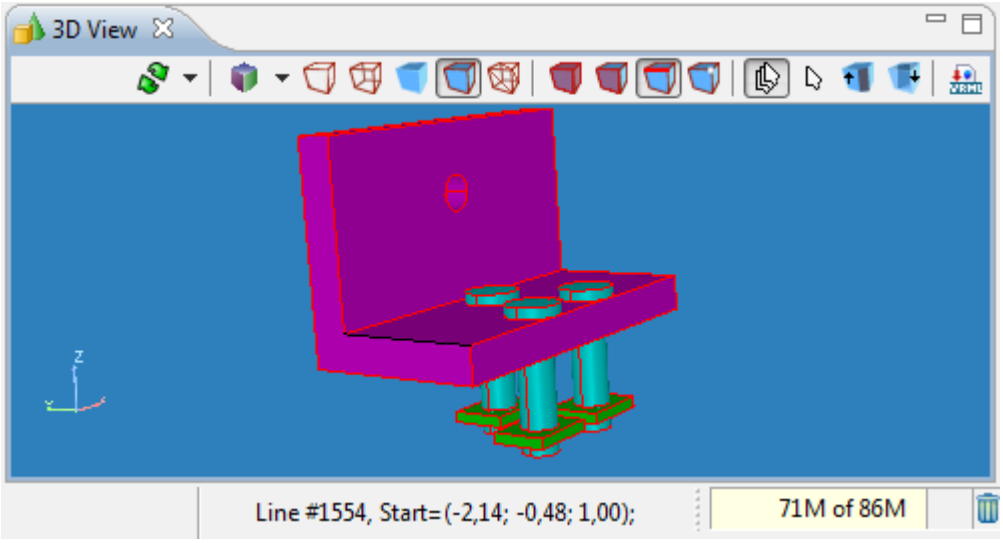

Figure 3. Status bar information: type of underlying curve ('Line'), entity instance identifier ('#1554'), Start point absolute coordinates ('Start=(-2,14; -0,48; 1,00)').

Note. The End point coordinates and measurement units are not visible in the Figure 3, because the window is minimized

- Point coordinates - absolute and relative, distance on the 3D object (see Figure 4). Refer to the Clause Measuring for more details.

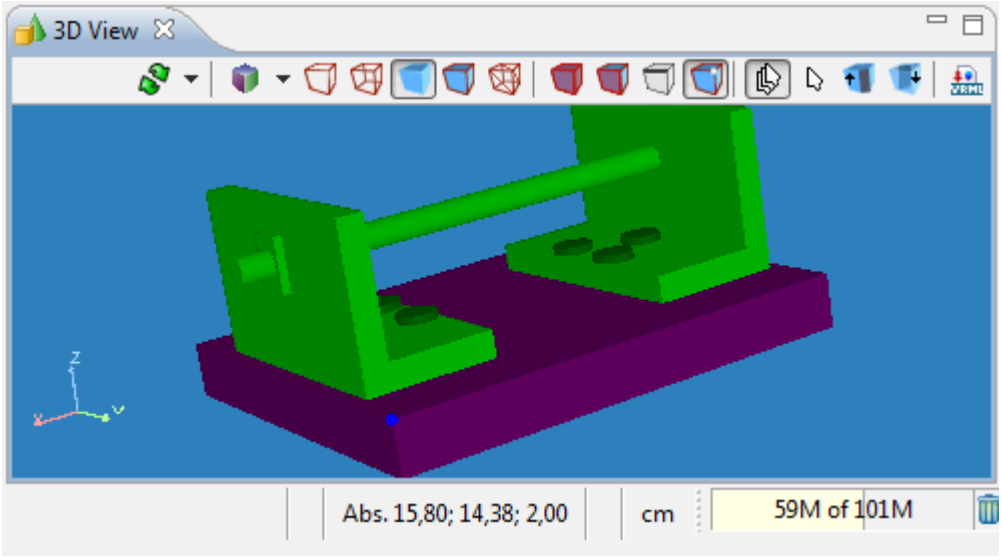

Figure 4. Status bar information: Absolute coordinates of blue point ('Abs. 15,80; 14,38; 2,00') and measurement units (cm)

Note. How to select particular object (component, surface, curve or point), please refer Clause Selection Modes.
The measurement units are displayed in the status bar according settings, selected in the **Preferences** page **Quantities and Units** panel (it can be accessed by selecting **Window -> Preferences -> Numbers -> Quantities and Units**).

Information in the status bar may be copied (CTRL + C) and pasted in a preferable editor.
Follow the routine to copy a dynamic information in the **3D View**:

- Press **Dynamic selection** button in the **3D View** tool bar to switch the dynamic selection mode *On*.
- Press **Face/Surface Selection Mode** button to set the surface selection mode *On* (another selection mode can be selected).
- Fly with a mouse pointer by the surface you want to inspect. The particular information is displayed in the status bar (see Figure 2).
- Press **CTRL + C** keys (Don't move the mouse!). The information, that is displayed in the status bar is copied into the clipboard.
- Now open preferable editor and paste the information from the clipboard.

To top

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9.2.11. Measuring

With a **Viewer Pro** component of IDA-STEP it is possible to measure distances between points, edges/curves and surfaces in the **3D View**. Measurement is done in a **Dynamic Selection Mode**. First a single object (point, edge/curve or surface) is selected. In the case of a point, it's absolute coordinates are displayed in the statusbar. Then with the dynamic selecting another object is highlighted by moving the mouse over the object and relative coordinates (Rel.) of this object are displayed in the statusbar. The distance between the selected object and the highlighted one under the mouse is visualized by a white air-line. The measured distance together with units is displayed in the statusbar (see Figure 1).

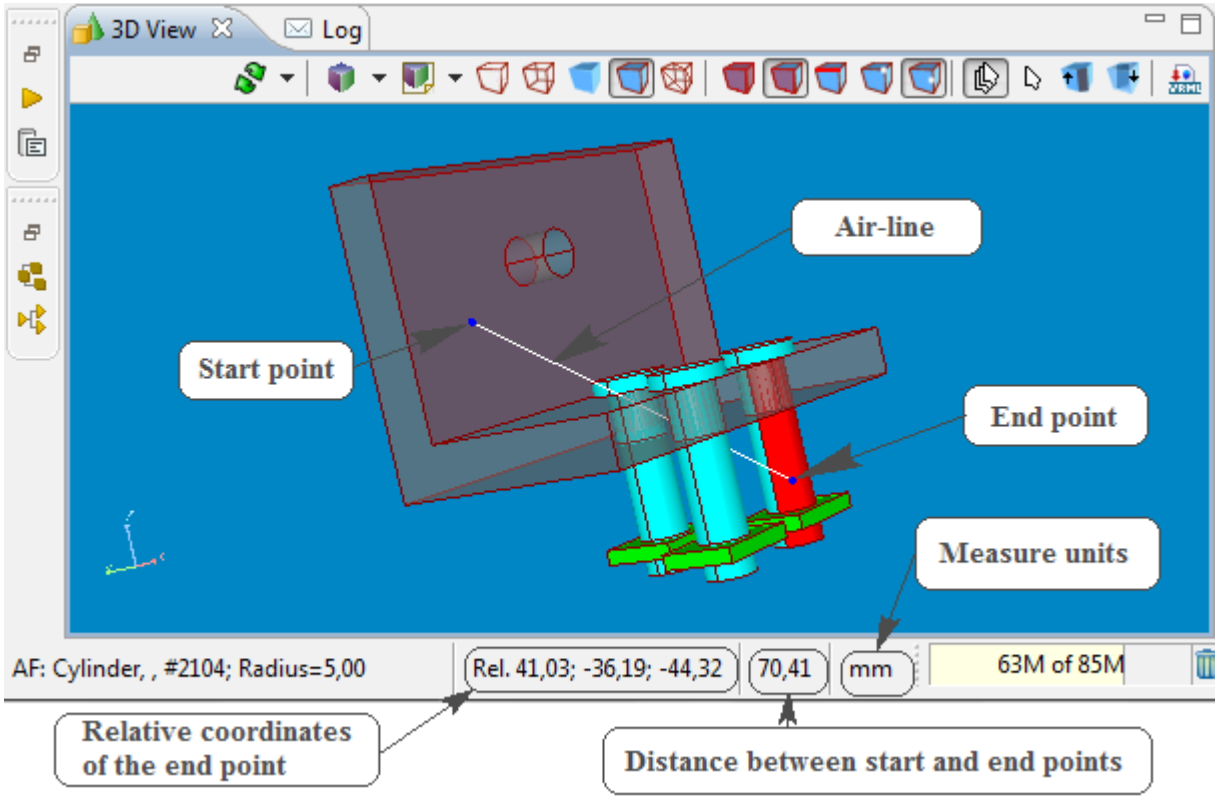

Figure 1. Measuring between two arbitrary points

There is a quite big number of possible combinations (i.e. 36) of what can be measured between. The object type should be selected by switching *On* a particular selection mode in the **3D View** tool bar. The components, which can be measured together with the selection mode combinations:

1. Face - **Face/Surface Selection Mode**,
2. Edge - **Edge/Curve Selection Mode**,
3. Point - **Vertex/Point** or **Point on Edge or Surface Selection Mode**,
4. Point on Component - **Component/Model** and **Point on Edge or Surface Selection Mode**,
5. Point on Face - **Face/Surface** and **Point on Edge or Surface Selection Mode**,
6. Point on Edge - **Edge/Curve** and **Point on Edge or Surface Selection Mode**.

Follow the routine to measure a distance between two points/edges/faces:

- Ensure the **Dynamic Selection Mode** is activated.
- First of all, you have to decide what object should be a *start* point and switch a particular selection mode *On* in the **3D View** tool bar. For example, the start point is a point on the face. So press on the **Face/Surface Selection Mode** and **Point on Edge or Surface** buttons in the tool bar.
- Now set a start point on the geometry object in the **3D View** by pointing with a mouse on an applicable face. In some cases (for **Point on Edge or Surface Selection Mode**) you can select a point by mouse right click and selecting **Select point** menu item in the appeared context sensitive menu. The absolute coordinates of this selected point are displayed in the status bar (string 'Abs' in front of the coordinates). Only when dynamically setting the start point a user will dynamically see absolute coordinates. Otherwise only relative coordinates are displayed in the status bar. To switch back to the dynamic first point selection, press ESC key.
  Note. When measuring distance between objects (faces, edges) the start point position is not set by the user, it is assumed to be the nearest point of the face or edge to the end object.
- Now it is a time to set an *end* point. If the end point or object is the same type as a start point, fly by with a mouse over the geometry object and the end point flashes up for all near points, which match the active selection mode. The shortest distance line between the start and end points is displayed and the length message appears in the status bar every time a new end point is highlighted. If the end point type differs from the start point or object, switch an applicable

selection mode *On* and then fly by with a mouse over the geometry object to measure a distance. For example the end point is an edge, so press on **Edge/Curve** (available for **Solid with Wireframe Style**) and fly by with a mouse over the geometry object. The point flashes up for all near points on the wire frames.
Note. Right after the mouse pointer is moved, the relative mode is switched *On*. That means the coordinates in the status bar are displayed with the string 'Rel.'. The coordinates are displayed as delta-x, delta-y, delta-z from the start point to the end point. To switch back to the dynamic start point selection, press ESC key.

Please find some examples below.

Measure distance between two Faces

Let's measure a distance between two faces (see Figure 2):

- Turn *On* the **Dynamic Selection Mode** button.
- Turn *On* the **Face/Surface Selection Mode** button.
- Select an applicable face on the geometry model in the **3D View**.
- Fly by with a mouse over an applicable face/surface. The shortest distance between highlighted faces is displayed in the statusbar.

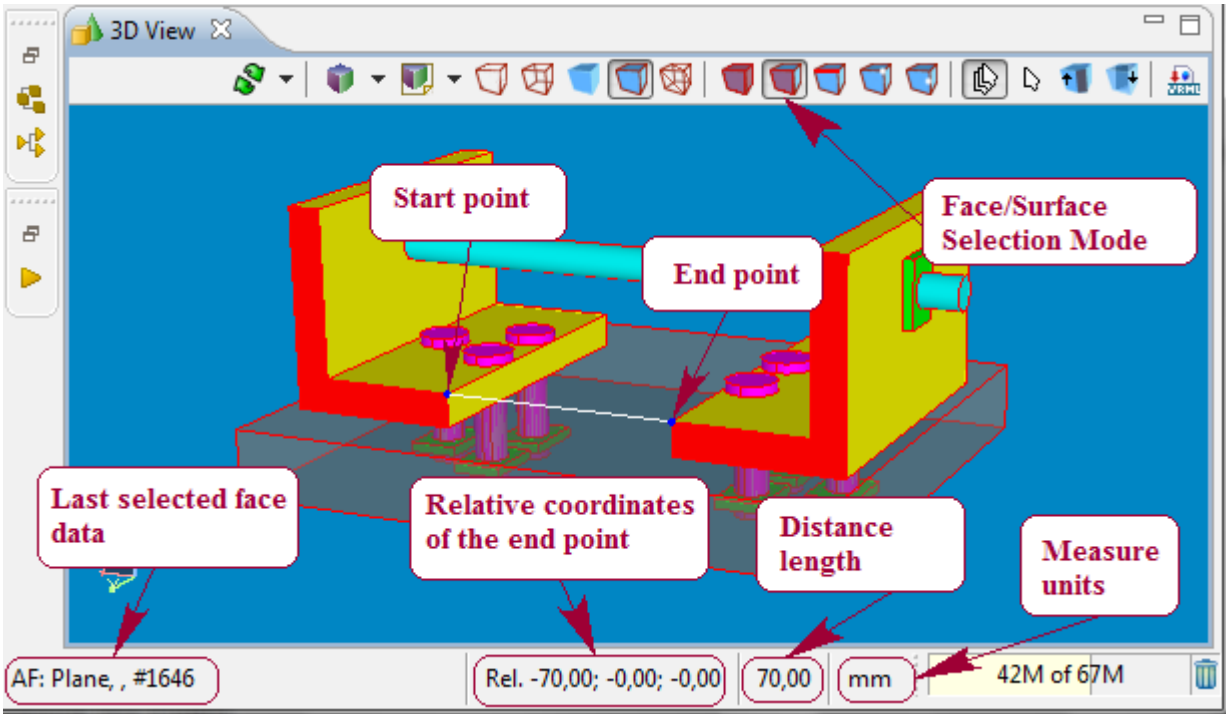

Figure 2. Measurement between two faces

Measure distance between Face to Edge

Let's measure the shortest distance between a face and an edge:

- Turn *On* the **Dynamic Selection Mode** button.
- Turn *On* the **Face/Surface Selection Mode** button.
- Select an applicable face on the geometry model in the **3D View**.
- Turn *On* the **Edge/Curve Selection Mode** button.
- Fly by with a mouse over an applicable edge/curve. The shortest distance is displayed in the statusbar.

Measure distance between two Points

Let's examine, how to measure distances between two vertex points on components on objects within the **3D View**:

- Turn *On* **Dynamic Selection Mode**.
- Press **Vertex/Point Selection Mode** button - to select a vertex.
- Fly by with a mouse pointer over the 3D object. The point flashes up for all near points. The absolute coordinates ('Abs.') of these points are displayed in the status bar (see Figure 3).
- Click on an object in the **3D View** to set an anchor point - the start point of the measuring. **Note.** Only when dynamically setting the first point a user will dynamically see absolute coordinates. Otherwise only relative coordinates are displayed in the status bar. To switch back to the dynamic start point selection, press ESC key.

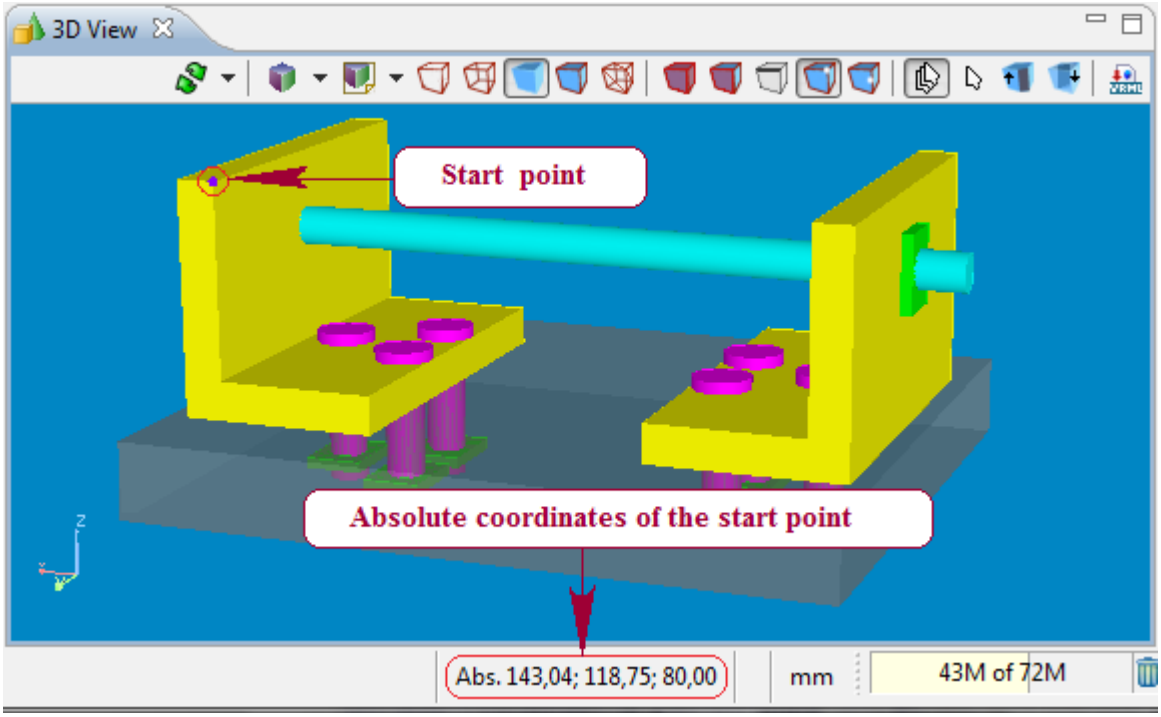

Figure 3. Start point on geometry model, absolute coordinates of the start point in status bar

- Select an *end* point of the measuring line by flying by with a mouse on a 3D object. The point flashes up for all near vertex points. Right after the mouse pointer is moved, the relative mode is switched *On* and relative coordinates are displayed in the statusbar. The "air-length" between the start and end points is displayed on the right of the coordinates (see Figure 4).

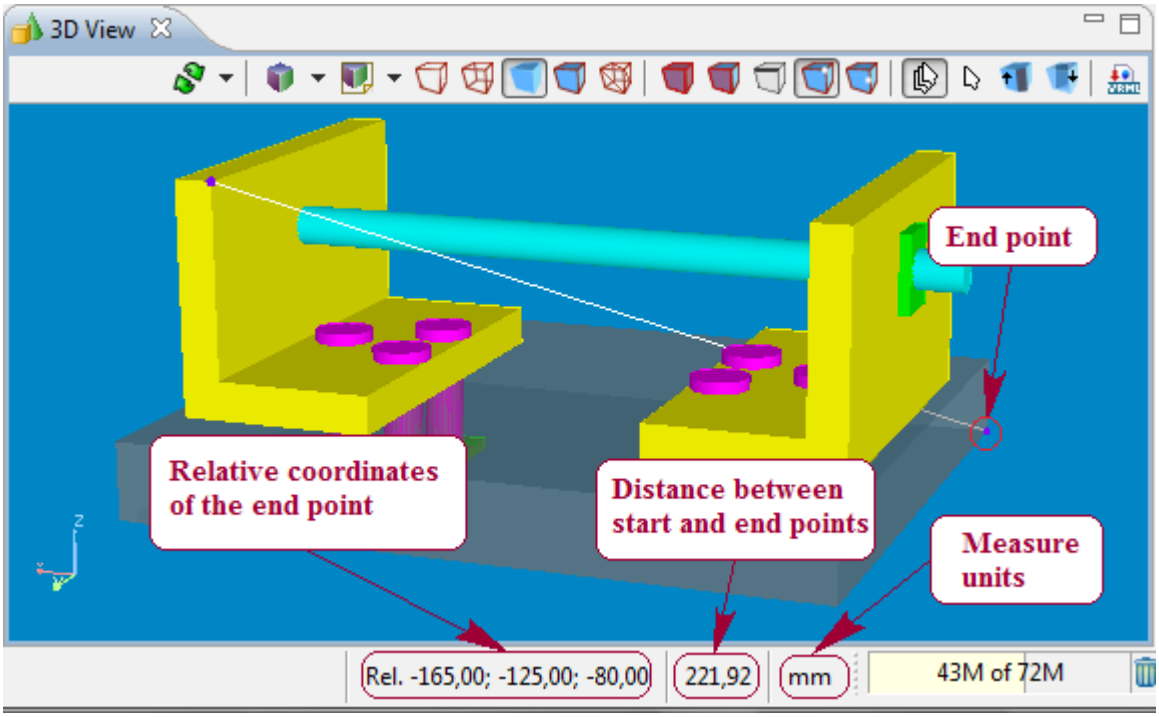

Figure 4. End point and measuring data in statusbar

The measuring results are presented in the statusbar (see Figure 4):

1. Point coordinates (x, y, z):
    1. Abs. - absolute anchor point coordinates;
    2. Rel. - relative end point coordinates: delta-x, delta-y, delta-z from the anchor point (white point) to the end point (blue point);
2. Distance between the start and end points;
3. Measure units.

Note. The above measurement numbers and units are displayed according settings, selected in the **Preferences** page **Quantities and Units** panel (it can be accessed by selecting **Window -> Preferences -> Numbers -> Quantities and Units**). The colours of points (start/end point) can be changed in **Window -> Preferences -> 3D View -> Display Styles -> Element colors** (Highlighted color - for Vertex/Point).

Changing Measure Units

Let's examine, how to switch to another measure units, e.g. "inch" instead of "cm" (you can find measure units in the status bar of IDA-STEP). First of all open preferences page by selecting in the main menu bar **Window -> Preferences -> Numbers -> Quantities and Units**.

1. Select the *length* quantity in the table of the **Quantities and Units** page.
2. Select *inch* value from the list of available units for the selected quantity, which is located on the right side of the **length settings** layout.
3. Press **Set As Default** button to set the selected unit as default unit.
4. Press **OK** button to confirm changes.

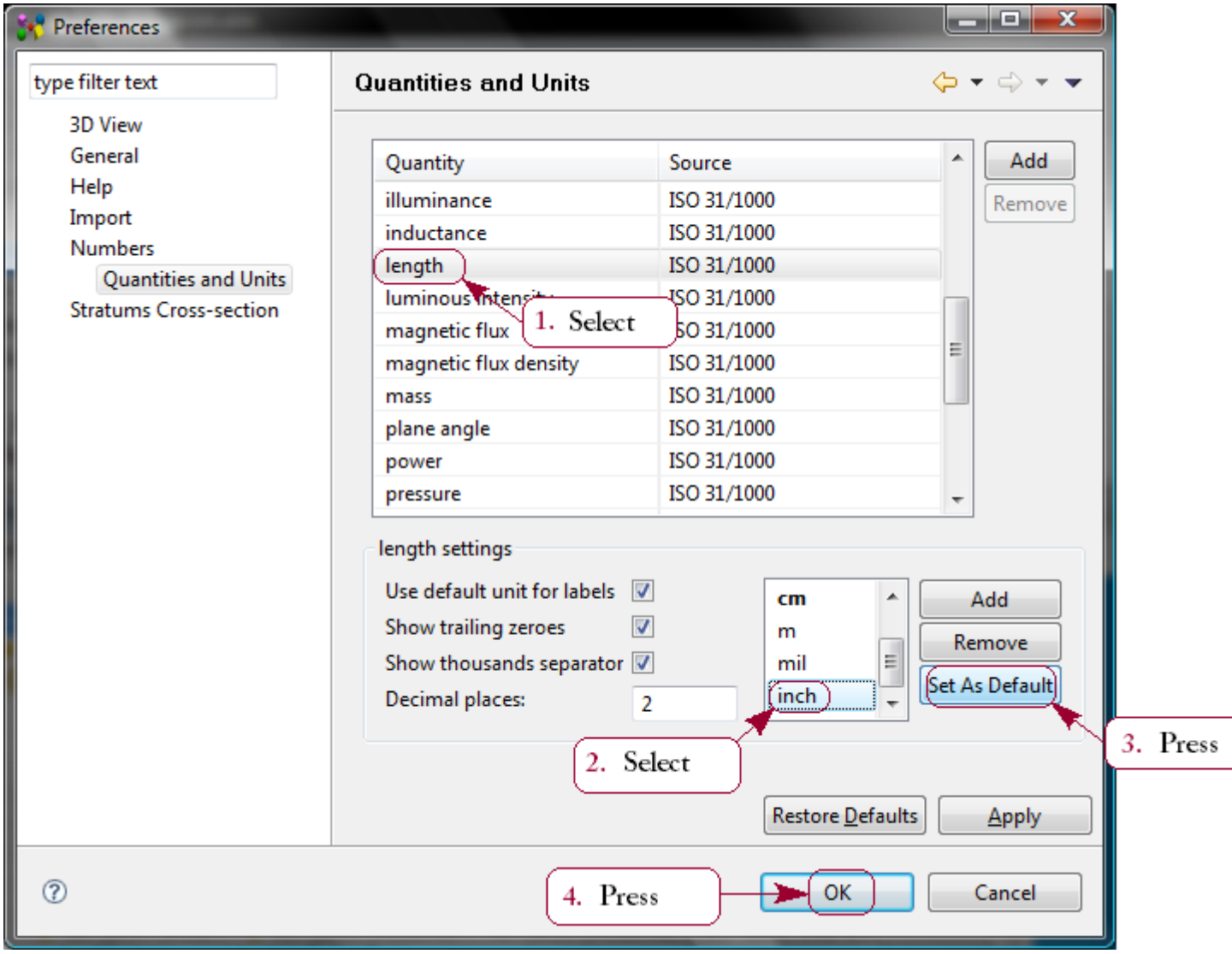

Figure 5. Quantities and Units page with steps to change measure units form current units (cm) to "inch"

The currently set units are displayed in the status bar of IDA-STEP. All measurements are recalculated (see Figure 6).

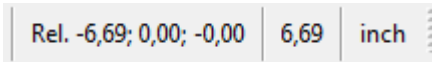

Figure 6. Status bar with changed units and recalculated coordinates

For more details see Preferences -> Numbers -> Quantities and Units.

To top

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9.2.12. Optimize Geometry

Optimize geometry is a tool to combine neighbouring faces (cylinders, cones, spheres) when their underlying surface is the same or equivalent. The faces (i.e. cylinders, cones or spheres) must have the same radius (maybe with some tolerance) and must have the same orientation in space.
Optimizing algorithm also converts selected bspline surface to elementary surface (plane, cylinder, etc.) and converts bspline curves to elementary curves (lines, circles, etc.) if possible.
The *Optimize geometry* command works either on the whole 3D model or only on selected faces, curves (multiple selection).

Optimize geometry of a whole 3D model

The **Normal** + **Component/Model Selection Mode** should be switched on in the **3D View**. Follow the routine to combine all faces that belong together and convert bsplane faces/curves into elementary surfaces/curves:

- Select (highlight) a component in the **3D View**.
- Right click and select **Optimize geometry** menu item in the context sensitive menu (see Figure 1).
- The **Optimize geometry** message window appears on the screen, containing information about number of modified faces and curves (see Figure 2). Click **OK** button to close the message window.

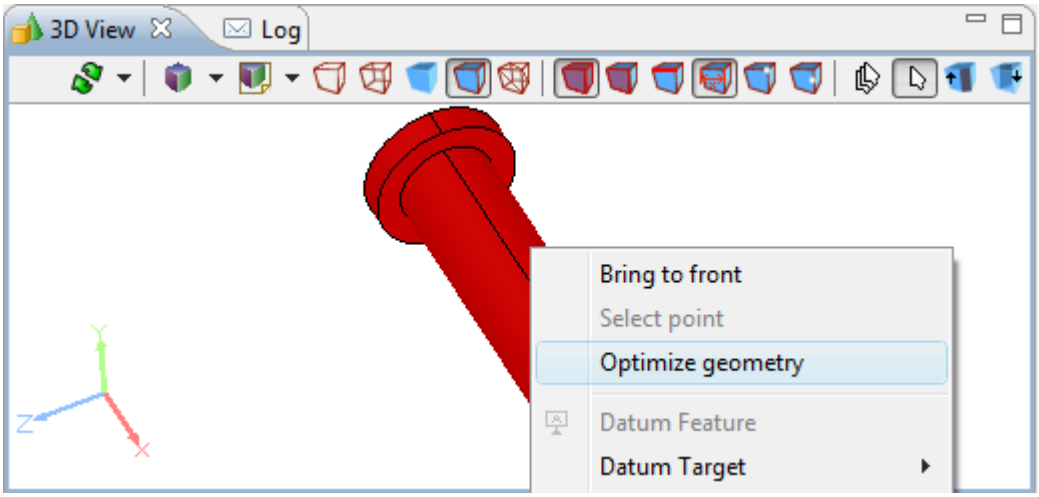

Figure 1. Whole component highlighted in 3 D View and Optimize geometry menu item selected

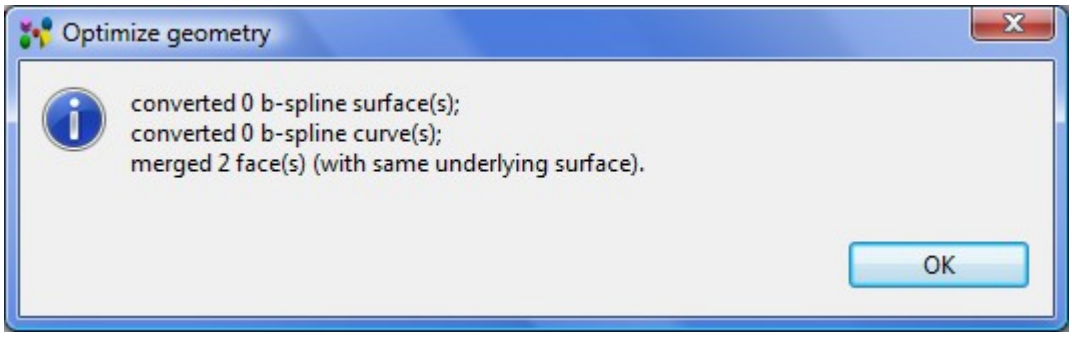

Figure 2. Optimize geometry message window with list of modified elements

In case the geometry was previously optimized or there are no faces/curves to optimize, the particular message 'Nothing is modified from the list of curves, faces and surfaces' appears in the **Optimize geometry** message window (see Figure 3).

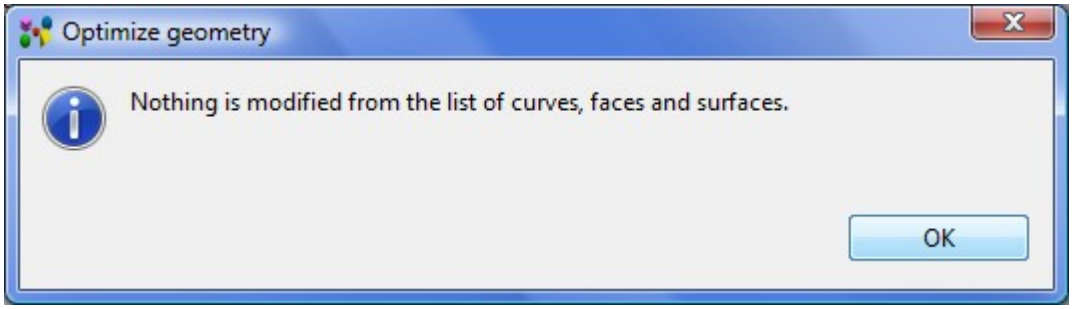

Figure 3. Optimize geometry message window informing there were nothing modified from the list of curves, faces and surfaces

Merge selected faces

First of all the **Normal** + **Face/Surface Selection Mode** should be switched on in the **3D View**. Follow the routine to merge two selected faces:

- Select the first face to be merged on the 3D geometry object in the **3D View** (see Figure 4).

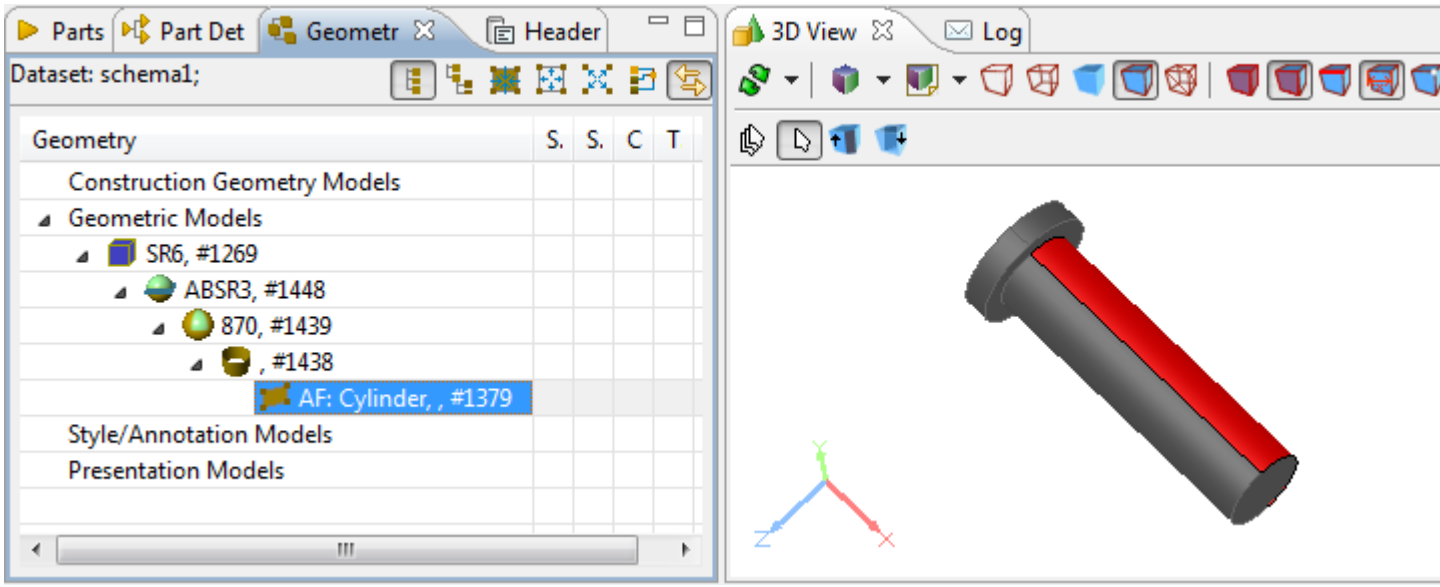

Figure 4. Selected surface in 3D view is highlighted in Geometry Details view tree (Cylinder, #1379)

- Select the second face on the 3D geometry object in the **3D View** by holding **Ctrl** key and clicking on the surface (see Figure 5).

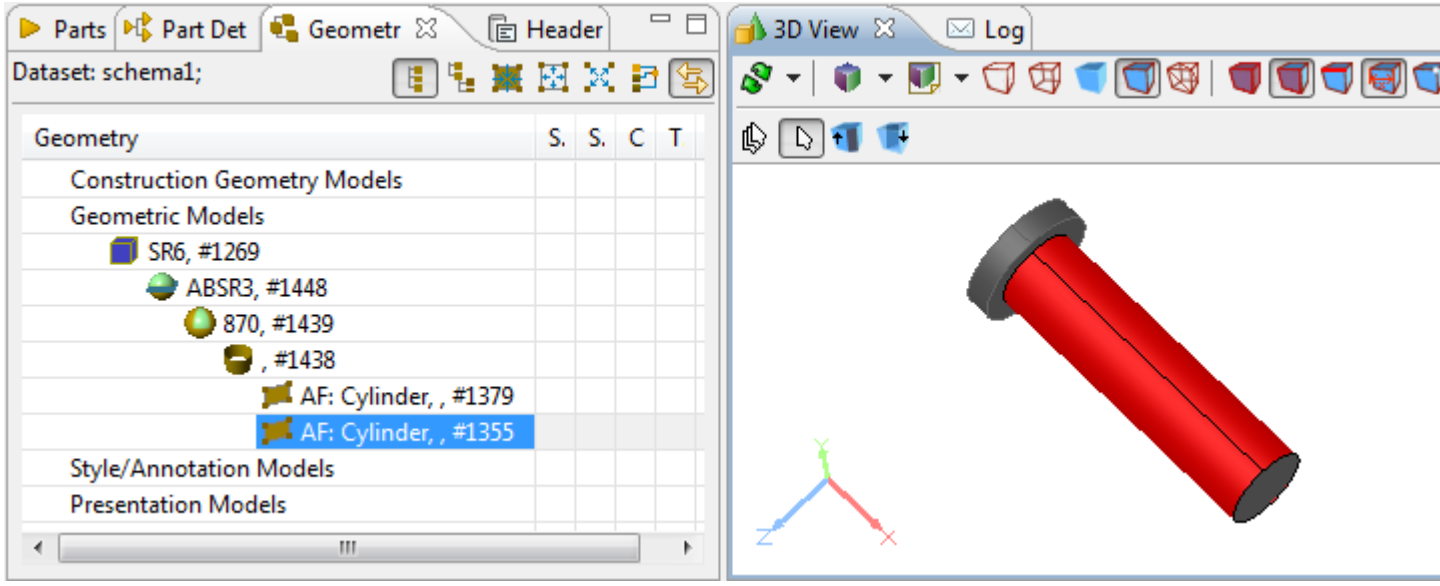

Figure 5. Two surfaces are selected in 3D View, the last selected surface is highlighted in Geometry details view tree

- Right click and select **Optimize geometry** menu item in the appeared context sensitive menu.

The message window **Optimize geometry** appears on the screen, informing, what number of faces were modified by merging (see Figure 6). Click **OK** button to close the window.

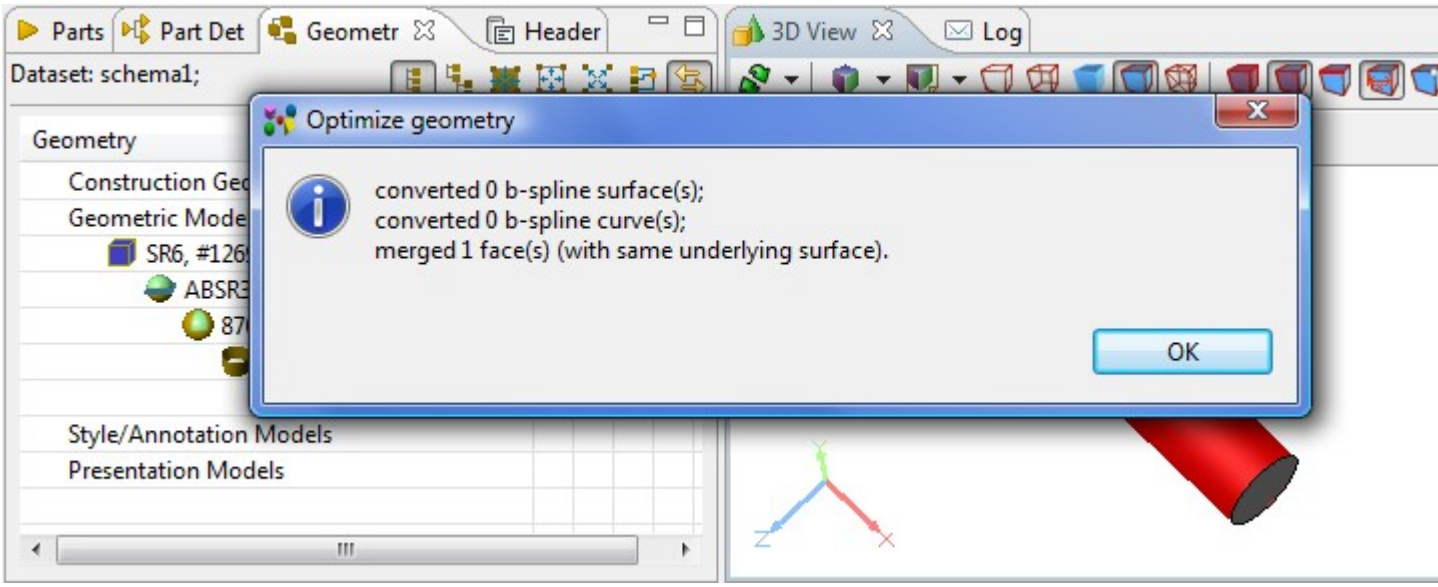

Figure 6. Optimize geometry message window with information about modified surfaces

In case selected faces are different and could not be merged (see Figure 7), the **Optimize geometry** message box appears with the message, that particular faces are not the same or equivalent (see Figure 8).

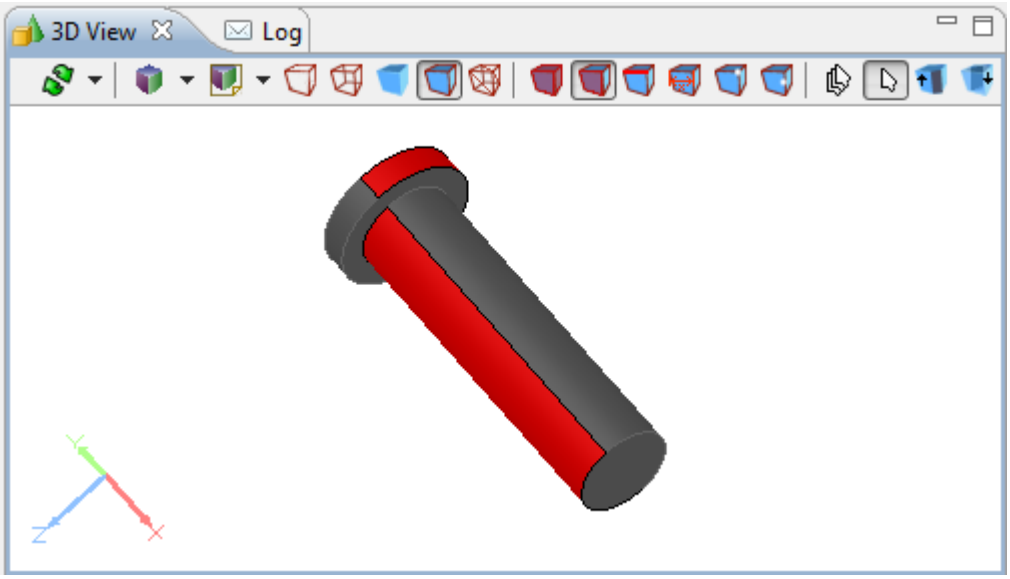

Figure 7. Selected different surfaces in 3D View

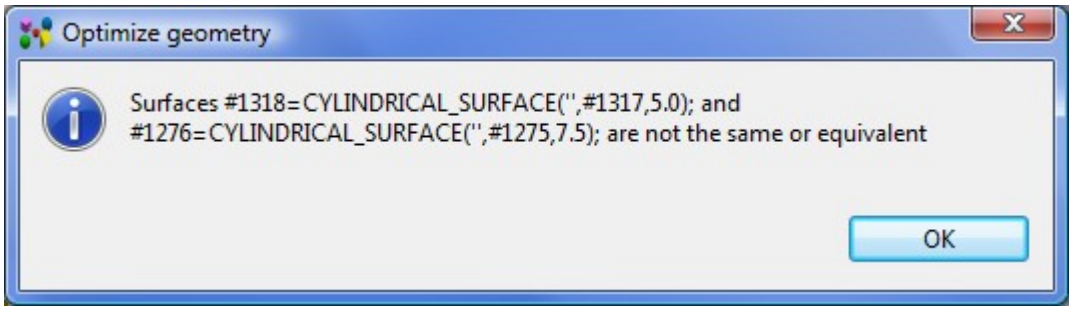

Figure 8. Optimize geometry window informing, that faces are not the same or equivalent

There can be selected more than two faces in the 3D model. The **Optimize geometry** command mergers all faces that belong together. After the merging is finished, the **Optimize geometry** window appears on the screen with the list of merged faces (see Figure 2).

Convert selected bspline surfaces/curves to elementary surfaces/curves

Follow the routine to convert selected bspline surface to elementary surface (i.e. *cylinder*):

- Ensure the **Normal** + **Face/Surface Selection Mode** is turned *On*. **Note.** In case the bspline curve, turn *On* the **Edge/Curve Selection Mode**.
- Select a bspline surface in the **3D View**. The type of selected geometry is visible in the left bottom corner of the page (see Figure 9).

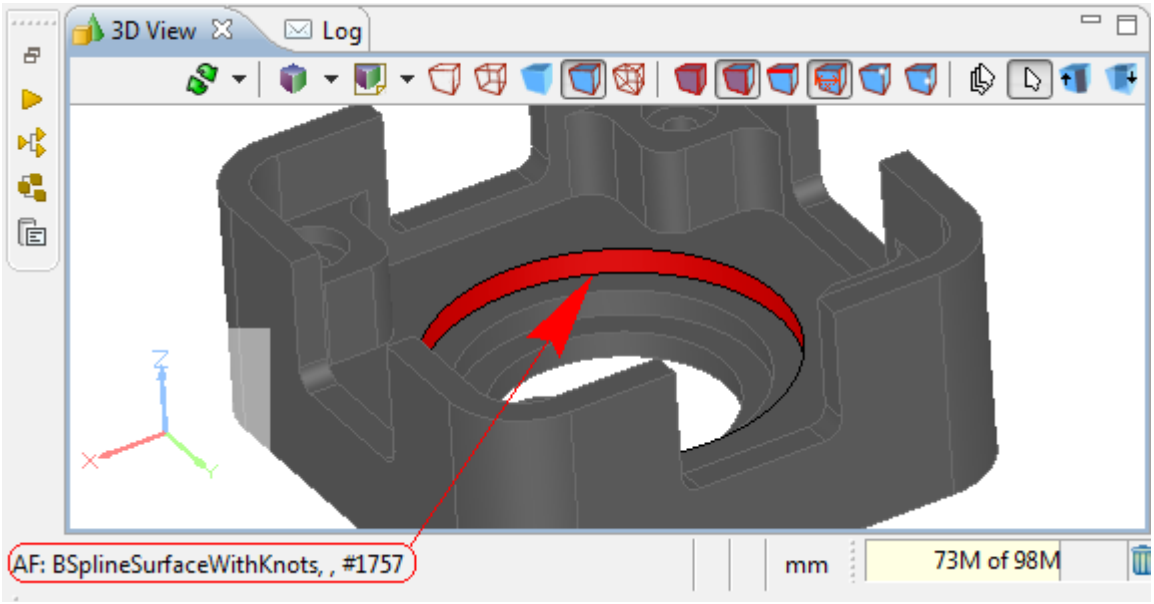

Figure 9. Selected Bspline surface

- Right click and select **Optimize geometry** menu item in the appeared context sensitive menu.
- The message window **Optimize geometry** appears on the screen, informing, what number of faces were modified by merging (see Figure 10). Click **OK** button to close the window.

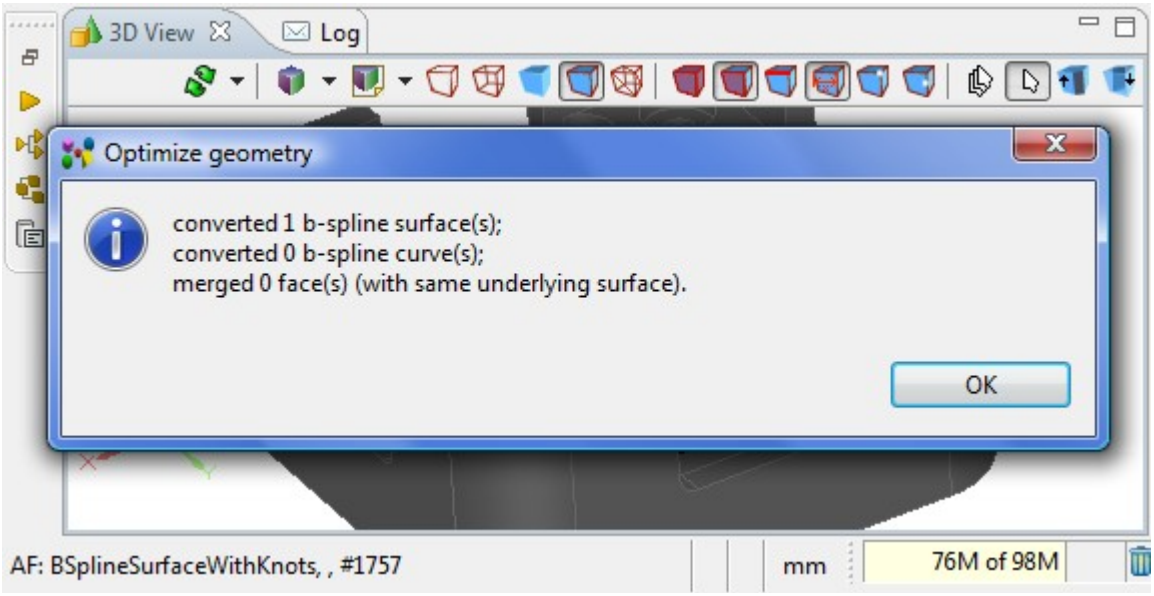

Figure 10. Optimize geometry message window with information about modified surfaces

- Select the same surface in the **3D View**. The current surface type with available data (i.e hash number and radius) is displayed in the left bottom corner of the page (see Figure 11).

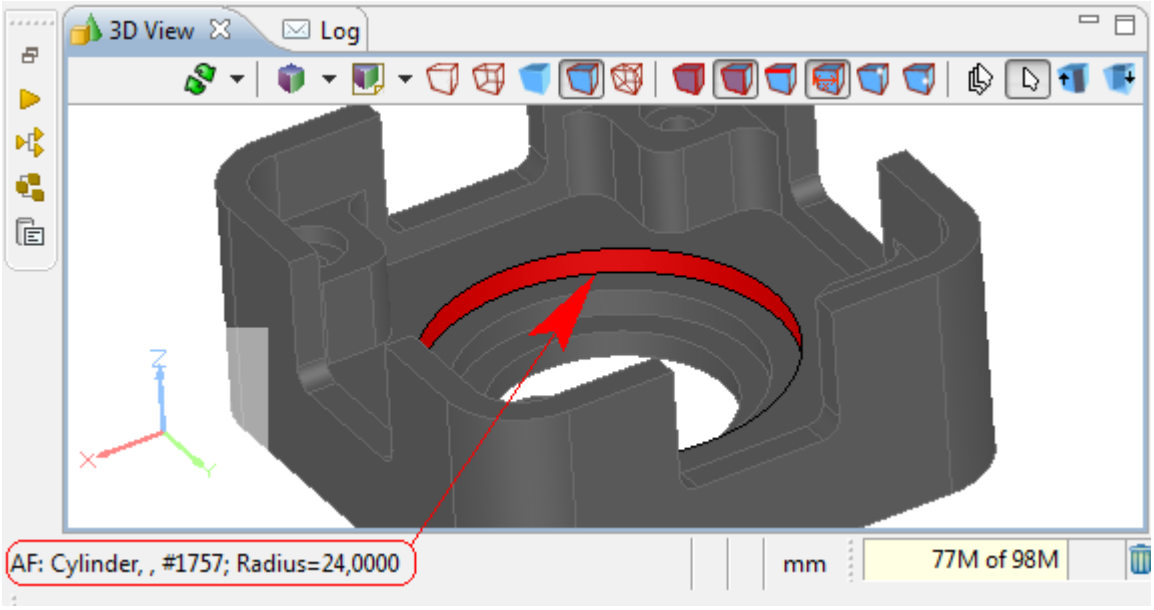

Figure 11. Converted surface

To top

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9.2.13. Re-triangulate

On the initial opening of a 3D model, the optimized triangulation is applied. It takes less time and is sufficient for reviewing of the hole model in the overview mode. When the user zooms in (moves the viewpoint closer to the model) it is apparent that the features are faceted (see Figure 1 on the left). To get a smooth 3D model view for the current viewpoint, IDA-STEP introduces the *Retriangulate* command, which can be executed by pressing the **Retraingulate** button in the **3D View** tool bar.
Note. If the user zooms in more and the view looks faceted again, the user can repeat the re-triangulation command.

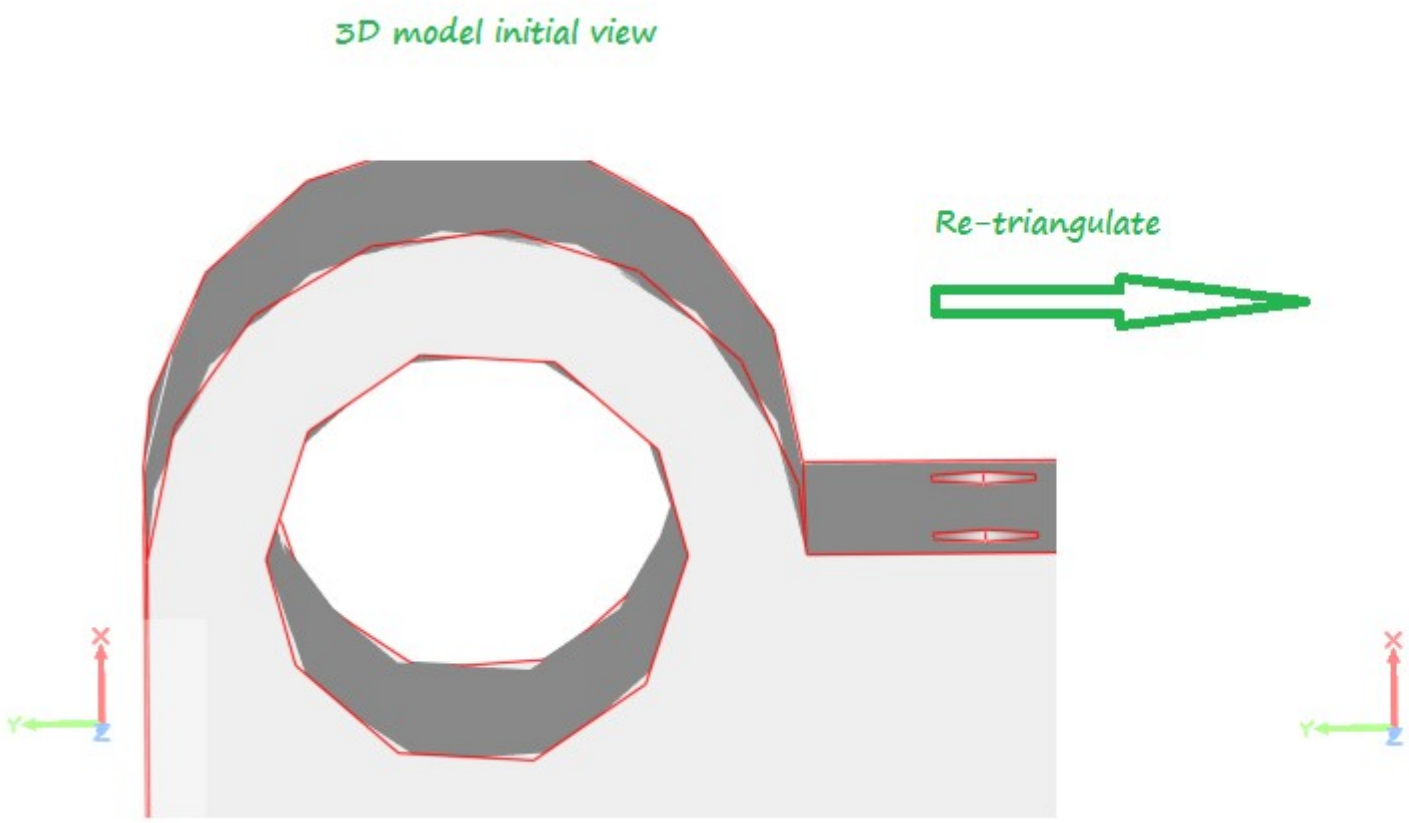

Figure 1. 3D model view before re-triangulation and after

In case of a complex assembly, the triangulation may take more time. During the triangulation process the **Progress Information** window with a message '*Retriangulate current shape*' appears on the screen (see Figure 2).

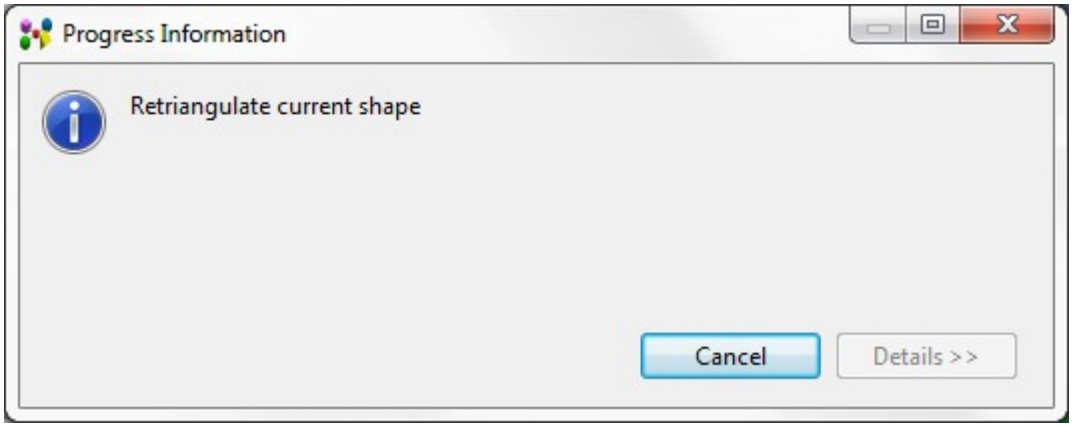

Figure 2. Progress Information window

To top

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9.2.14.1. Datum Feature

A *datum feature* in IDA-STEP is the actual surface component to establish a datum (points, lines, surfaces, planes, cylinders (see Figure 1) and etc., from which a location or geometric relationships of other features may be established or related).

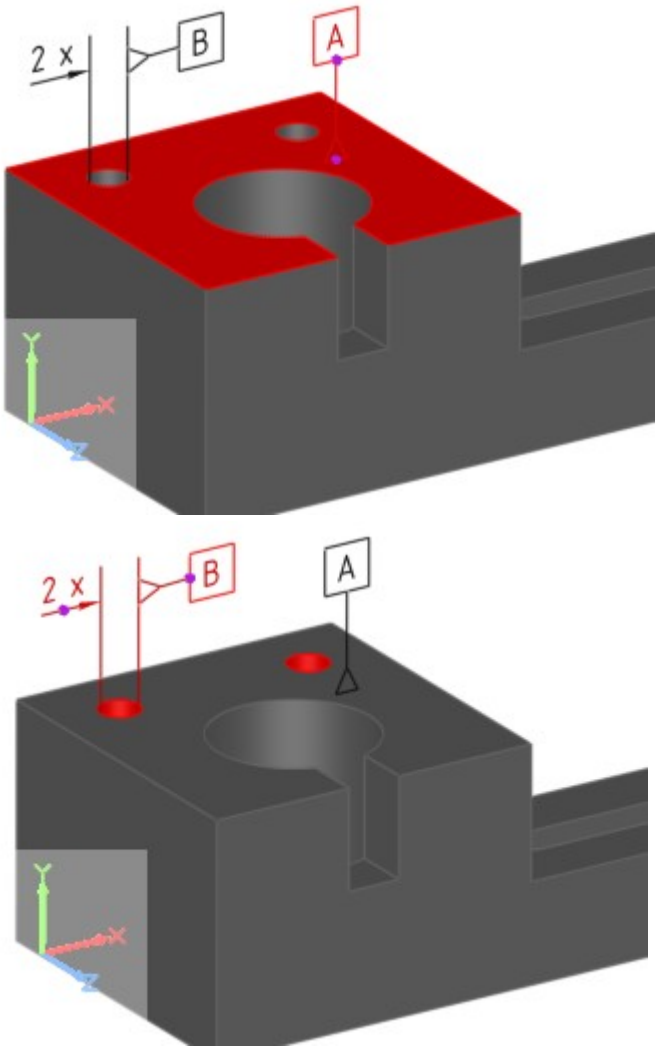

Figure 1. Datum feature of surface ('A') and feature of size of two cylinders 2 x ('B')

Create datum feature

Follow the routine to create a datum feature:

- Point with a mouse on a surface component in the **3D View**. **Note.** The selected point (even if no point selection mode is switched on) will be assumed as a start point of the leader line of the datum feature to be created.
- Right click and select **Datum Feature** item in the appeared context sensitive menu.
- The **Create Datum Feature** wizard page appears in the **GD&T** view area. Enter needed data in the wizard fields and click **Create** button. For more information see Create Datum Feature Wizard.

The datum feature is created and the datum feature symbol appears in the **3D View** (see Figure 1). The leader line of the created datum feature starts at the point, the user clicked while selecting a surface component.

Move leader line

There is a possibility to move a datum feature leader line with symbol on the selected datum feature surface and modify leader line:

- Click with a mouse on the datum feature symbol. Two points on the leader lines are highlighted.
- Click with a mouse on the point:

    - which is in contact with the datum feature object (i.e. plane) - to change the datum feature symbol position on the datum feature.
    - which is in contact with the block - to modify leader line.
- Drag with a mouse to the desired direction. Release the mouse button to set the position.

To top

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9.2.14.2. Datum Target

A *Datum Target* in IDA-STEP is a specified point, line or area on a part that is used in establishing a datum.

Create datum target

Follow the routine to create a datum target in the **3D View**:

- Ensure the **Vertex/point** or **Point on Edge or Surface Selection Mode** is turned *On* in the **3D View** tool bar.
- Point with a mouse on a geometry model on the area the datum target to be created.
- Right click and select **Datum Target** item in the appeared context sensitive menu.
- Select an item in the appeared of sub menu to open a particular datum target creation wizard in the **GD&T** view details area:
    - **Point Datum Target** - to open the Create Target Point Wizard. The Figure 1 displays, how the created datum target (point) is presented on the geometry model in the **3D View**.

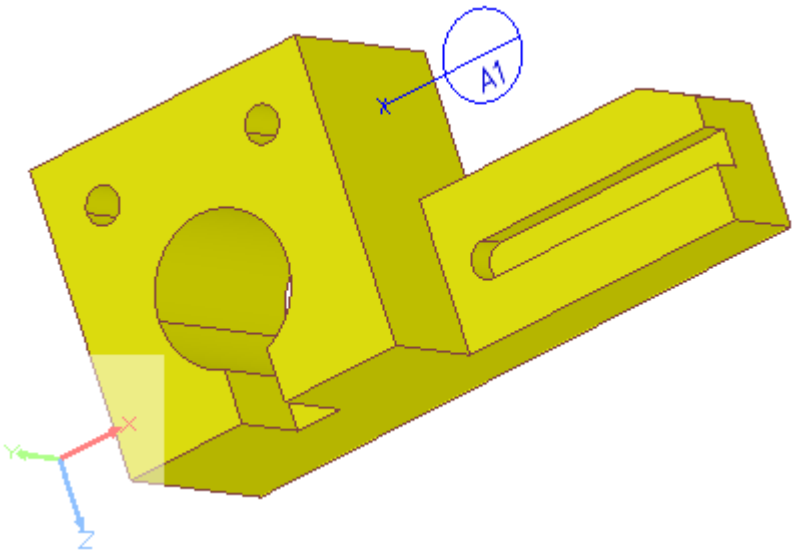

Figure 1. Datum target point with leader line and symbol

    - **Line Datum Target** - to open the Create Target Straight Line Wizard. The Figure 2 displays, how the created datum target (line) is presented on the geometry model in the **3D View**.

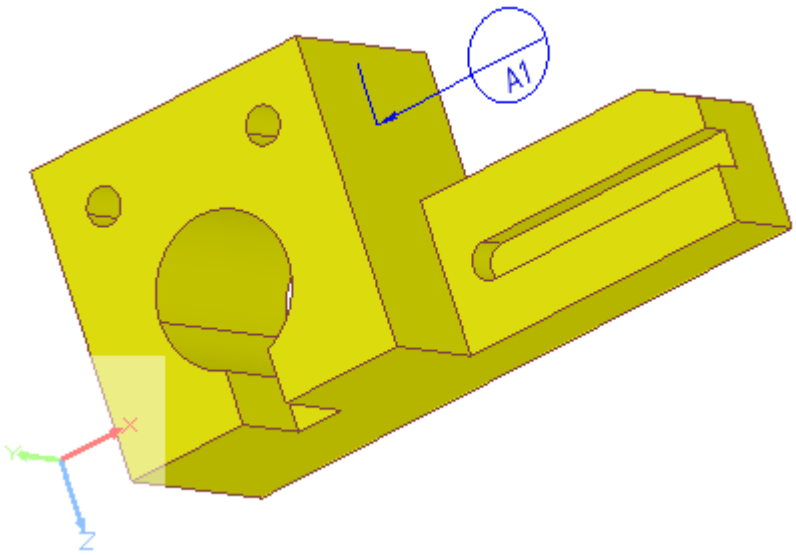

Figure 2. Datum target straight line with leader line and symbol

- **Circle Datum Target** - to open the Create Target Circle Wizard). The Figure 3 displays, how the created datum target (circle) is presented on the geometry model in the **3D View**.

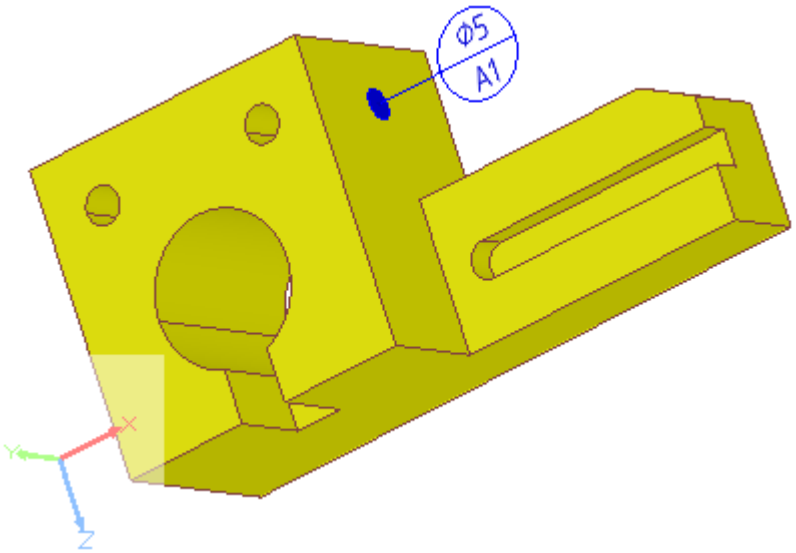

Figure 3. Datum target circle with leader line and symbol

- **Circular Curve Datum Target** - to open the Create Target Circular Curve Wizard. The Figure 4 displays, how the created datum target (circular curve) is presented on the geometry model in the **3D View**.

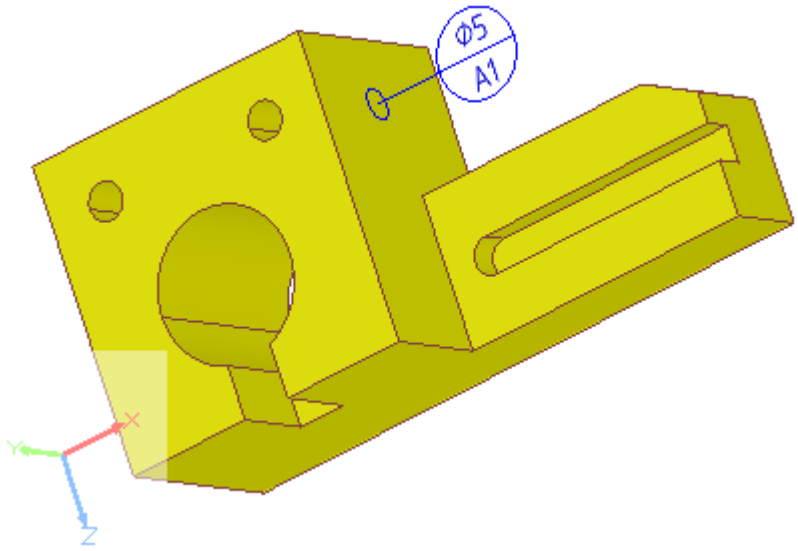

Figure 4. Datum target circular curve with leader line and symbol

- **Rectangle Datum Target** - to open the Create Target Rectangle Wizard. The Figure 5 displays, how the created datum target (rectangle) is presented on the geometry model in the **3D View**.

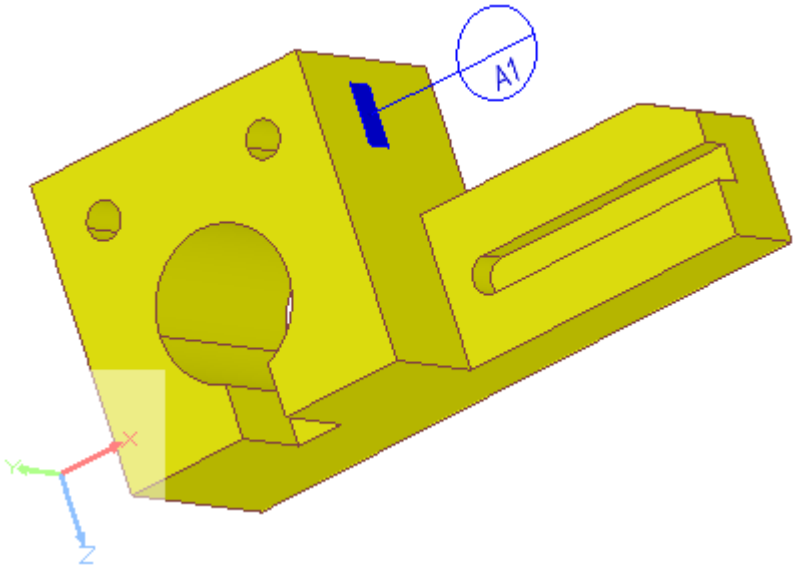

Figure 5. Datum target rectangle with leader line and symbol

Change location of datum target

To change location of a datum target start point (selected point/circle/line/rectangle on a surface, or surface component):

- Click with a mouse on the datum target symbol of an applicable datum target. Two points on the leader lines are highlighted (see Figure 6).
- Click with a mouse on the point, which is located on the datum target referenced surface or edge (the contact point or area).
- Drag with a mouse to the desired direction to change a location of the selected datum target. Release the mouse button to set the location (see Figure 6 b - the location of the datum target (rectangle) is changed).

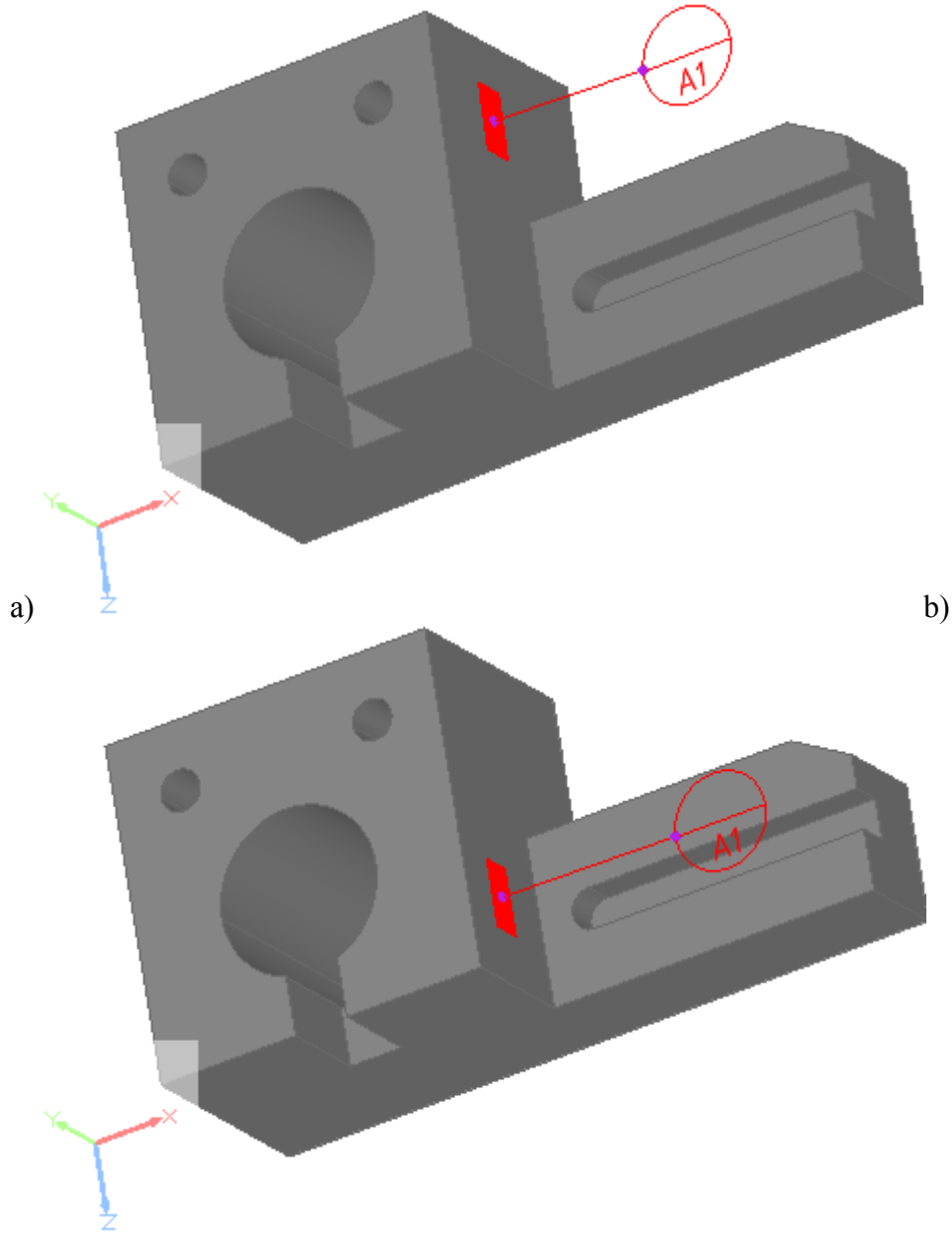

Figure 6. a) Selected datum target leader line and symbol with highlighted dragging points, b) changed location of datum target (rectangle)

Change datum target symbol position

To change position of a datum target symbol (end point of a leader line):

- Click with a mouse on the datum target symbol. Two points on the leader lines are highlighted (see Figure 6b).
- Click with a mouse on the point, which is in contact with the datum target symbol.
- Drag with a mouse to the desired direction. Release the mouse button to set the position (see Figure 7).

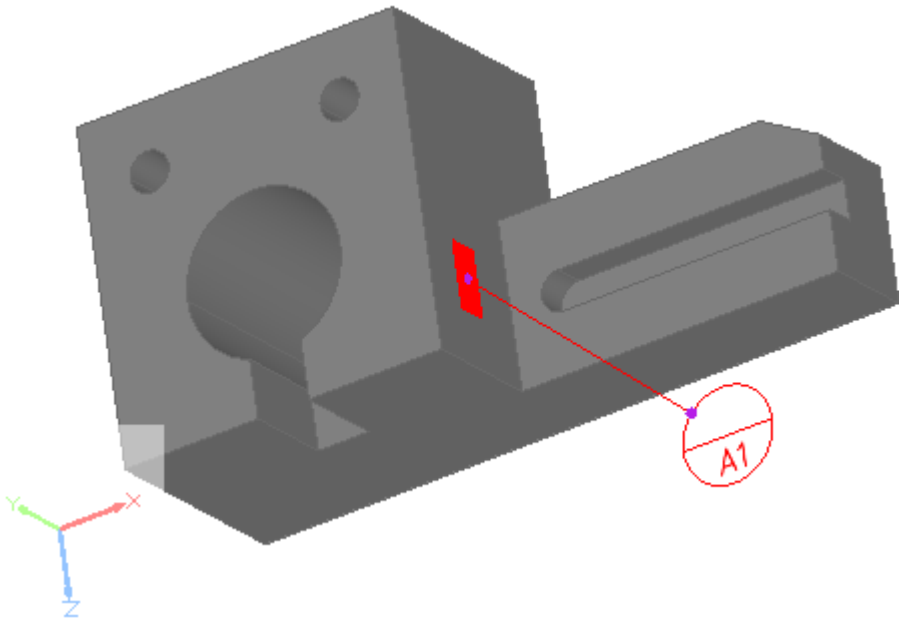

Figure 7. Datum target point with leader line and symbol, which position was changed

To top

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9.2.14.3. Datum System

A *Datum system* in IDA-STEP is a group of two or more separate datums used as a combined reference for a toleranced feature. **Note.** A datum system may be comprised of a single datum.

Create datum system

In order to create a datum system, at least one datum feature should exist (how to create datum feature, see Datum Feature -> Create datum feature).
Follow the routine to create a datum system in the **3D View**:

- Right click on the **3D View** area. The context sensitive menu appears.
- Select **Datum System** menu item. The **Create Datum System** wizard appears in the **GD&T** view details area. How to manipulate with the wizard, see Clause Create Datum System Wizard.
- After the creation, a new entry of the datum system appears in the **GD&T** view table. Press on the datum system entry to view presentation in the **3D View** (see Figure 1).

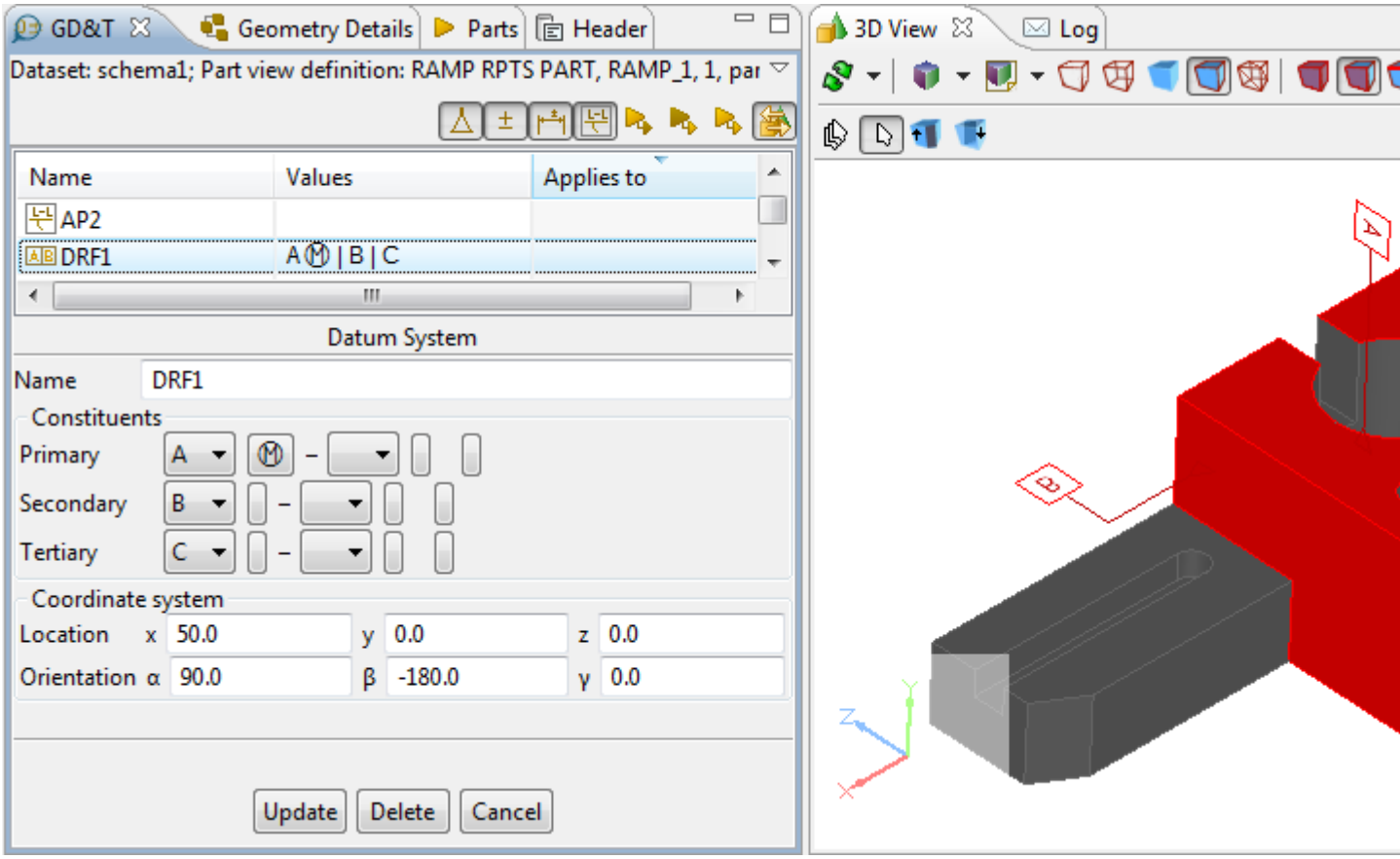

Figure 1. Selected datum system entry in GD&T view is highlighted in 3D view (all included datums are highlighted)

To top

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9.2.14.4. Geometric Tolerances

In the **3D View** geometric tolerance values are displayed in feature control frames, connected with the shape feature with a leader line (see Figure 1). In case the leader line is connected with a surface, the terminator symbol is a dot, in other cases (edge, vertex point), the terminator symbol is an arrow.

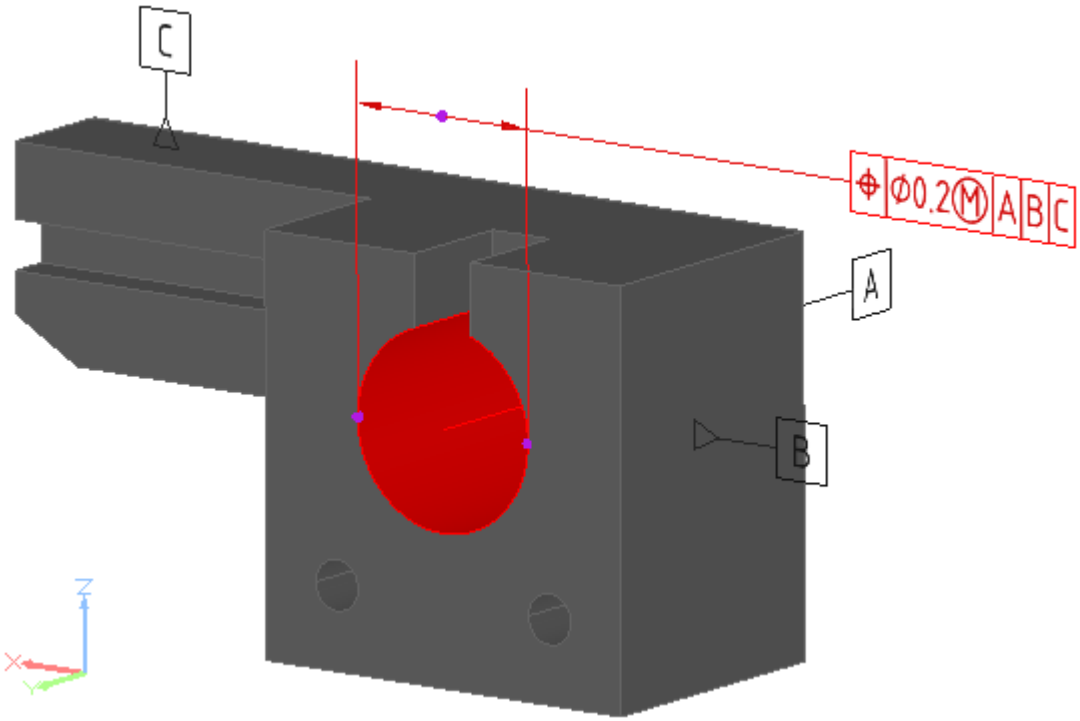

Figure 1. Single feature control frame of geometric tolerance (position)

Create Geometric Tolerance

Follow the routine to create a geometrical tolerance for selected geometry component:

- Select an applicable geometry component in the **3D View**. **Note.** To create a geometric tolerance for two or more geometry components, press CTRL key and point with a mouse on appropriate components.
- Right click and select **Geometric Tolerance** item in the appeared context sensitive menu.
- The **Create Geometric Tolerance** wizard appears in the **GD&T** view area.
- Enter needed parameters into the wizard fields and press the **Create** button. For more information see Create Geometric Tolerance Wizard.

The created tolerance feature control frame with the entered data appears in the **3D View** (see an example in Figure 1). **Note.** In case the geometry tolerance annotation is the first annotation for the selected feature, the start point of the leader line of the created feature control frame is the point, which a user points for selecting a component. In case a datum feature is created before or after a geometric tolerance creation, the annotations of a datum feature and a geometric tolerance are grouped (see Figure 2). If a group of geometric tolerances is created for the same feature, annotations of these geometric tolerances are combined (see Figure 3). In case the composite geometric tolerance is created, the first compartment of the tolerance frame is merged (see Figure 4).

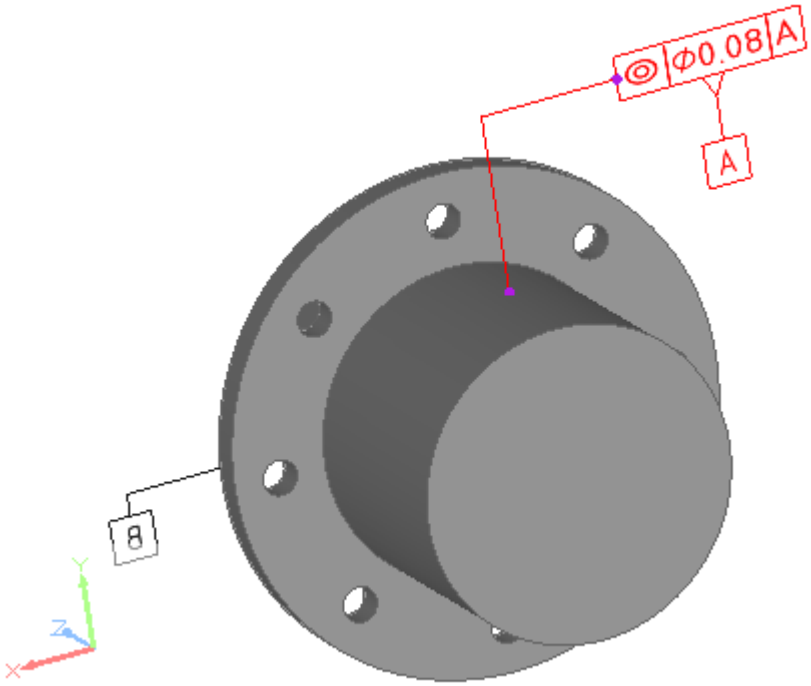

Figure 2. Grouped annotations (datum feature and coaxiality tolerance)

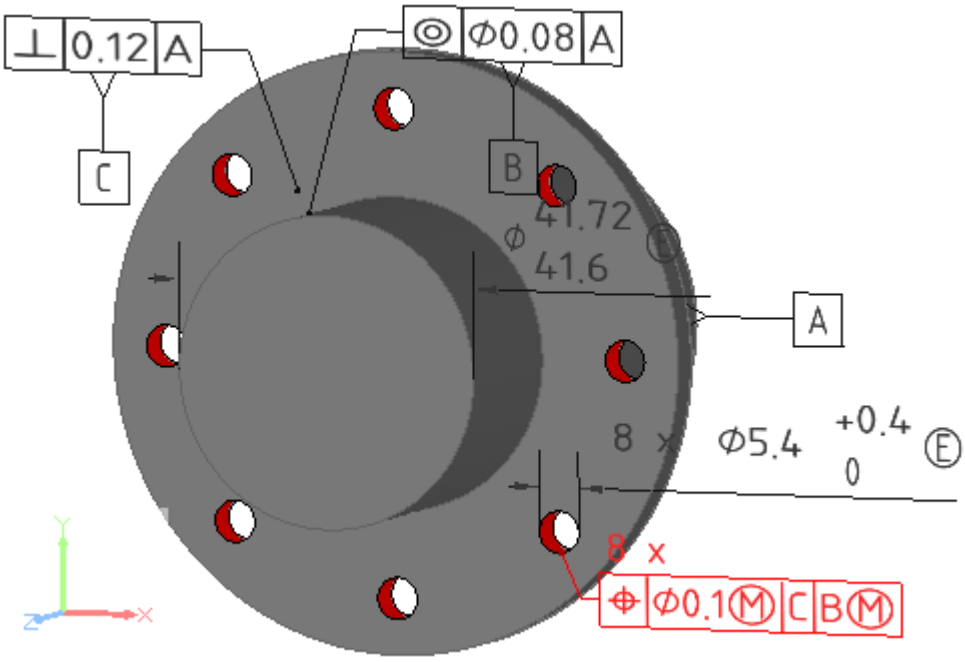

Figure 3. Combined annotation (position tolerance for 8 holes) with highlighted related surfaces

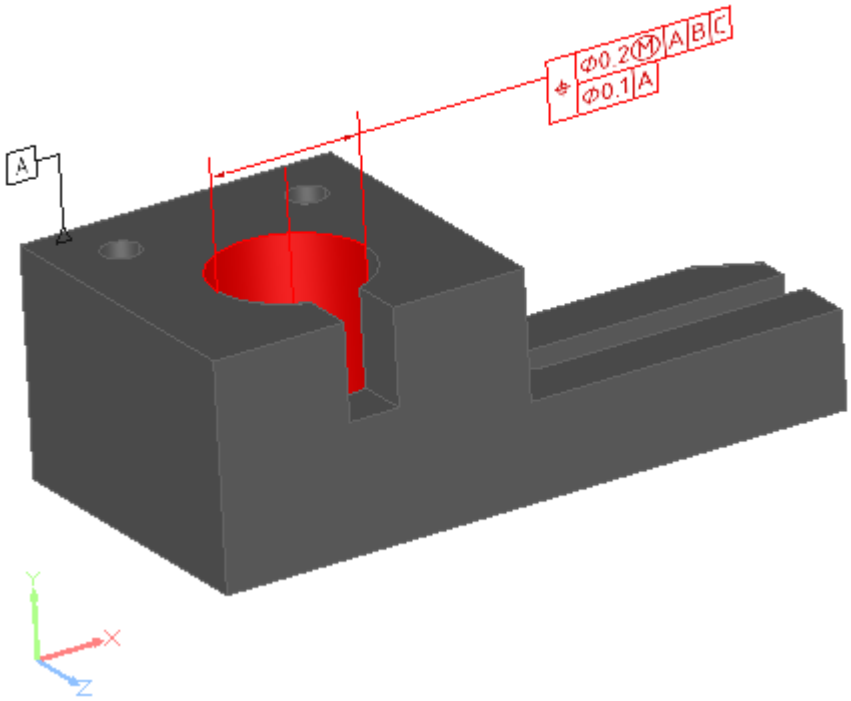

Figure 4. Composite geometric tolerance annotation frame with merged first compartment

Move Geometric Tolerance Annotation

When clicking on the feature control frame of a geometric tolerance, the control points are highlighted on the annotation lines (see purple points in Figure 1). The point, which is on the selected feature (edge, surface) can be moved along this feature.
Note. If the tolerance is applied to more than one component, this point can be moved along all selected features.
Another point, which is located on the feature control frame, can be moved up and down, left and right to adjust an applicable feature control frame position.
Note. In case of projected tolerance, the user can drag only on one side/edge, where projection end is specified.

To top

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9.2.14.5. Dimensions

Dimensioning is the process of adding measurement annotation to a geometry object. The dimensions, created in IDA-STEP **GD&T** view, are presented on a geometry model in the **3D View**. The following dimensions can be created in IDA-STEP:

- **Size** (radius, diameter, length):
  - *Radius dimension* (see Figure 1). The letter symbol R precedes the dimension value of a radius. The single terminator is used for presentation of radius dimension. It is indicated at the intersection of the dimension line and an arc. The arrowed head can be either inside or outside the outline of the feature (see Figure 1 a). In case the radius dimension goes outside of a feature (cylinder, arc, cone, sphere) it is attached to, the extra extension arc is added, which connects a dimension line of radius and closest point on the feature. (see Figure 1 b).

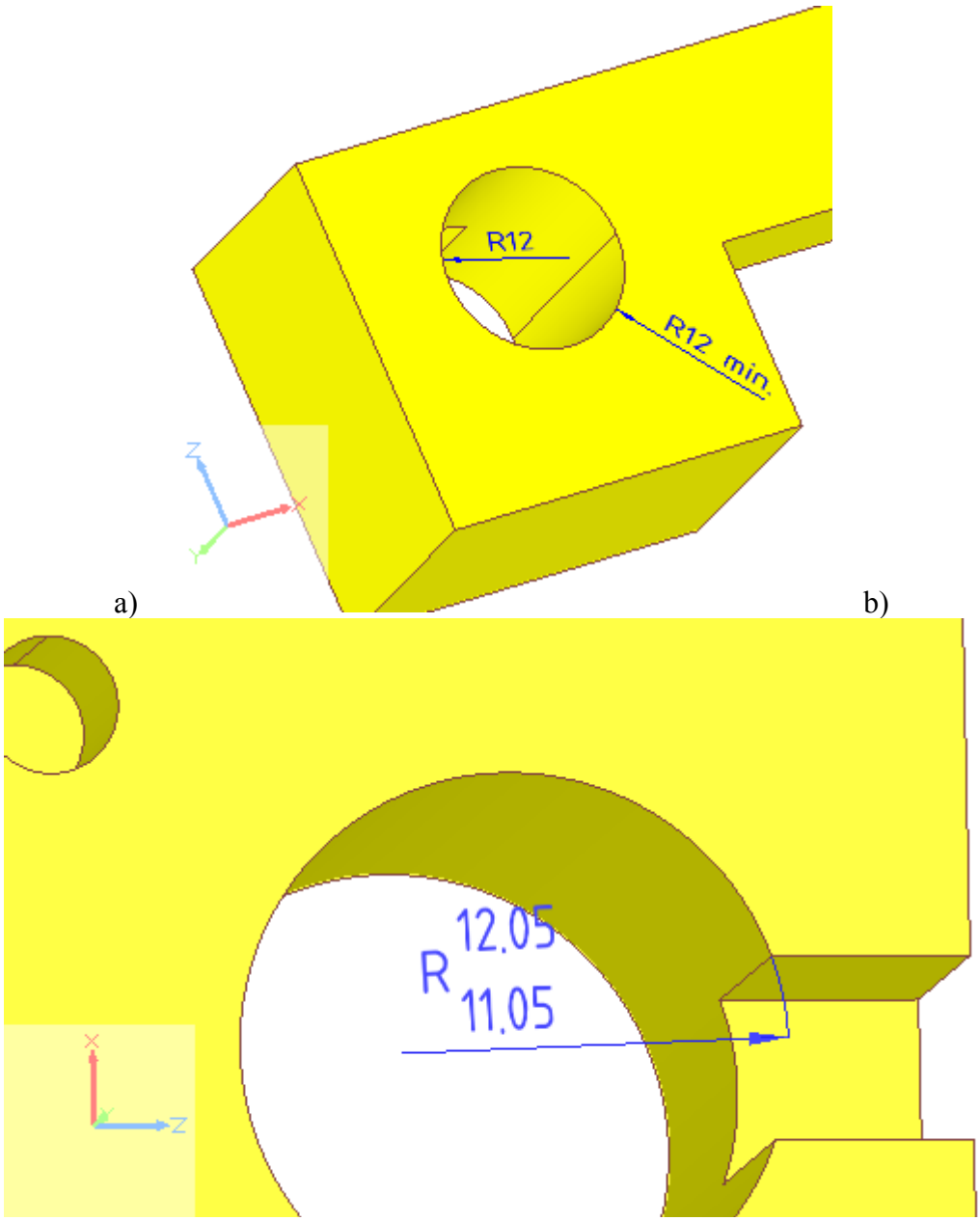
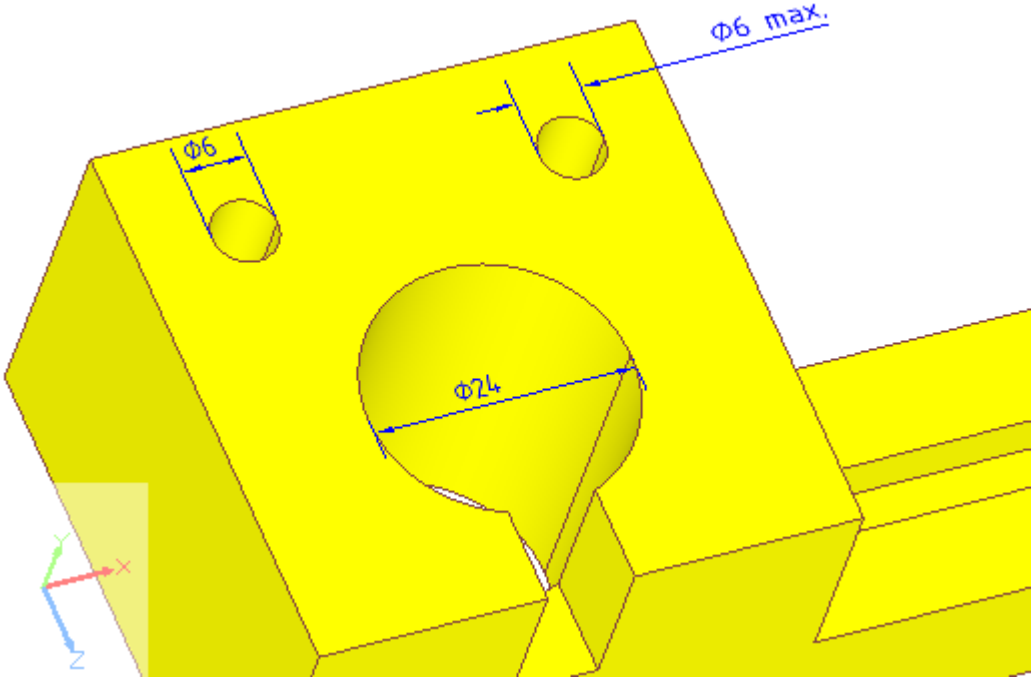


Figure 1. Radius dimension presentation in 3D View: **a)** radius dimension inside outline (i.e. 'R12') and radius limit dimension outside outline (i.e. 'R12 min.'); **b)** radius dimension with extra extension arc

- *Diameter dimension* (see Figure 2). The graphical symbol ∅ precedes the dimensional value. In case of the *Spherical diameter dimension*, the S letter precedes the symbol ∅.

a) 

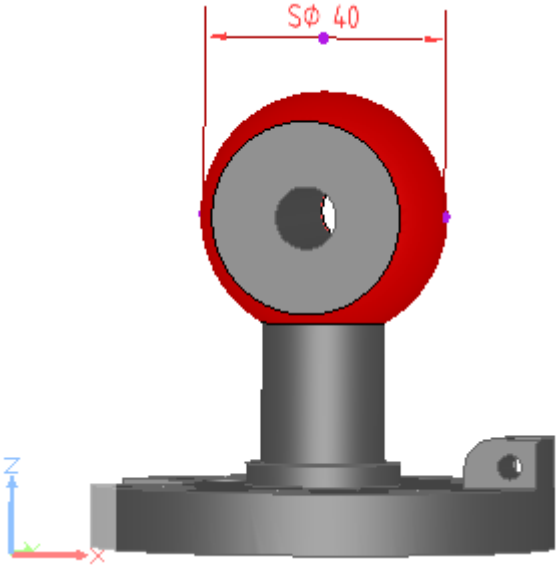
b) 

Figure 2. Diameter dimension presentation in 3D View: **a)** cylinder/circle diameter dimension, **b)** spherical diameter dimension

Diameter dimension in IDA-STEP can be presented in two different ways (see Figure 3):

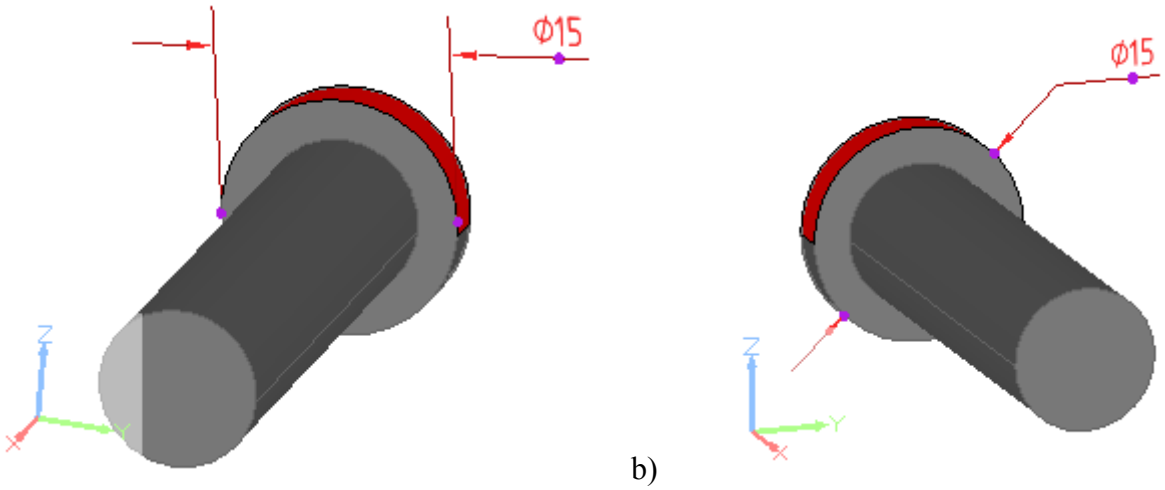
1. with two parallel leader curves, and
2. with only one leader curve in line with the diameter axis. **Note.** In case an annotation plane is in-line with a centre line of a circle/cylinder, this presentation is not possible, a dimension is presented with projection curves.

The default presentation of a diameter dimension is controlled via GD&T preferences by selecting a particular radio button: *always with projection curves* or *without projection curves when possible* (see Preferences -> GD&T -> Annotation Styles). The instantaneous presentation of the diameter dimension can be changed directly in the **3D View** by using CTRL key:

- Press CTRL key.
- Select a diameter dimension annotation in the **3D View** and drag the annotation using a mouse button.
- Release CTRL key and the mouse button.

Note. The presentation of the diameter dimension annotation will be changed to the default as soon as the

dimension is updated or dragged.



a) b)

Figure 3. Diameter dimension presentation in 3D View: **a)** with two parallel leader curves, **b)** with only one leader curve in line with the diameter axis

- *Arc length dimension* (see Figure 4). The graphical symbol ⌒ precedes the dimensional value.

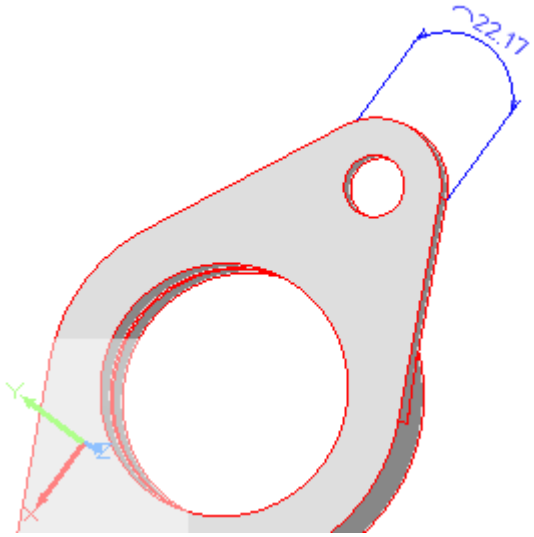


Figure 4. Arc length dimension presentation in 3D View

- *Size dimension with tolerance class*, defined in ISO 286 (see Figure 5).

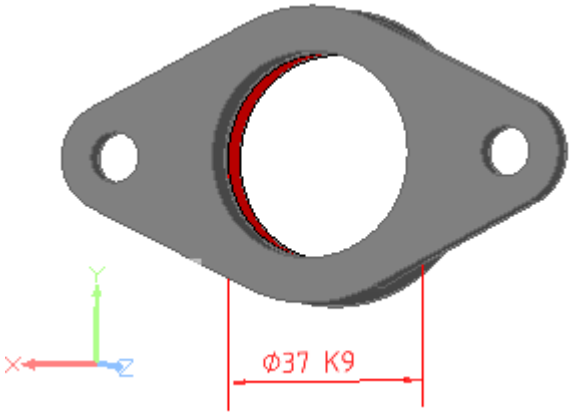


Figure 5. Size dimension with tolerance class ('K9') presentation in 3D View

- **Distance** dimensions (see Figure 6).

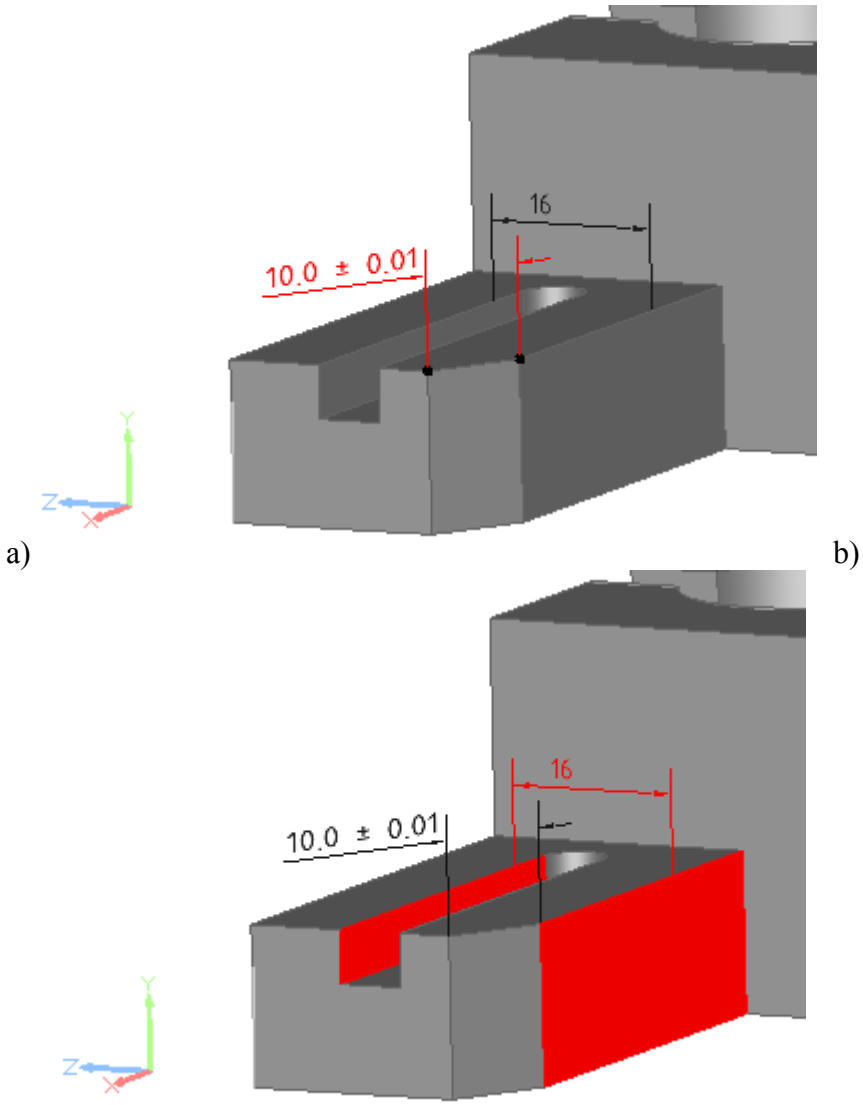


Figure 6. Presentation of distance dimensions in 3D View: a) between two vertex points and b) between two planes

- **Angular**:
  - *Angular dimension* (see Figure 7). The graphical symbol ° follows the dimensional value.

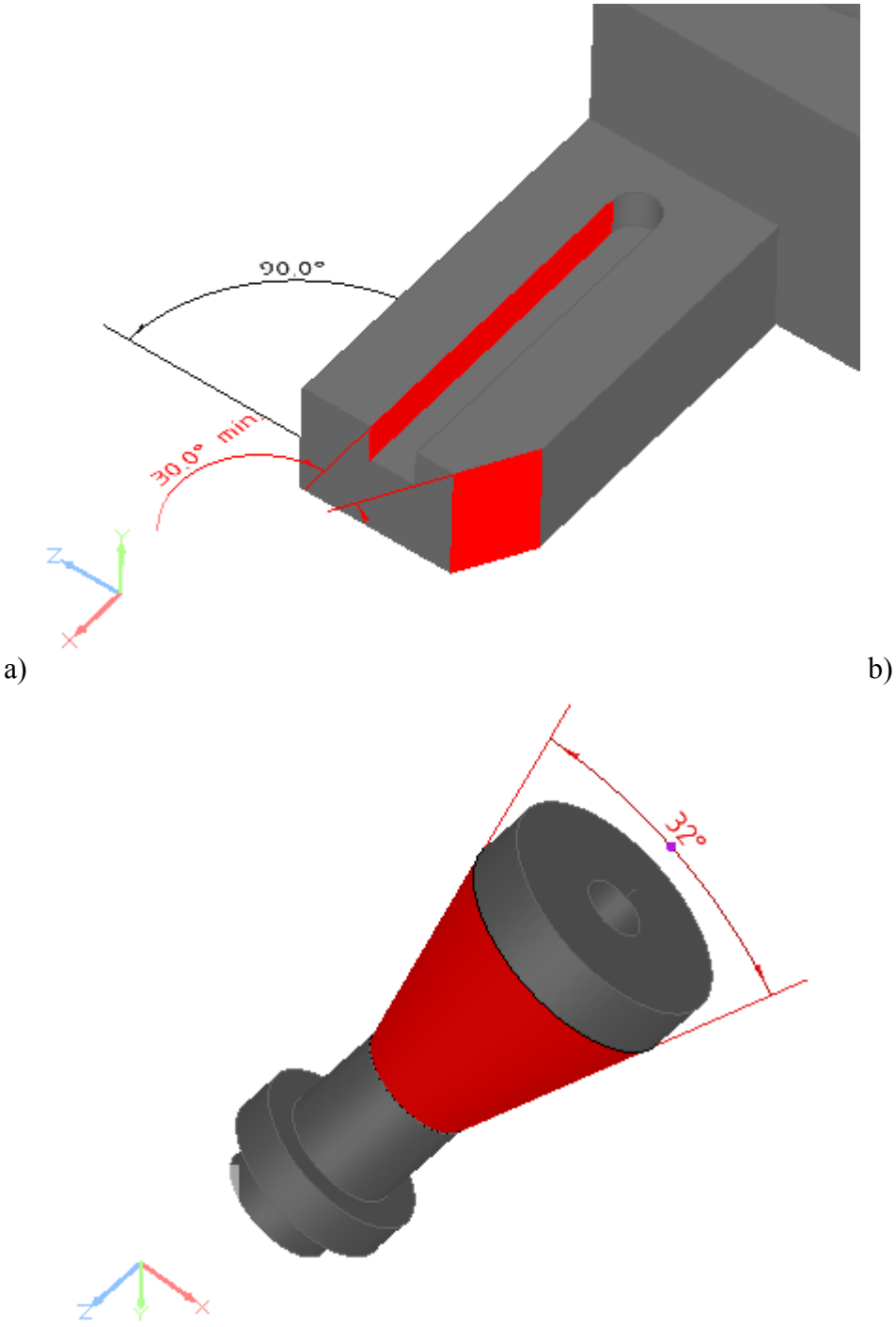


Figure 7. Angular dimension presentation in 3D View: **a)** between two not parallel planes, **b)** angular dimension for conical face

In case angular dimensions (ISO), the text is oriented according positive direction on X axis of an annotation plane. A text 'swaps' depending on the angle of the text and that positive X axis (see Figure 8). In this example, the positive reading direction of text (so positive X direction of annotation plane) is equal to Z axis of 3D object.

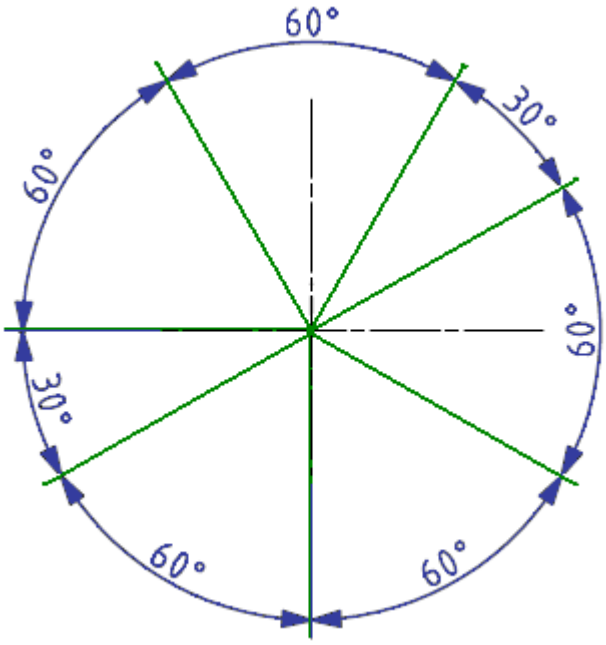


Figure 7. Orientation of angular dimension (ISO)

In case an annotation ends on the feature, the projection lines of this annotation are skipped (see Figure 9).

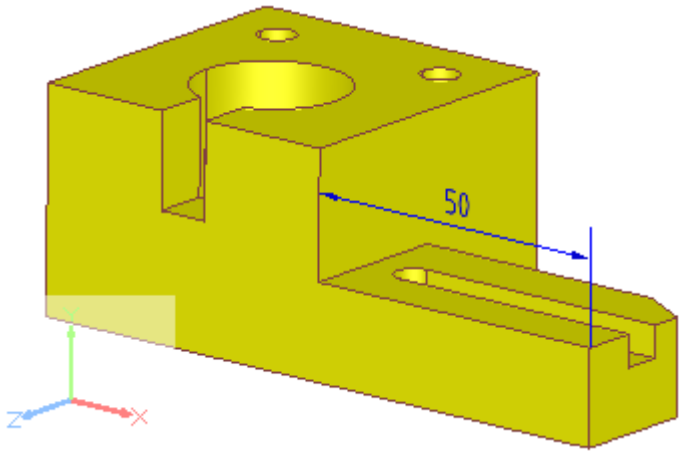


Figure 9. Annotation with skipped projection line (no projection line on the left side of annotation)

Create Dimension

To create a linear or an angular dimension:

- Select an applicable surface element on a geometry object by pointing with a mouse.
- Right click and select the **Dimension -> Angle/ Size/ Distance** item in the appeared context menu. **Note.** In case there is no possibility to create any dimension for the selected geometry element, the **New Dimension** sub menu items are disabled.
- The **Angular/Size Dimension/Distance** wizard appears in the **GD&T** view details area (see Angular/Distance/Size Dimension Wizard).

The created dimension is displayed in the **3D View** (dimension text, line and extension line). To move dimension annotation lines:

- Click with a mouse on the annotation text area. **Note.** To turn annotation into editing mode, the particular selection modes should be switched on:

    - In case the **Wire Frame and Solid** or **Solid**, or **Triangle Style** is currently set *On*, the **Face/Surface** or **Edge/Curve Selection Mode** should be switched *On*.
    - In case the **Wire Frame** or **Wire Frame with Hidden Line Remove Style** is currently switched *On*, the **Edge/Curve** selection mode should be switched *On*.
- Selection points are highlighted on the annotation lines. To adjust an applicable annotation line position, click with a mouse on a control point and drag the mouse into needed direction.

For more information see GD&T -> Dimensions.

To top

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9.2.14.6. Annotation Plane

An *annotation plane* in IDA-STEP is a conceptual plane, which contains annotations.
Follow the routine to create an annotation plane in the **3D View**:

- Select a planar surface in the **3D View** to create an annotation plane, which is on the selected surface (see Figure 1). **Note.** It is not required selection. In case no planar surface is selected, more parameters have to be entered manually.
- Right click and select the **New Annotation Plane** item in the appeared context sensitive menu .
- The **Annotation Plane** wizard appears in the **GD&T** view (see Annotation Plane Wizard).

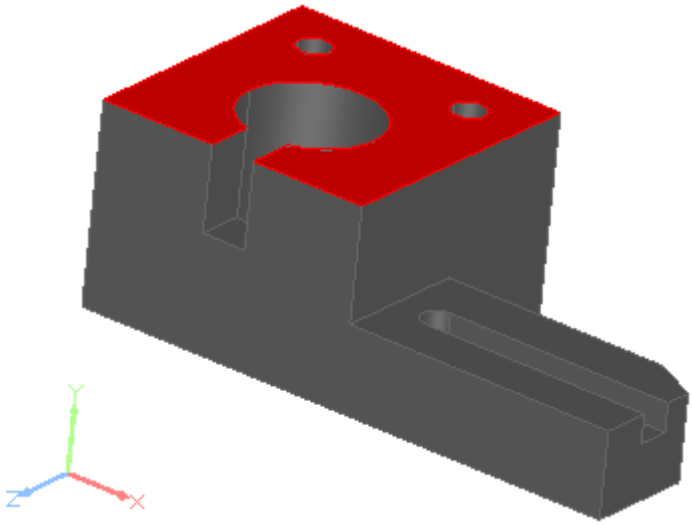

Figure 1. Selected surface on which annotation plane should be created

The created annotation plane is not visible in the **3D View**. To view the created annotation plane, select the recently created annotation plane entry in the **GD&T** view table. The selected annotation plane is highlighted in the **3D View** (see Figure 2).
Note. An annotation plane is defined by Cartesian point (origin point of annotation plane) and three vectors. Those three axes are shown at the origin point of the annotation plane in the **3D View**. The size of axes is determined by the 3D view preference *Coordinate system size* (refer Clause Preferences -> 3D View). Axes are visible for an annotation plane only when it is selected.

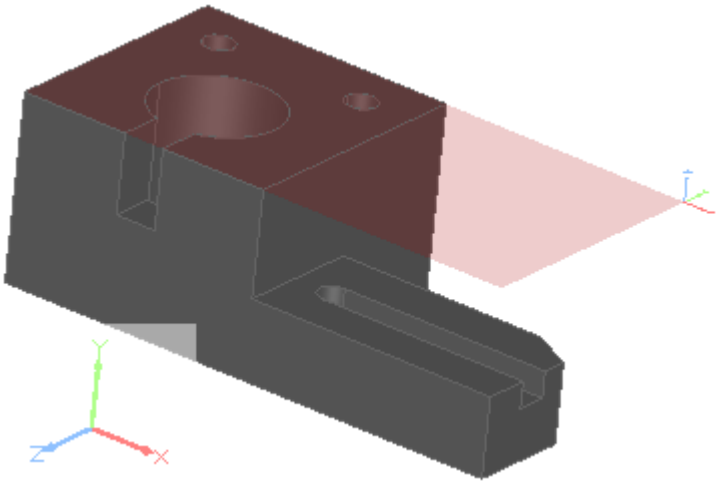

Figure 2. Created annotation plane and coordinate axes at origin point of annotation plane

In most cases the annotation plane is automatically created, while creating some GD&T element, which has an association with an annotation plane and there is no suitable previously created annotation plane. See Figure 3 - where an annotation plane ('AP1') is created for a datum feature ('A' and 'B' reuses the same annotation plane), size dimension and geometric tolerance annotation; and Figure 4, where a new annotation plane ('AP2') is created for a datum feature 'C', size dimension and position tolerance. **Note.** In these examples the origin point of annotation plane is hidden inside geometry, so the coordinate axes of the selected annotation plane are not visible.

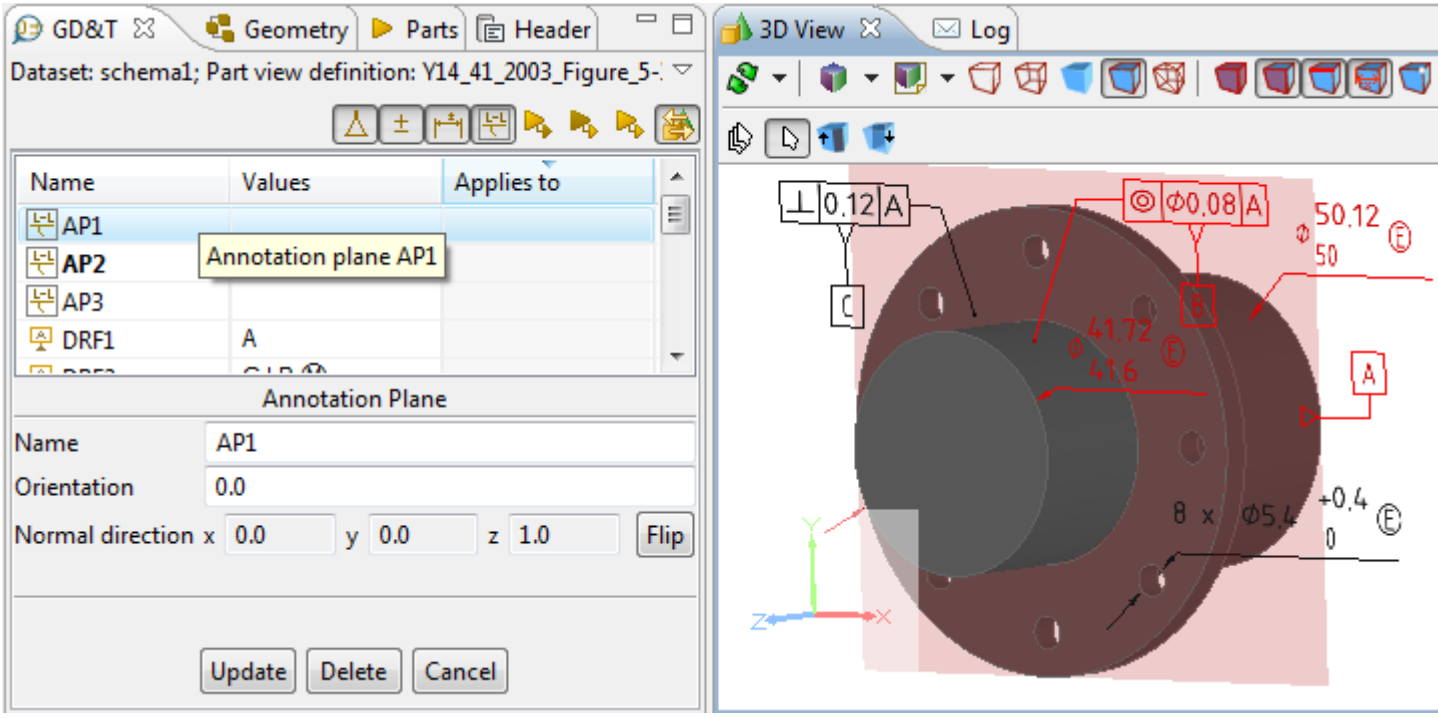

Figure 3. Annotation plane 'AP1' with the related annotation (both highlighted in 3D View)

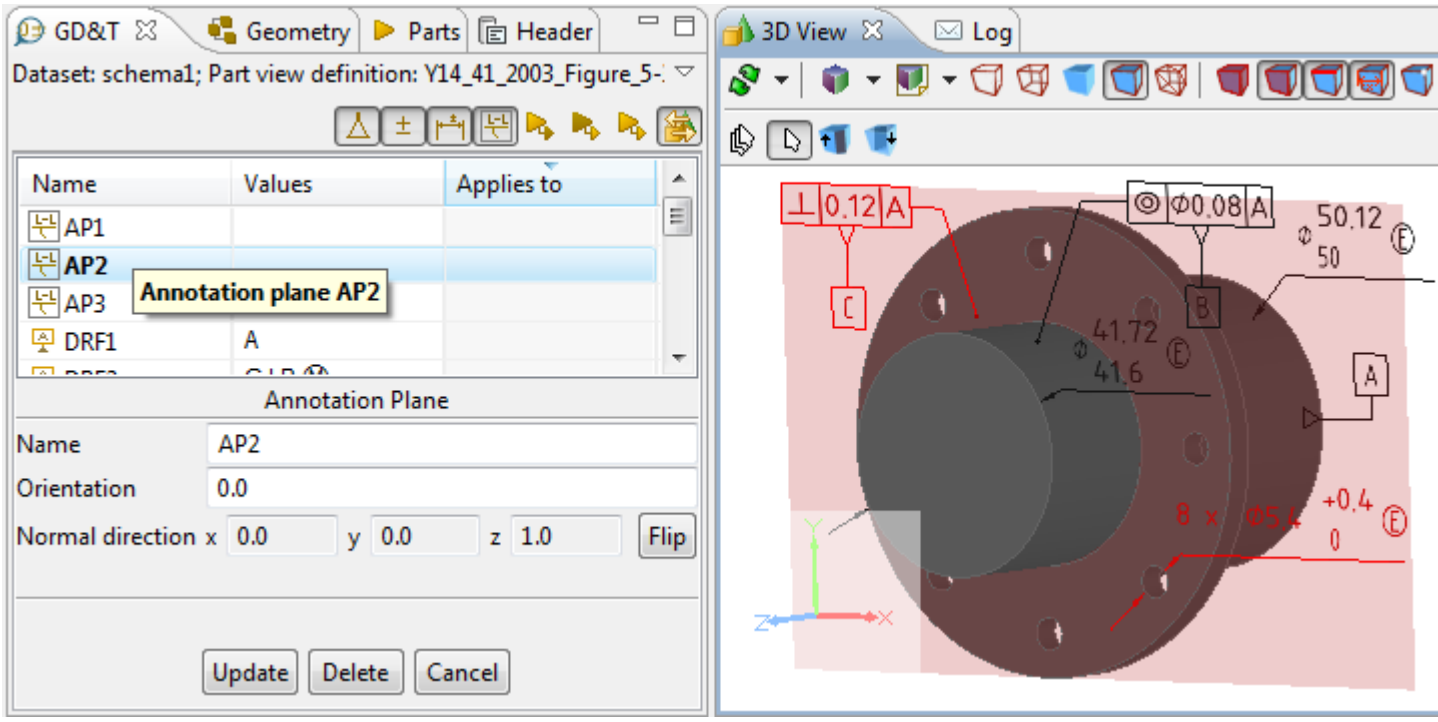

Figure 4. Active annotation plane 'AP2' with related annotations

See also:

GD&T -> Annotation Plane;
3D Viewer -> Style/Annotation Model -> Change active and bring to front Annotation Plane

To top

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9.2.14.7. Text Annotation

A *Text Annotation* in IDA- STEP is a 3D annotation with a free text, one or several lines (see Figure 1).

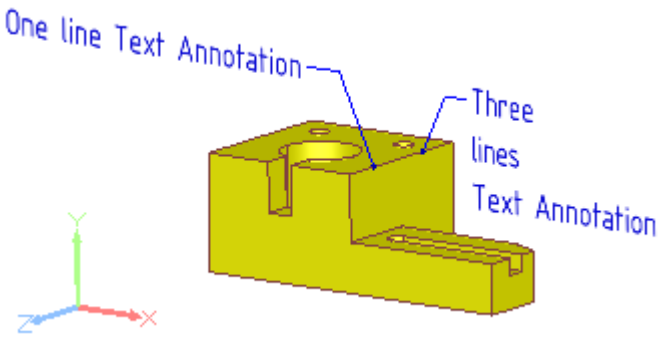

Figure 1. Text annotations in 3D View (one line and three lines)

Create text annotation

Follow the routine to create a text annotation in the **3D View**:

- Point with a mouse on an applicable geometry component in the **3D View**. **Note.** The selected point (even if no point selection mode is switched on) will be assumed as a start point of the leader line of the text annotation to be created.

- Right click and select **Text Annotation** item in the appeared context sensitive menu.
- The **Create Text Annotation** wizard page appears in the **GD&T** view area. Enter needed data in the wizard fields and click **Create** button. For more information see Create Text Annotation Wizard.

The text annotation is created and appears in the **3D View** (see Figure 1). The leader line of the created text annotation starts at the point, the user clicked while selecting a geometry component.

Move leader line

There is a possibility to move a text annotation leader line on the selected feature/entire shape and modify leader line:

- Click with a mouse on the text annotation. Two points on the leader lines are highlighted.
- Click with a mouse on the point:
  - which is in contact with the feature object the text annotation is applied to (i.e. *line*) - to change the text annotation position on the feature.
  - which is in contact with the text - to modify leader line.
- Drag with a mouse to the desired direction. Release the mouse button to set the position.

Note. In case a text annotation is applied to the entire shape, the point of a leader line, which is in a contact with the shape, can be moved all over the entire shape.

To top

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9.2.14.8. Shape Aspect

A *shape aspect* is an element of the shape of a product. A *Composite Shape Aspect* in IDA-STEP is a shape aspect, that associates aspects of the product shape for a specific purpose.
In the **3D View** there is a possibility to create a single shape aspect and a composite shape aspect.

Create single shape aspect

To create a single shape aspect:

- Point with a mouse on any geometry element (point, line, plane and etc).
- Right click and select the **Shape Aspect** item in the appeared context sensitive menu (see Figure 1).

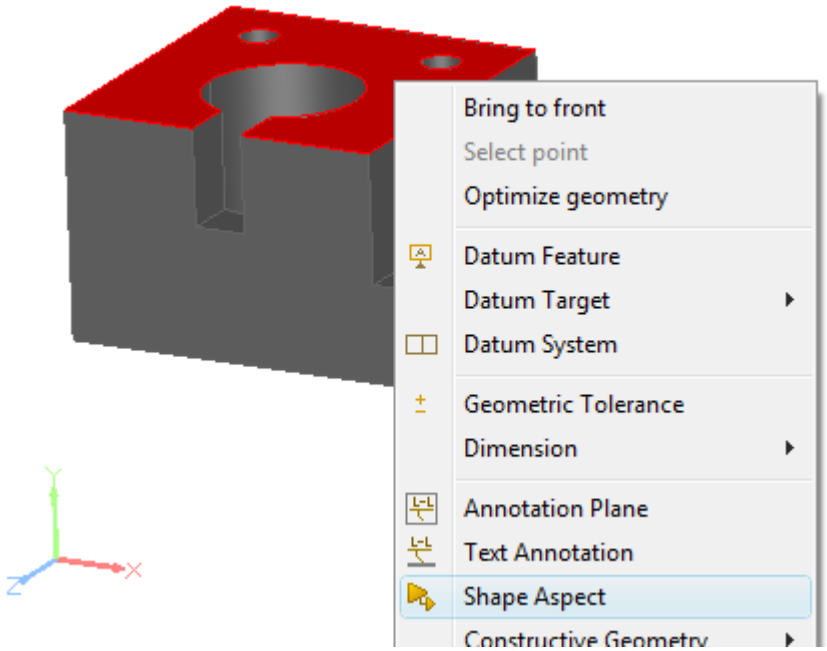

Figure 1. Selected plane and context sensitive menu item to create a shape aspect

The shape aspect is created for the selected geometry element. It is not visible any changes in the **3D View**, but a new entry appears in the **GD&T** view table.
For more information about the shape aspects in the **GD&T** view, see Clause GD&T -> Shape Aspects.

Create composite shape aspect

To create a composite shape aspect:

- Select two or more geometry elements on the geometry object in the **3D View**.
- Right click and select the **Shape Aspect** item in the appeared context sensitive menu (see Figure 2).

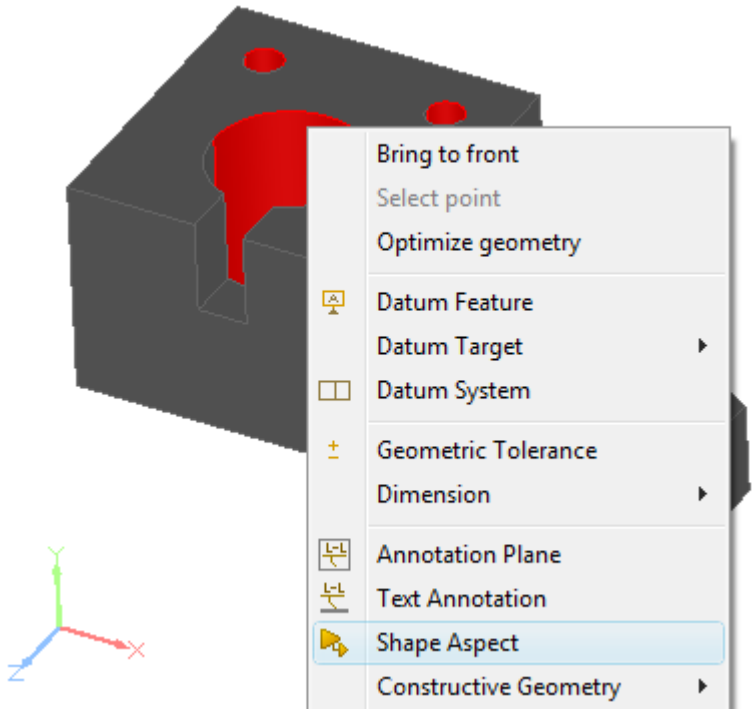

Figure 1. Selected three cylinders and context sensitive menu item to create a composite shape aspect

A new entry of the composite shape aspect (labeled with CS letters and a number) in the **GD&T** view table. **Note.** It is additionally created a shape aspect for each selected geometry element, which has no associated shape aspect.
For more information about the composite shape aspect, see Clause GD&T -> Composite Shape Aspect.

To top

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9.2.14.9. Constructive Geometry Elements

The **3D View** displays constructive geometry elements, which are assigned to construction geometry models, displayed in **Geometry Details** view. IDA-STEP presents possibility to create simple constructive geometry elements, such as line and plane, and derived shape elements (apex, symmetry point, line and plane, common line and plane, intersection, offset, extension, perpendicular line and plane, tangent plane). Those elements are created by using wizards, which appear in the **GD&T** view.
Created constructive geometry elements are displayed in all above mentioned views:

- **3D View** - on the geometry object and highlighted together with the referenced geometry element.
- **Geometry Details** view - under Construction Geometry Models -> Model branch for the currently active model. In case no active model is selected or there are no models at all, a new model is created under the Construction Geometry Models branch. Selection of the created constructive geometry element initiates a highlighting in the **GD&T** view and in the **3D View.**
- **GD&T** view - a new entry appears in the table.If this entry is selected, the particular wizard dialog appears in the details area under the table and the element is highlighted in the **3D View** together with referenced geometry element, and in the **Geometry Details** view.

For more information, see Geometry Details -> Construction Geometry Models, GD&T -> Constructive Geometry.

To top

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9.2.14.9.1. Apex

An *apex* in IDA-STEP is the highest vertex of the cone.
To create an apex in the **3D View**:

- Select a cone surface of the geometry object.
- Right click and select the **New Constructive Geometry -> Apex** item in the appeared context sensitive menu.

After the menu item is selected the apex is instantly created.
Note. The apex can be rather far away from the cone face. In cases the resulting apex is not visible in the **3D View**, it is recommended to zoom out the 3D part to view the apex point.

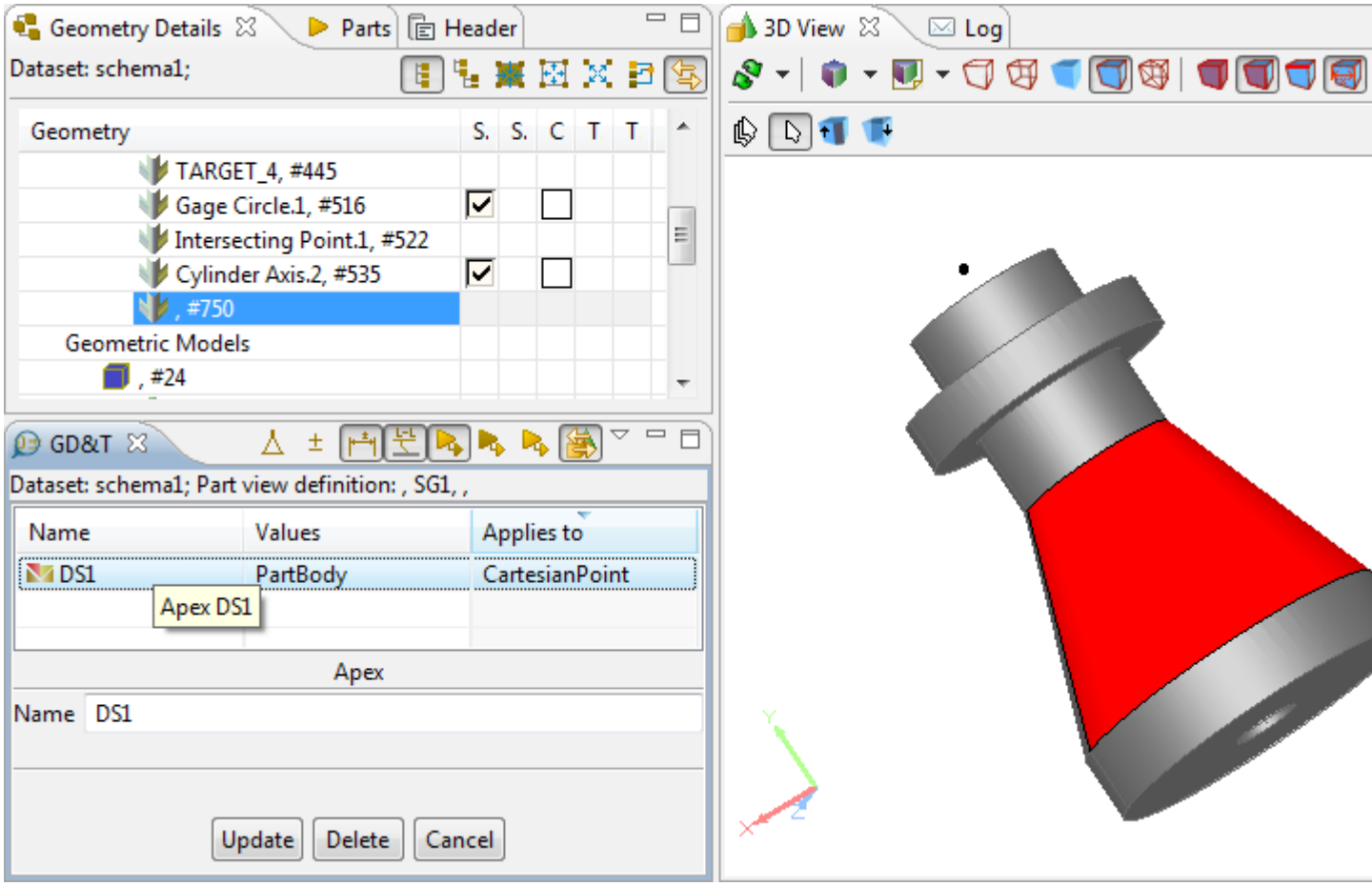

Figure 1. Apex (selection in GD&T view) - highlighted in Geometry Details view under the construction geometry model branch, GD&T view (table and wizard) and 3D View, together with the referenced cone surface

For more information, see GD&T -> Create Apex.

To top

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9.2.14.9.2. Symmetry Axis

A symmetry axis in IDA-STEP is a center line of a cylinder, cone, toroid and revolution.
To create a symmetry axis in the **3D View**:

- Select a cylinder, cone, toroid or revolution.
- Right click and select the **New Constructive Geometry -> Symmetry Axis** item in the appeared context sensitive menu.

After the menu item is selected the symmetry axis for the selected geometry element is instantly created.

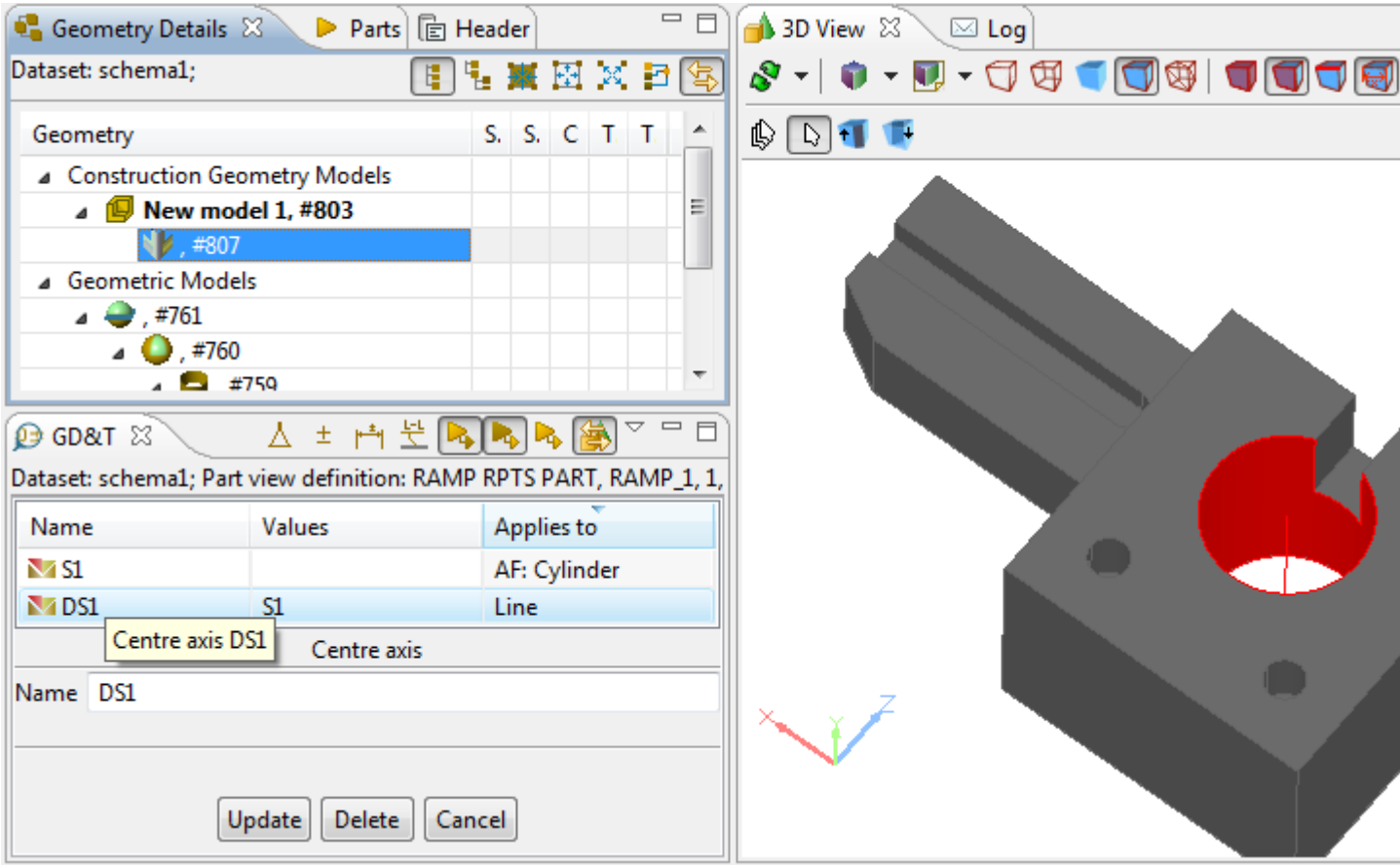

Figure 1. Symmetry axis of cylinder (selection in GD&T view) - highlighted in Geometry Details view under the construction geometry model branch, GD&T view (table and wizard) and 3D View, together with the referenced cylinder surface

See also GD&T -> Create Symmetry Axis.

To top

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9.2.14.9.3. Symmetry Plane

A symmetry plane in IDA-STEP is a center plane between two planar surfaces.
To create a symmetry plane in the **3D View**:

- Select two planar surfaces.
- Right click and select the **New Constructive Geometry -> Symmetry Plane** item in the appeared context sensitive menu.

After the menu item is selected the symmetry plane of selected planar surfaces is instantly created.

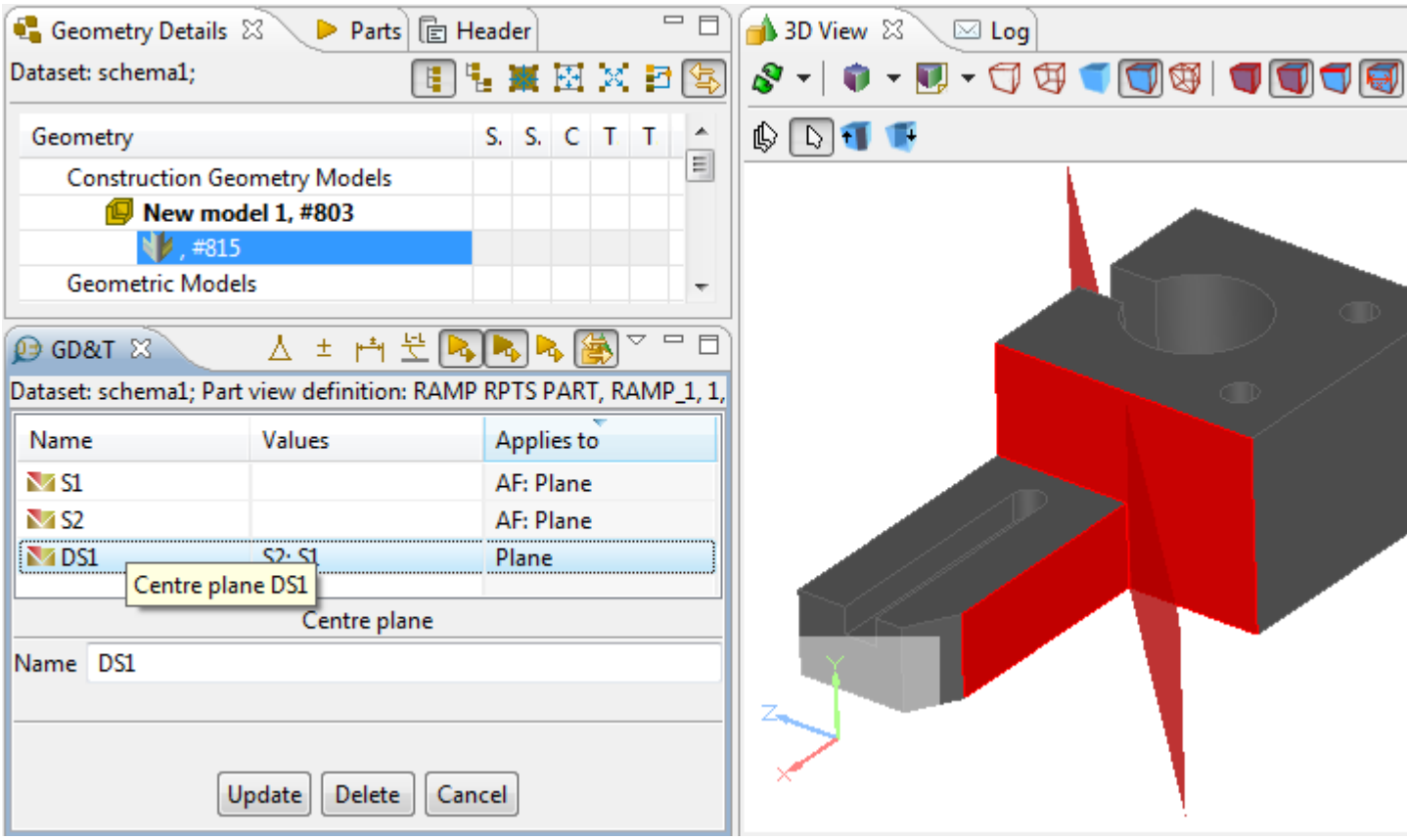

Figure 1. Symmetry plane (selection in GD&T view) - highlighted in Geometry Details view under the construction geometry model branch, GD&T view (table and wizard) and 3D View, together with the referenced planes

See also GD&T -> Symmetry Plane.

To top

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9.2.14.9.4. Symmetry Point

A *symmetry point* in IDA-STEP is a center point of a circle or a sphere.
To create a symmetry point in the **3D View**:

- Select a circle line or sphere surface.
- Right click and select the **New Constructive Geometry -> Symmetry Point** item in the appeared context sensitive menu.

After the menu item is selected the symmetry point of the selected circle or sphere is instantly created.

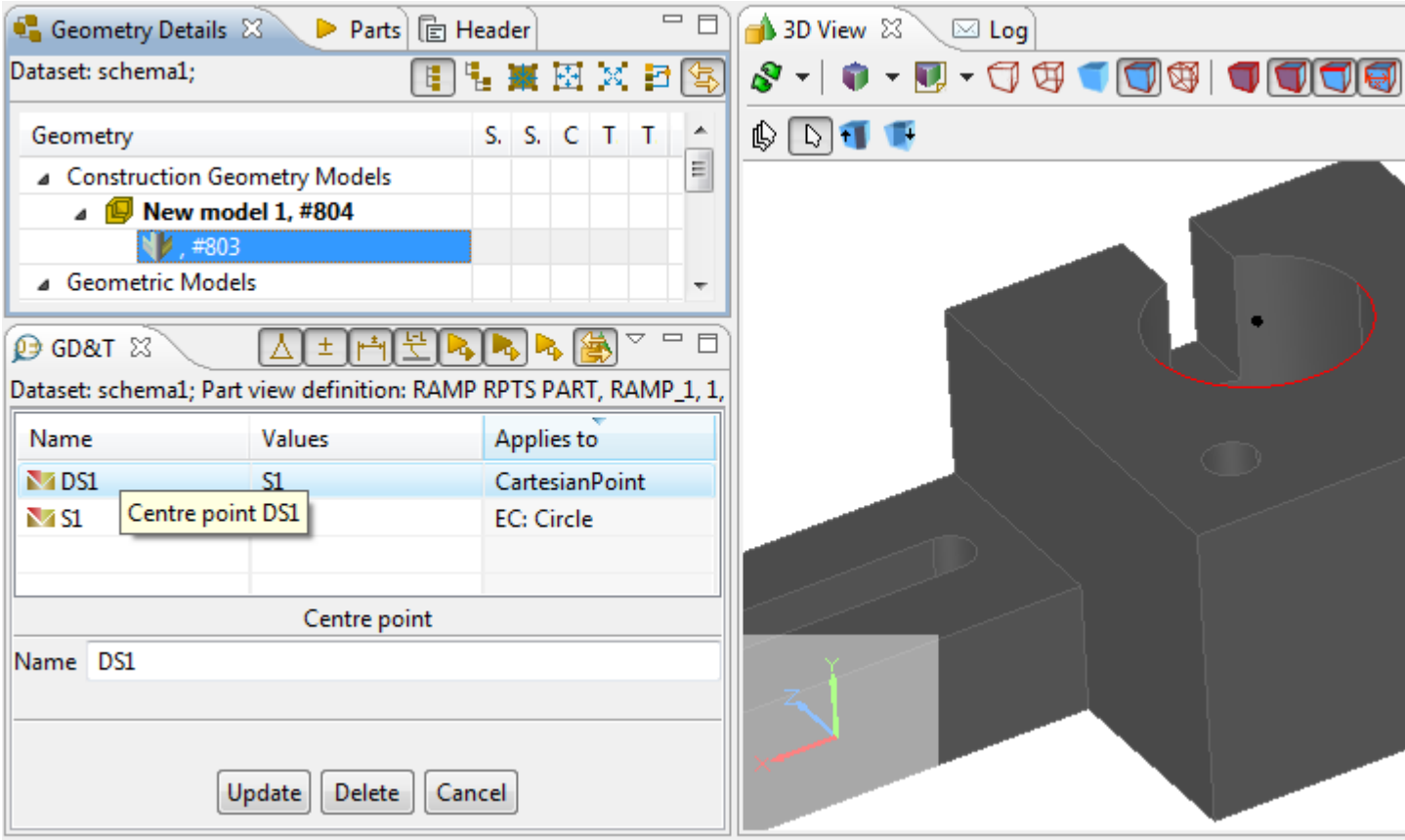

Figure 1. Symmetry point (selection in GD&T view) - highlighted in Geometry Details view under the construction geometry model branch, GD&T view (table and wizard) and 3D View, together with the referenced circle

See also GD&T -> Create Symmetry Point.

To top

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9.2.14.9.5. Common Line

A *common line* in IDA-STEP is a line, which goes through two selected points.
To create a common line in the **3D View**:

- Select two points on a geometry object.
- Right click and select the **New Constructive Geometry -> Common Line** item in the appeared context sensitive menu.

After the menu item is selected the common line through the selected points is instantly created.

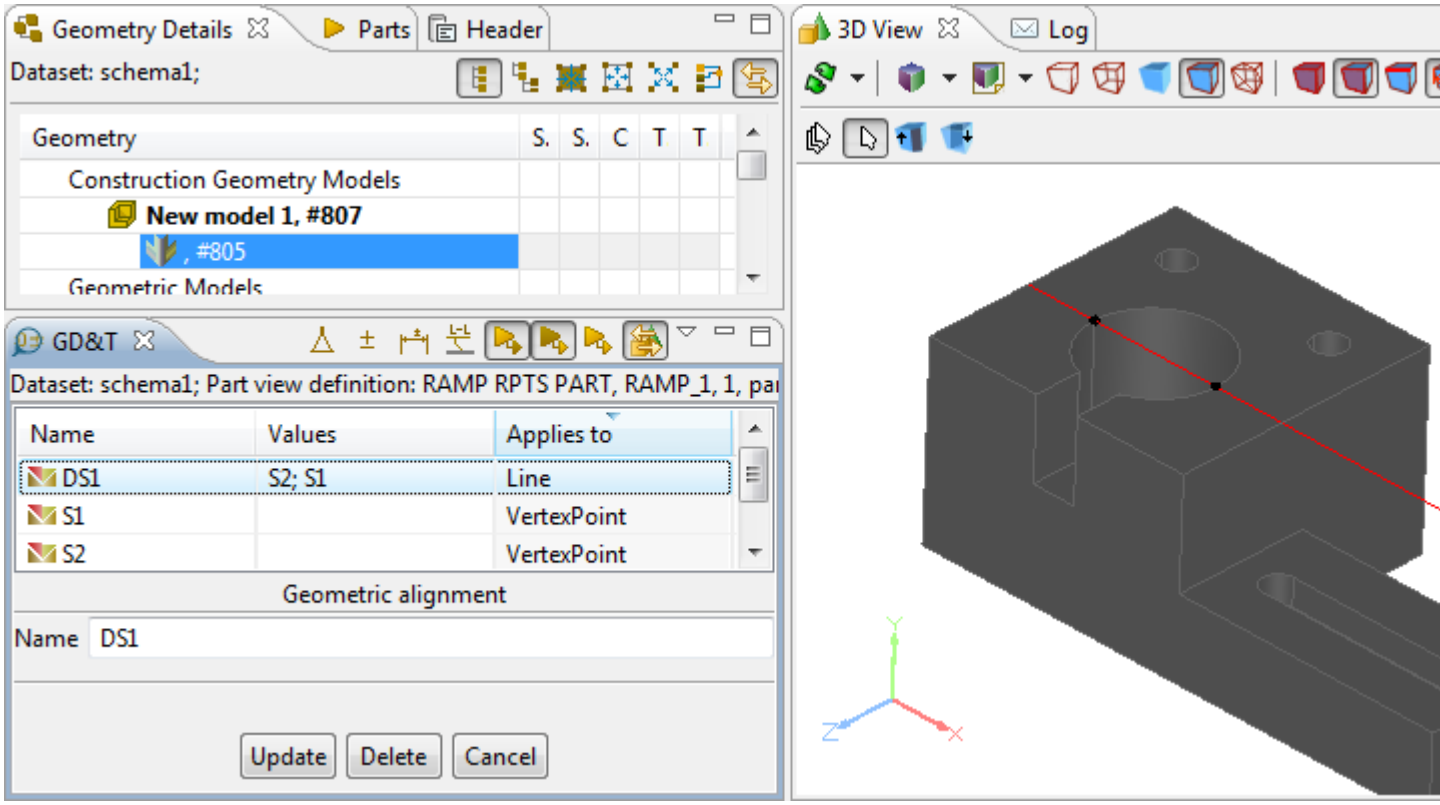

Figure 1. Common line (selection in GD&T view) - highlighted in Geometry Details view under the construction geometry model branch, GD&T view (table and wizard) and 3D View, together with referenced points

See also GD&T -> Create Common Line.

To top

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9.2.14.9.6. Common Plane

A *common plane* in IDA-STEP is a plane, which goes through:

- two **lines**,
- a **line** and a **point**.

To create a common plane in the **3D View**:

- Select two lines or a line and a point on a geometry object.
- Right click and select the **New Constructive Geometry -> Common Plane** item in the appeared context sensitive menu.

After the menu item is selected the common plane through the selected elements is instantly created.

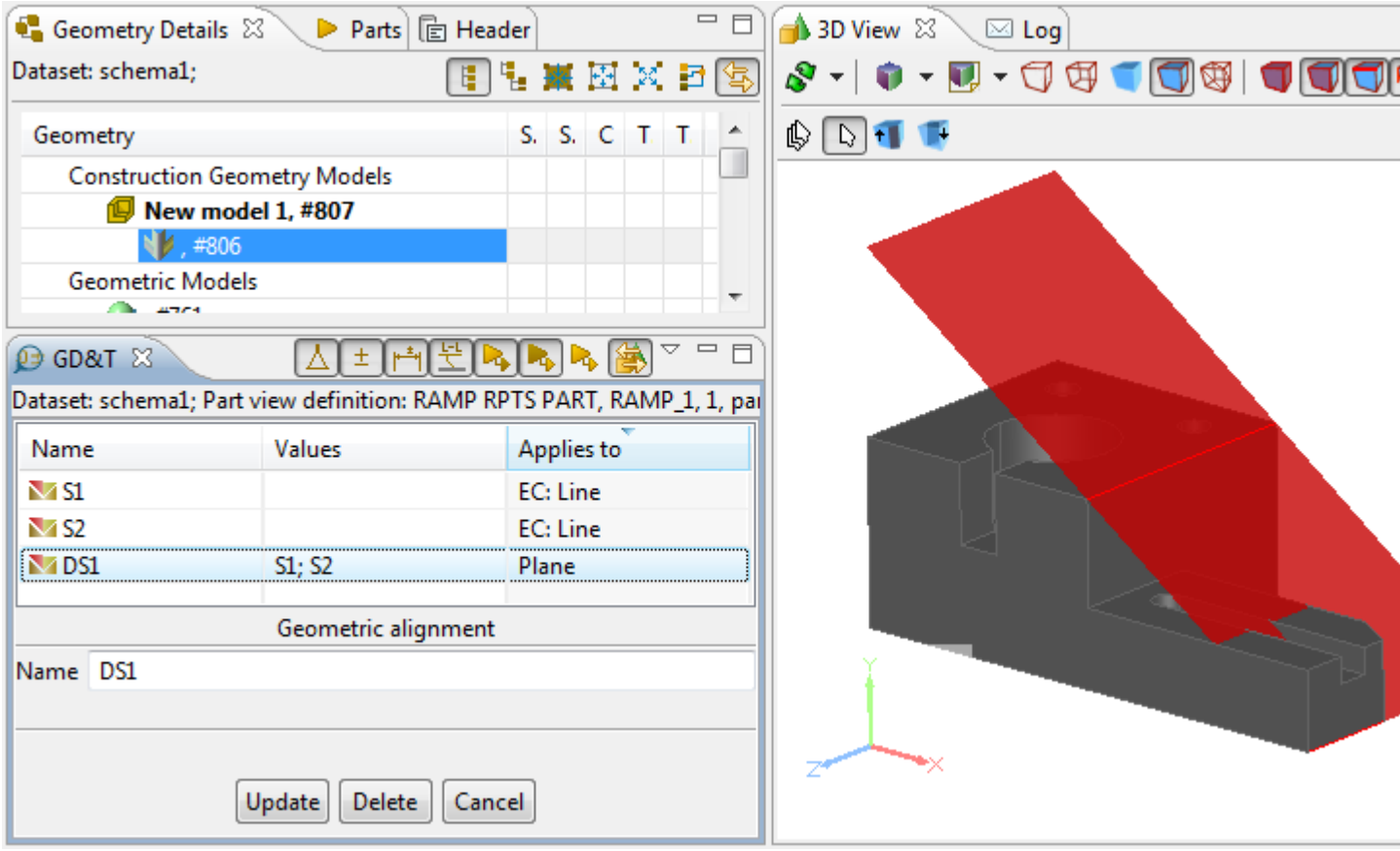

Figure 1. Common plane (selection in GD&T view) - highlighted in Geometry Details view under the construction geometry model branch, GD&T view (table and wizard) and 3D View, together with referenced two lines

See also GD&T -> Create Common Plane.

To top

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9.2.14.9.7. Intersection

An intersection in IDA-STEP is an intersection between two elements:

- two **lines** - the result is a Cartesian *point*, which is an intersection of two non parallel lines.
- two **planes** - the result is a *line*, which is an intersection of two non parallel planes.
- **plane** and **cylinder** -the result is a *circle*, an *ellipse* or a *line*:
  - in case the selected plane goes across the cylinder, an intersection is a *circle* or an *ellipse*.
  - in case the selected plane goes along the cylinder, an intersection is a *line* (the result can be one or two lines).
- **plane** and **line** - the result is a Cartesian *point*, which is an intersection of non parallel objects - line and plane.
- **line** and **cylinder** - the result is a Cartesian *point*, which is an intersection of non parallel objects - line and cylinder.
- **line** and **sphere** - the result is a Cartesian *point*, which is an intersection of line and sphere.
- **plane** and **sphere** - the result is a *circle* or a Cartesian *point*:
  - in case the selected plane crosses the sphere, an intersection is a *circle*.
  - in case the selected plane is a tangent plane for the selected sphere, an intersection is a Cartesian *point*.

To create an intersection in the **3D View**:

- Select two elements (see the above list) on a geometry object.
- Right click and select the **New Constructive Geometry -> Intersection** item in the appeared context sensitive menu.

After the menu item is selected the intersection of selected elements is instantly created.

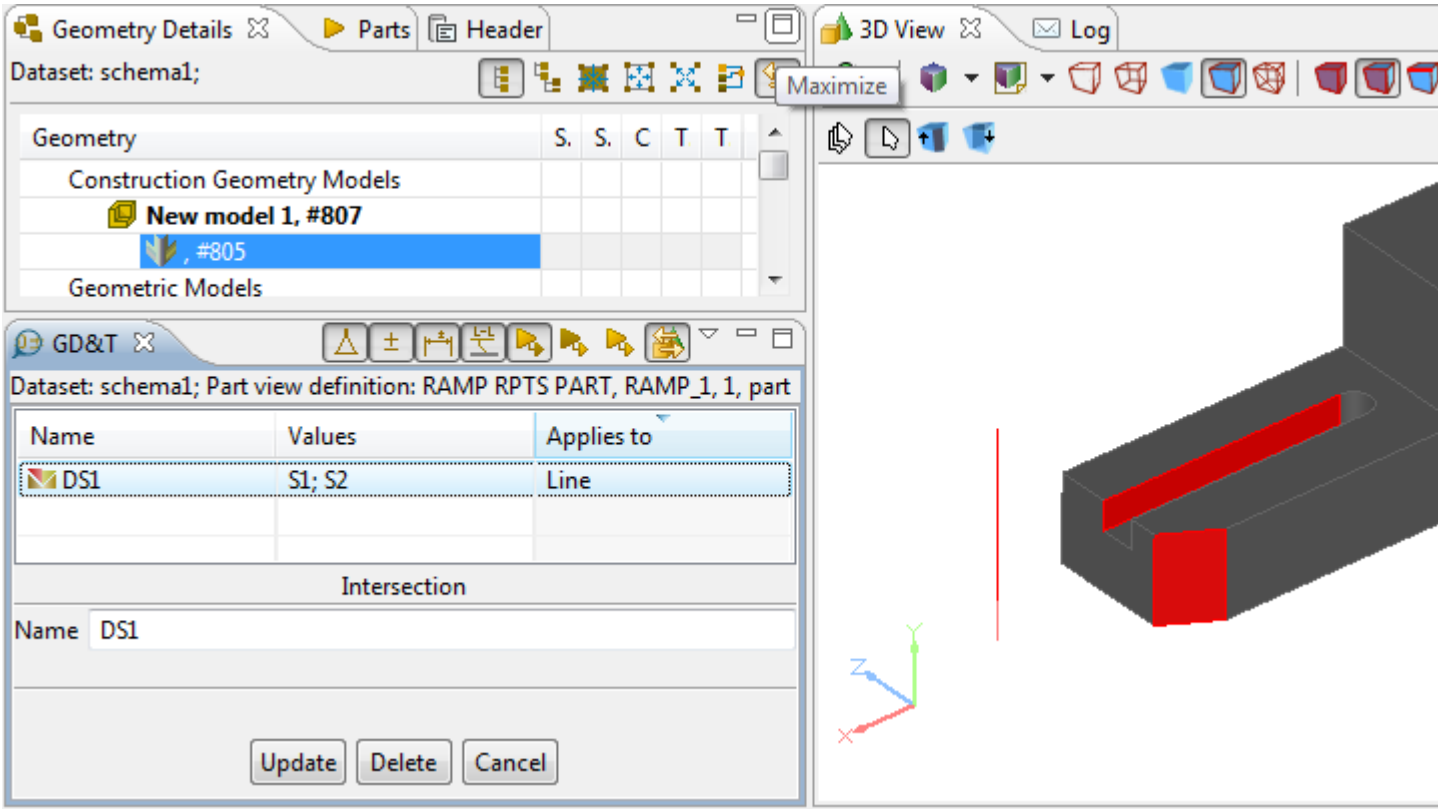

Figure 1. Intersection line (selection in GD&T view) - highlighted in Geometry Details view under the construction geometry model branch, GD&T view (table and wizard) and 3D View, together with referenced planes

See also GD&T -> Create Intersection.

To top

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9.2.14.9.8. Offset

An *offset* in IDA-STEP is an object (plane, line or point) whose shape parallels the shape of the original object.
To create an offset in the **3D View**:

- Select an applicable object.
- Right click and select the **New Constructive Geometry -> Offset** item in the appeared context sensitive menu.
- Further details, how to set parameters for an offset, see GD&T -> Create Offset.

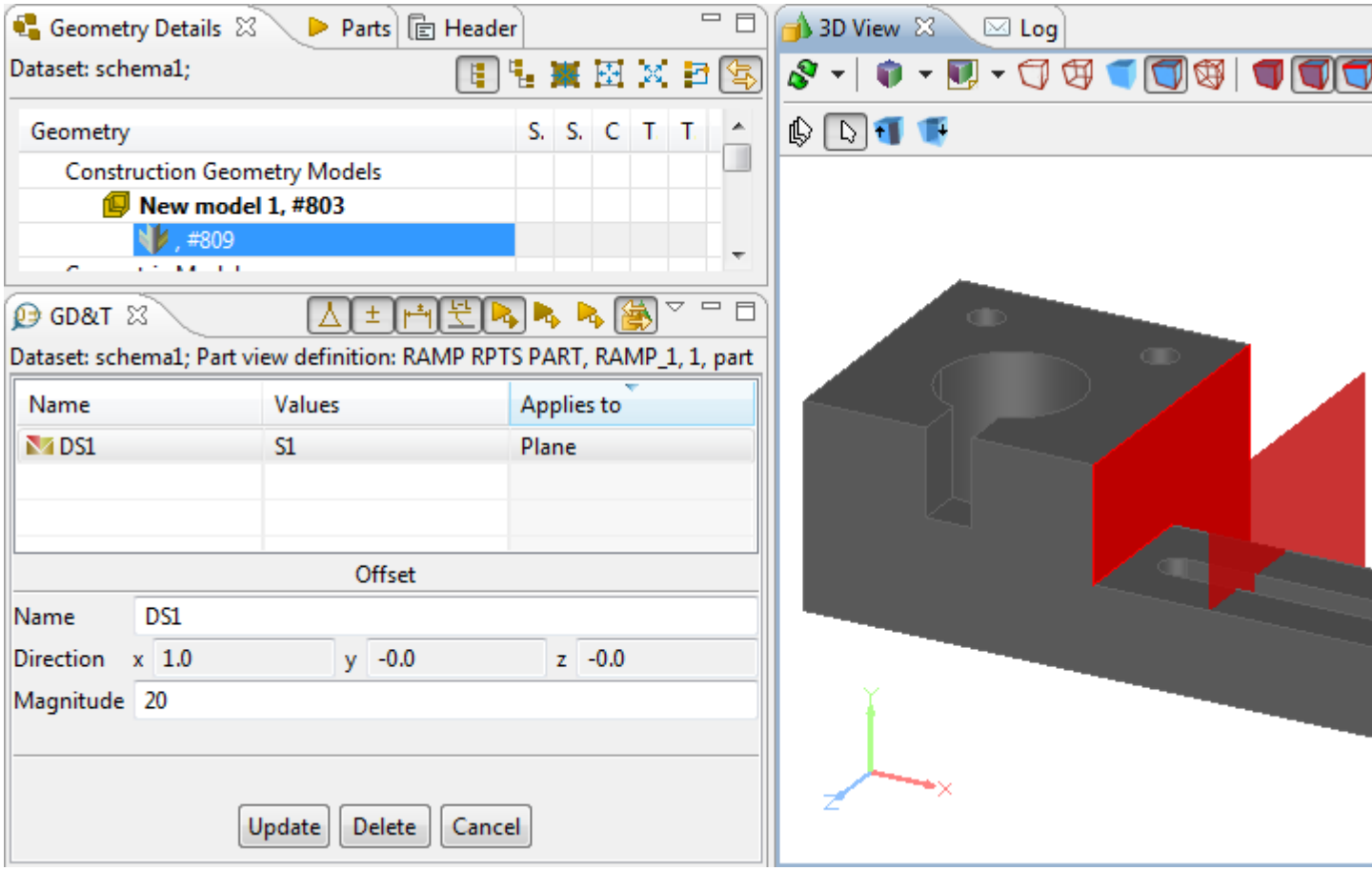

Figure 1. Offset plane (selection in GD&T view) - highlighted in Geometry Details view under the construction geometry model branch, GD&T view (table and wizard) and 3D View, together with referenced plane

To top

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9.2.14.9.9. Perpendicular Line

A *perpendicular line* in IDA-STEP is a line which maintains a 90-degree angle to a particular object and goes through the specified point. The following input objects can be selected for the creation of a perpendicular line:

- **plane** and **point** - to create a *line*, which is perpendicular to the selected plane and goes through the specified point.
- **sphere** and **point** - to create a *line*, which is a sphere center axis through the specified point.
- **cylinder** ant **point** - to create a *line*, which is perpendicular to the center axis of the selected cylinder and goes through the specified point.
- **cone** and **point** - to create a *line*, which is perpendicular to the surface of the selected cone at the specified point.

To create perpendicular line in the **3D View**:

- Select two applicable object (see the list above for available combination).
- Right click and select the **New Constructive Geometry -> Perpendicular line** item in the appeared context sensitive menu.

After the menu item is selected a perpendicular line is instantly created.

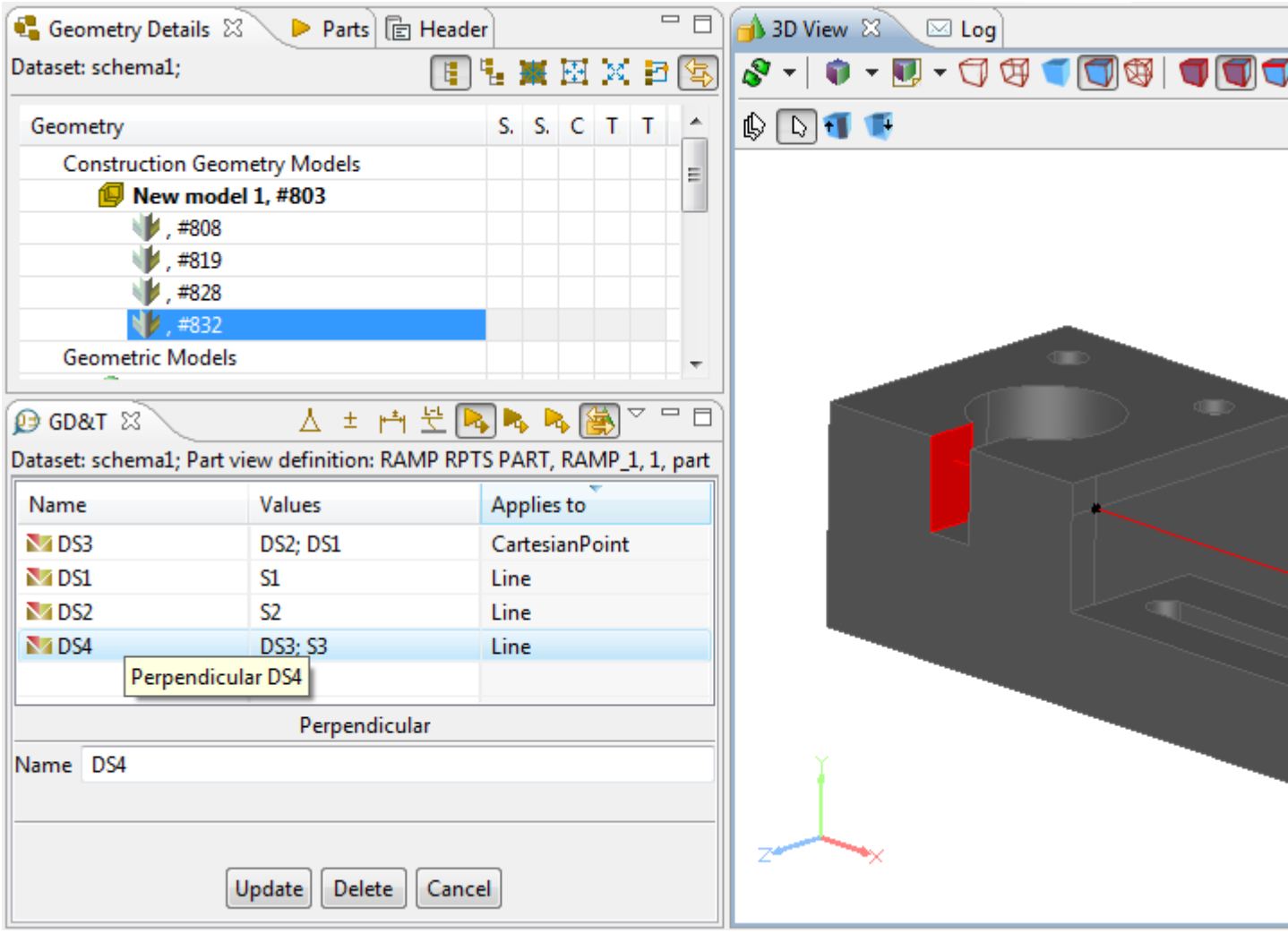

Figure 1. Perpendicular line to selected plane through the specified point (selection in GD&T view) - highlighted in Geometry Details view under the construction geometry model branch, GD&T view (table and wizard) and 3D View, together with referenced plane and point

To top

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9.2.14.9.10. Perpendicular Plane

A perpendicular plane in IDA-STEP is a plane, which maintains a 90-degree angle to a particular object and goes through the specified point/line. The following input objects can be selected for the creation of a perpendicular plane:

- **plane** and **line** - to create a *plane*, which is perpendicular to the selected plane and goes through the specified line.
- **line** and **point** - to create a *plane*, which is perpendicular to the selected line and goes through the specified point.
- **cylinder** and **point** - to create a *plane*, which is perpendicular to the center axis of the selected cylinder and goes through the specified point.

To create perpendicular plane in the **3D View**:

- Select two applicable object (see the list above for available combination).
- Right click and select the **New Constructive Geometry -> Perpendicular Plane** item in the appeared context sensitive menu.

After the menu item is selected a perpendicular plane is instantly created.

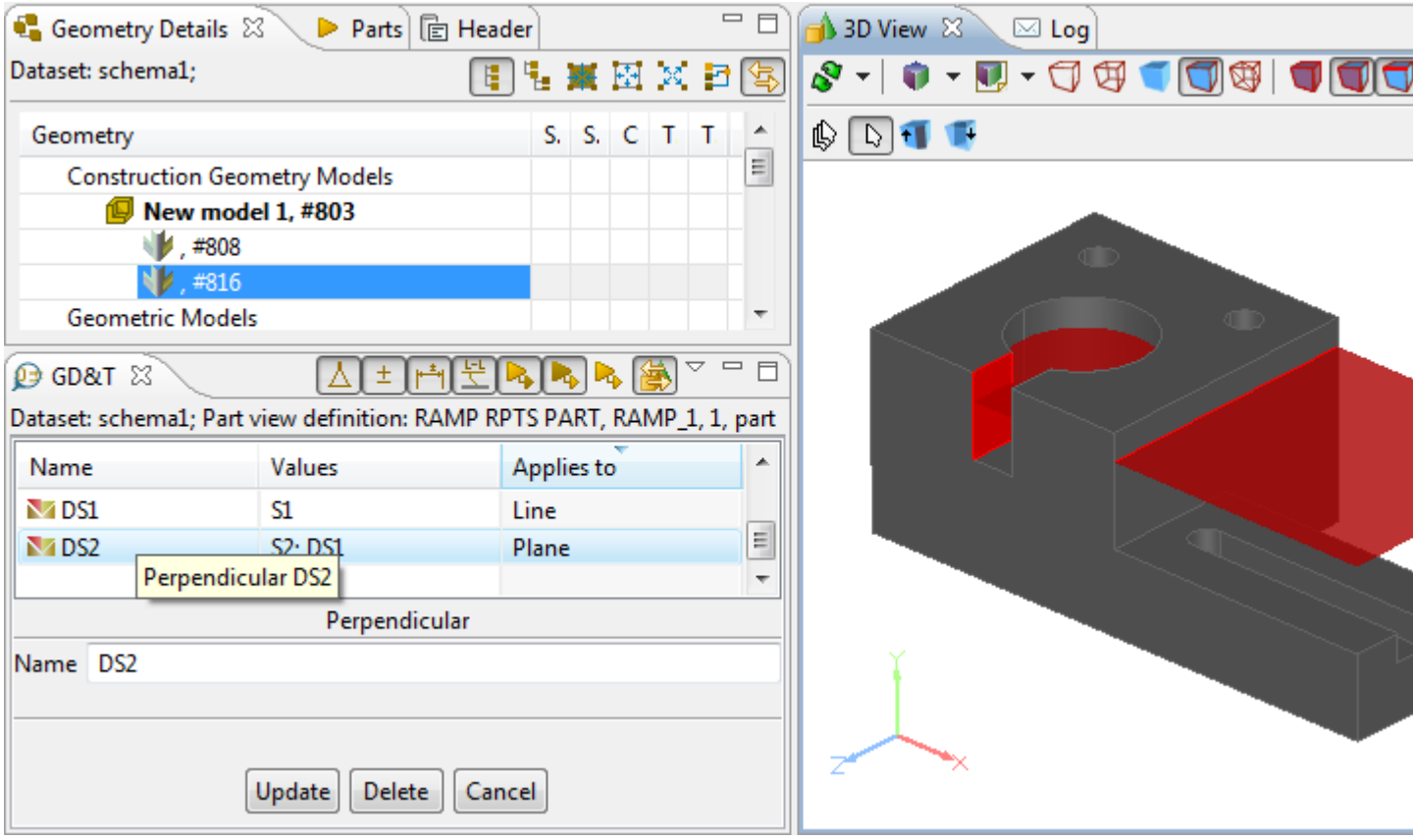

Figure 1. Perpendicular plane to plane surface through line (selection in GD&T view) - highlighted in Geometry Details view under the construction geometry model branch, GD&T view (table and wizard) and 3D View, together with referenced plane and line

To top

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9.2.14.9.11. Extension

An *extension* in IDA-STEP is an extension of a plane.
To create an extension in the **3D View**:

- Select a planar surface on the geometry object.
- Right click and select the **New Constructive Geometry -> Extension** item in the appeared context sensitive menu.

After the menu item is selected an extension of the selected plane is instantly created.

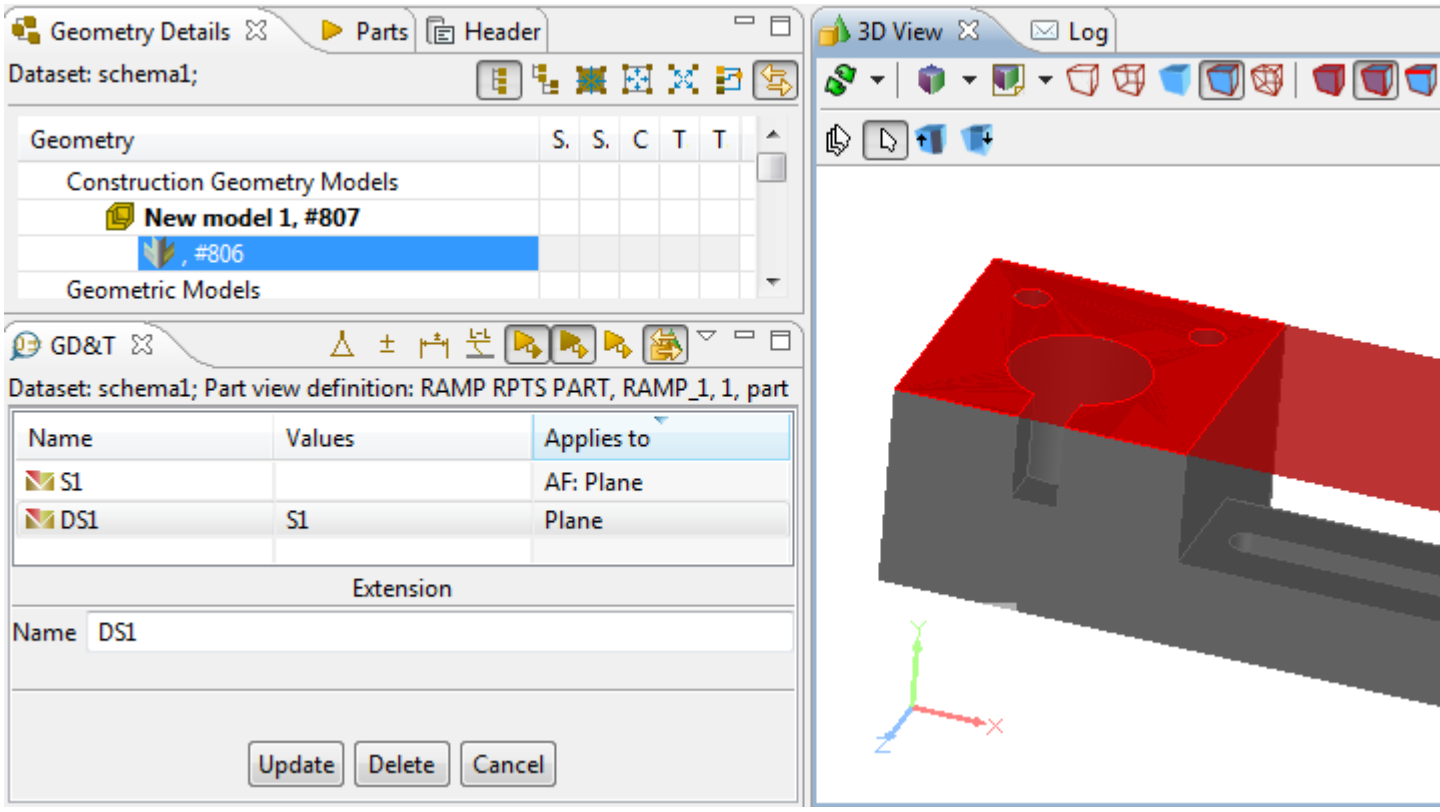

Figure 1. Plane extension (selection in GD&T view) - highlighted in Geometry Details view under the construction geometry model branch, GD&T view (table and wizard) and 3D View, together with referenced plane

See also GD&T -> Create Extension.

To top

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9.2.14.9.12. Tangent Plane

A *tangent plane* in IDA-STEP is a plane, that touches the selected object at the specified point. The following input objects can be selected for the creation of a tangent plane:

- **cylinder** and **point** - to create a *plane* that touches the selected cylinder at the specified point.
- **cone** and **point** - to create a *plane* that touches the selected cone surface at the specified point.
- **sphere** and **point** - to create a *plane* that touches the selected sphere at the specified point.

To create perpendicular plane in the **3D View**:

- Select two applicable object (see the list above for available combination).
- Right click and select the **New Constructive Geometry -> Tangent Plane** item in the appeared context sensitive menu.

After the menu item is selected a tangent plane is instantly created.

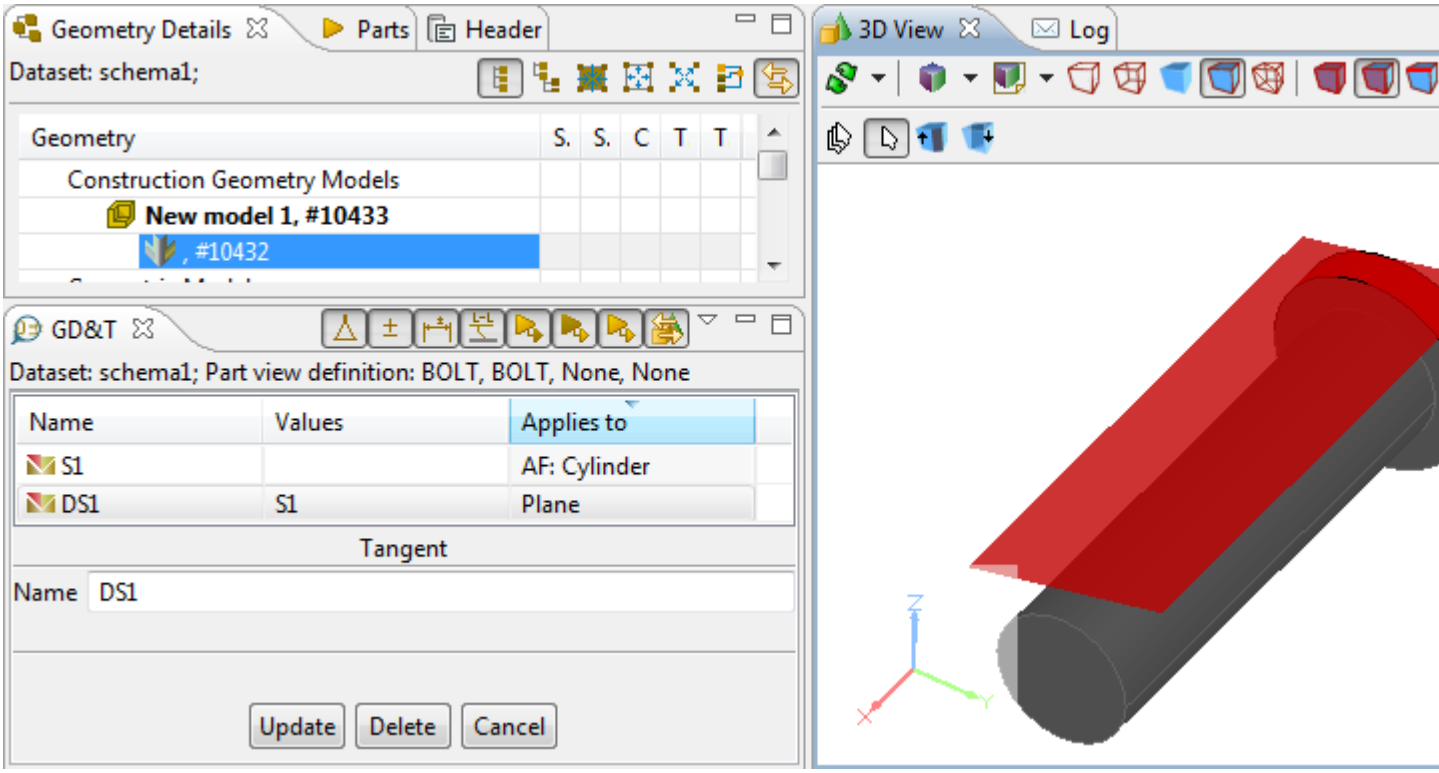

Figure 1. Tangent plane of a cylinder at specified point (selection in GD&T view) - highlighted in Geometry Details view under the construction geometry model branch, GD&T view (table and wizard) and 3D View, together with referenced cylinder and point

See also GD&T -> Create Tangent Plane.

To top

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9.2.14.9.13. Point

A *point* in IDA-STEP is a point for a construction geometry model, created using explicit entered geometric values.
To create a point in the **3D View**:

- Right click on the **3D View** area.
  Or
- Select any point on the 3D surface or edge and right click. **Note.** The **Vertex/Point** or **Point on Edge or Curve Selection Mode** should be switched *On*.
- Select the **Constructive Geometry -> Point** item in the appeared context sensitive menu.
- Further details, how to set parameters for a point, see GD&T -> Constructive Geometry -> New Point. **Note.** In case the point on edge or surface was selected in the 3D View, all parameter values of that point are filled automatically.

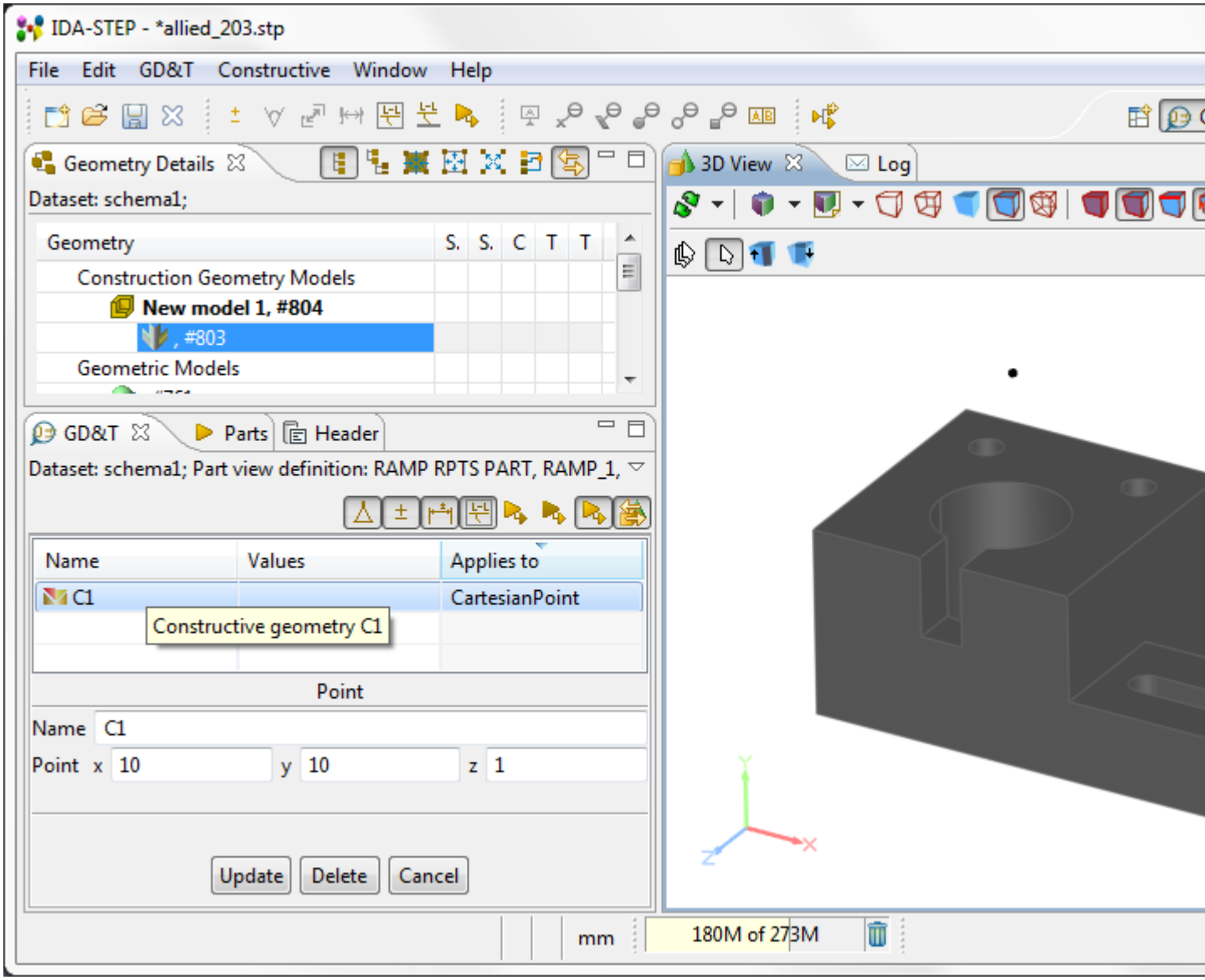

Figure 1. Point with specified coordinates - highlighted in Geometry Details view under the construction geometry model branch, GD&T view (table and wizard) and 3D View

To top

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9.2.14.9.14. Line

A *line* in IDA-STEP is a line for a construction geometry model, created using explicit entered geometric values.
To create a line in the **3D View**:

- Right click on the **3D View** area and select the **New Constructive Geometry -> Line** item in the appeared context sensitive menu.
- Further details, how to set parameters for a line, see GD&T -> Constructive Geometry -> New Line.

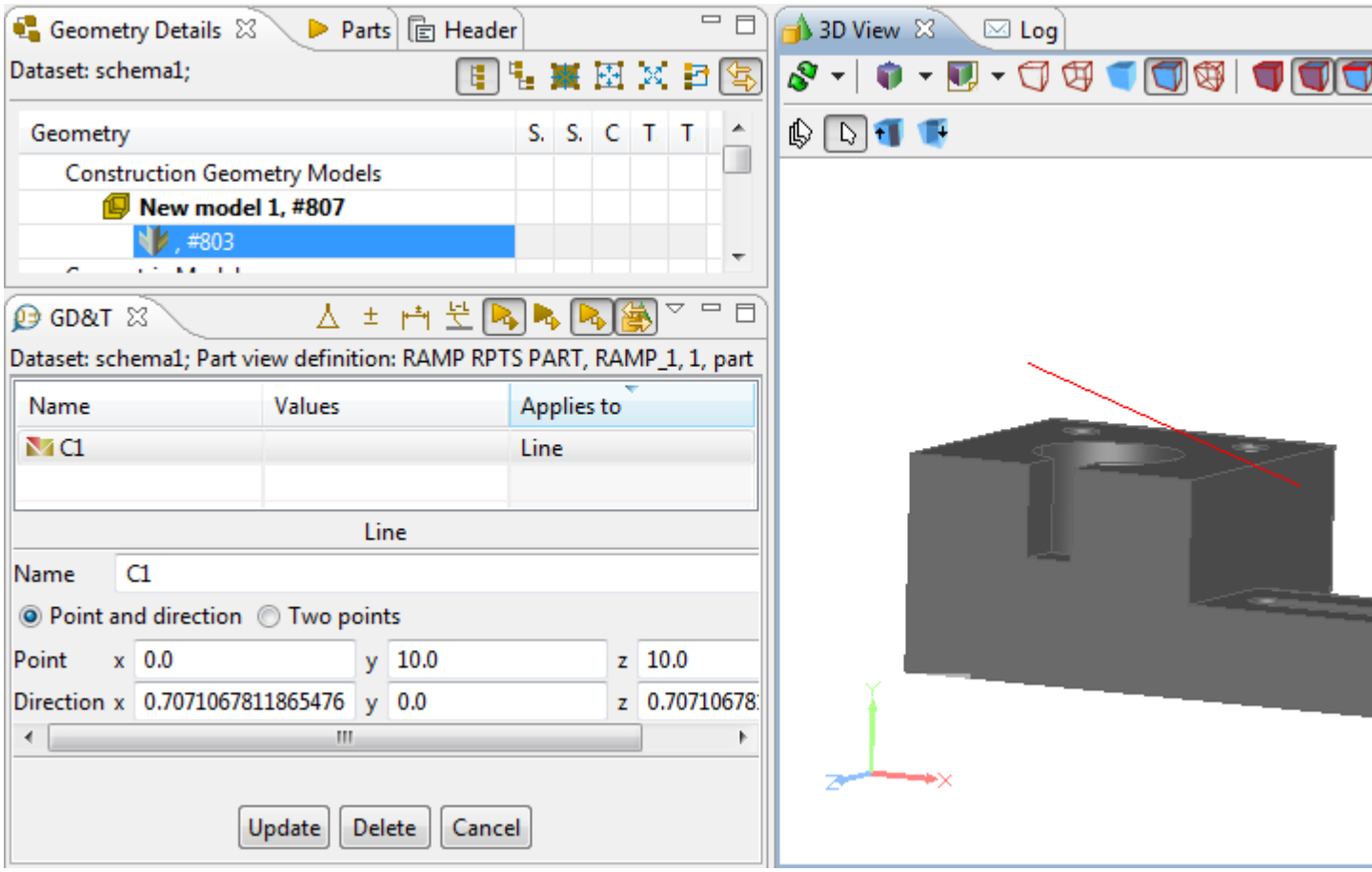

Figure 1. Line with specified point and direction (selection in GD&T view) - highlighted in Geometry Details view under the construction geometry model branch, GD&T view (table and wizard) and 3D View

To top

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9.2.14.9.15. Plane

A *plane* in IDA-STEP is a plane for a construction geometry model, created using explicit entered geometric values.
To create a plane in the **3D View**:

- Right click on the **3D View** area and select the **New Constructive Geometry -> Plane** item in the appeared context sensitive menu.
- Further details, how to set parameters for a plane, see GD&T -> Constructive Geometry -> New Plane.

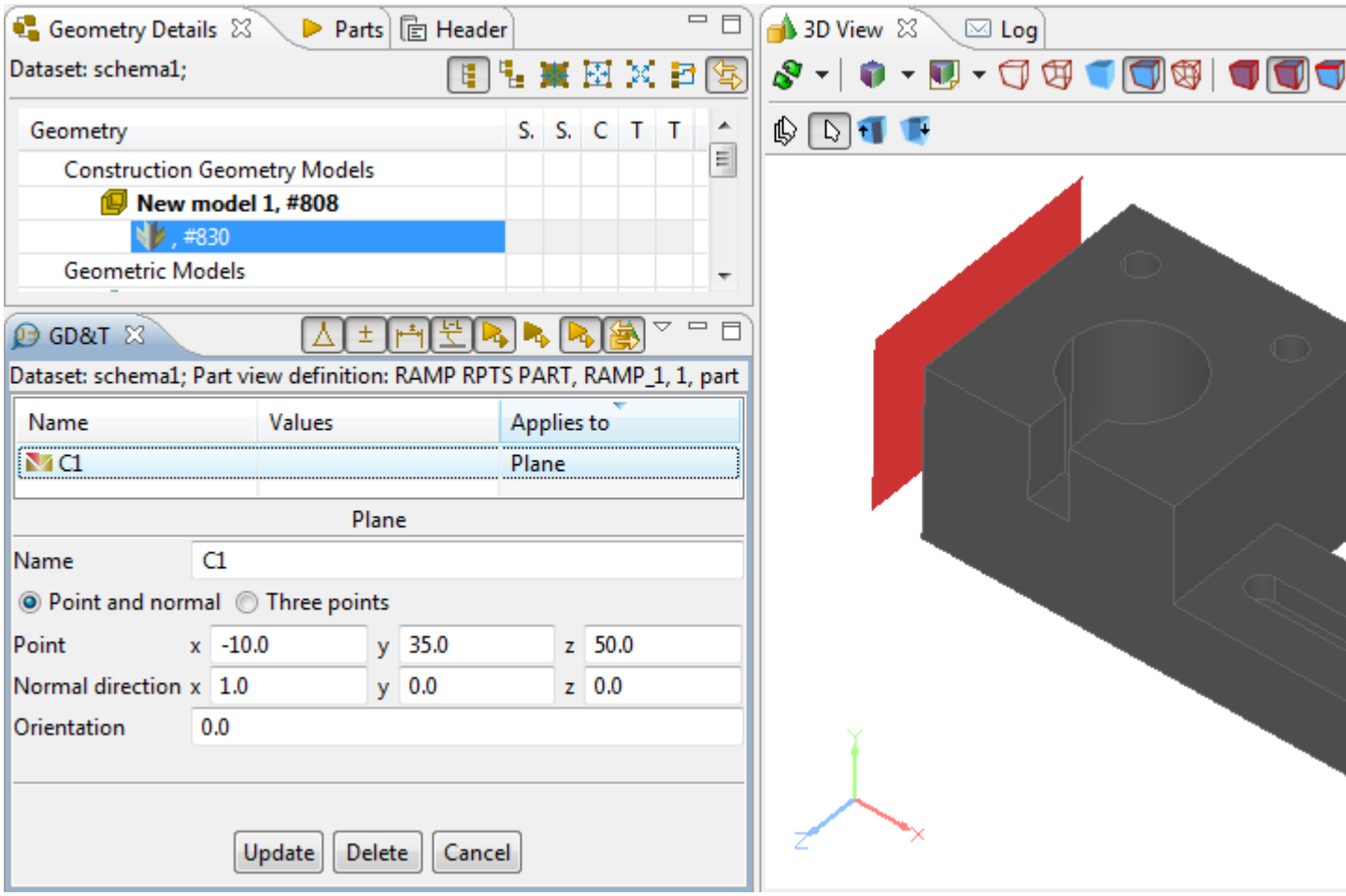

Figure 1. Plane through point and normal (selection in GD&T view) - highlighted in Geometry Details view under the construction geometry model branch, GD&T view (table and wizard) and 3D View

To top

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9.2.14.10. Align Offset

Align Offset is a function to align the offset of annotations on the same annotation plane (see Figure 1):

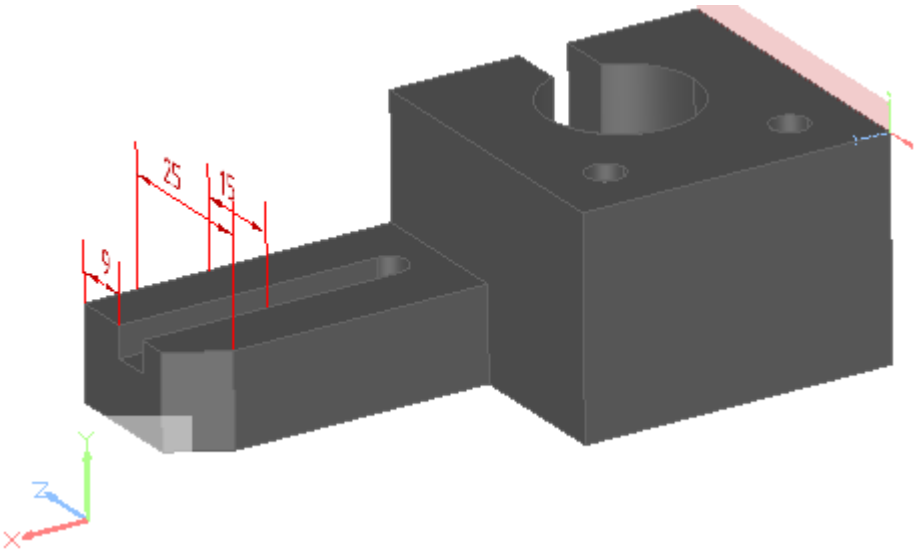
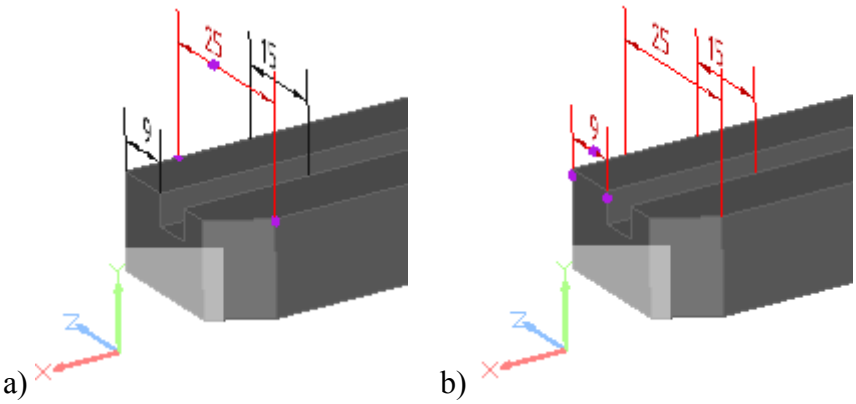


Figure 1. Dimension annotations on the same annotation plane

- Activate the **3D View**.
- Select the master annotation (see Figure 2a). Then select all the other ones that should use the same offset as the *master* one (see Figure 2b).



a) b)

Figure 2. a) Selected master annotation **b)** Selected all annotations to be aligned

- Right click and select **Align Offset** item in the appeared context sensitive menu (see Figure 3).

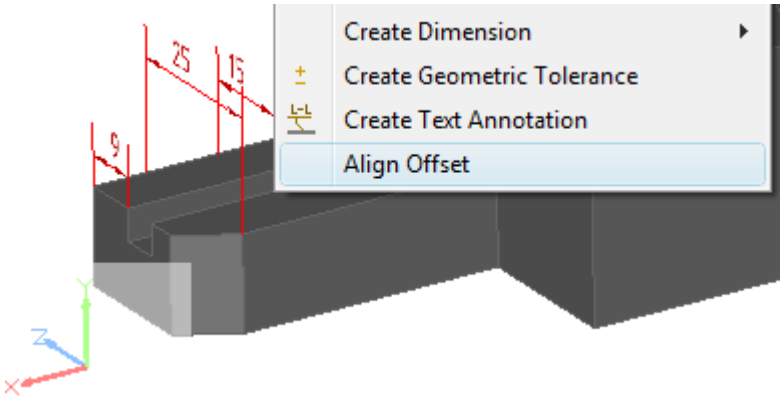


Figure 3. Align Offset item selected in context sensitive menu

The offset of annotations is aligned according to the *master* annotation offset (see Figure 4). The master annotation is highlighted in the **3D View**.

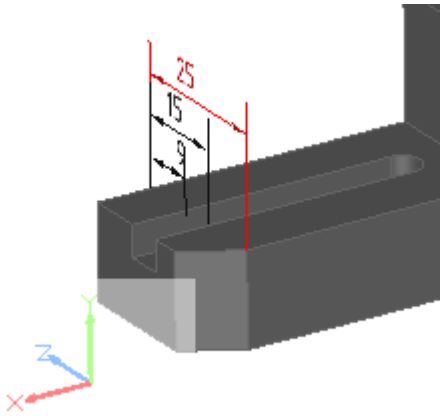


Figure 4. Aligned offset of annotations (master annotation is highlighted)

To top

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9.3.1. Datasets View

The **Datasets** window presents available datasets in the opened repository (see Figure 1). Datasets may have assigned types and a particular icon indicates a type of the dataset, for example:

- Parts,
- / Persons and Organizations,
- Tasks,
- Zonal Breakdowns,
- Part Details,
- Product as Individual,
- Stratum technology and etc.

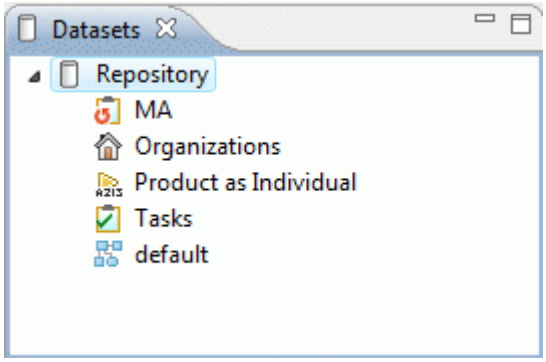

Figure 1. Datasets window

To top

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9.3.2. Creating Dataset

Follow the routine to create a new dataset in the Datasets window:

- Right click on the root node in the window. The context sensitive menu appears (see Figure 1).

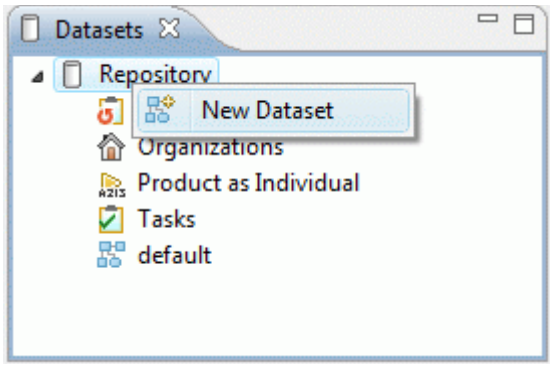

Figure 1. Context sensitive menu, New Dataset menu item

- Select **New dataset** menu item. The **Create New Dataset** wizard window appears (see Figure 2):

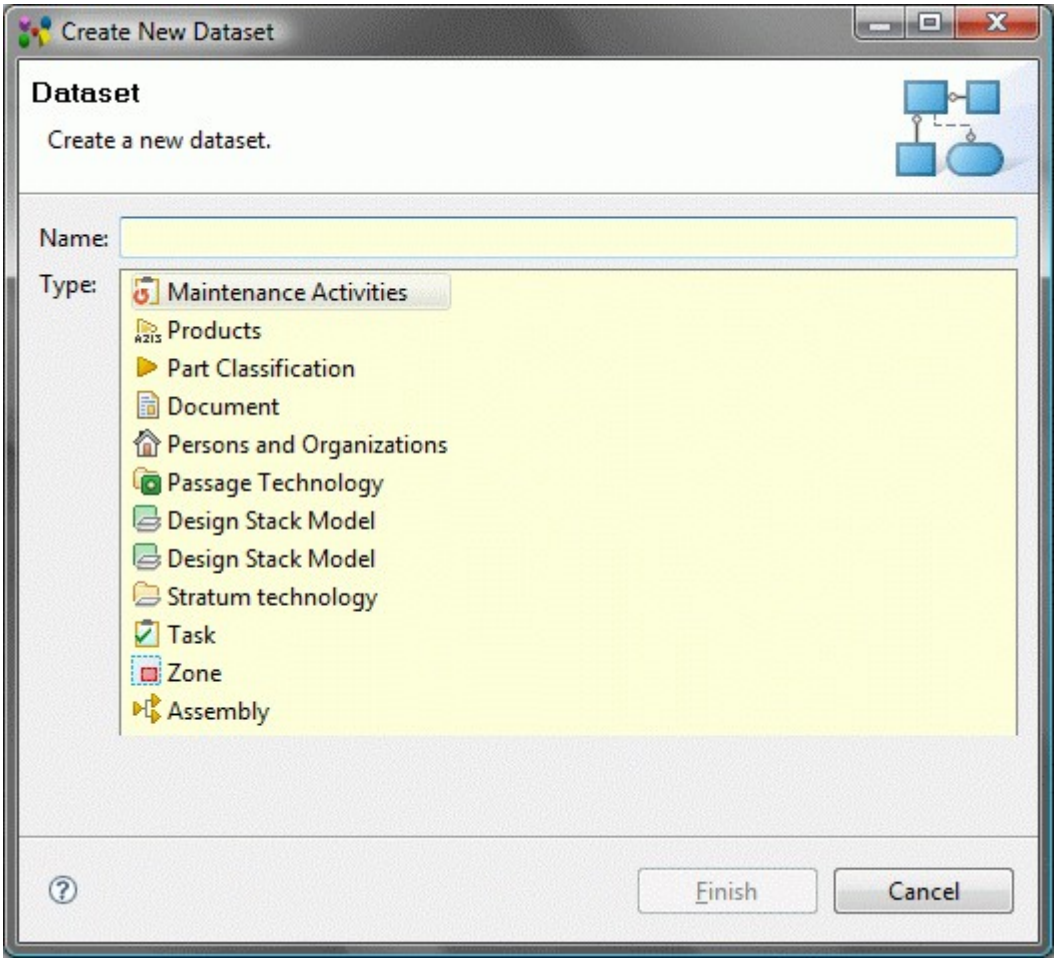

Figure 2. Create New Dataset wizard window

- Enter a name of the dataset to be created into **Name** field.
- Select a type of the dataset in the **Type** layout.
- Click **Finish** button to create a dataset of the selected type.

- After the creation of the dataset is completed, the created dataset appears in the **Datasets** window.

To top

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9.3.3. Renaming Dataset

Follow the routine to rename a dataset in the Datasets window:

- Right click on a dataset you want to rename. The context sensitive menu appears.
- Select **Rename** menu item. The name of the dataset is selected:
  - Enter new or edit the current dataset name.
  - Press **Enter** key.
- The selected dataset is renamed in the **Datasets** window.

To top

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9.3.4. Removing Dataset

Follow the routine to remove an incorrect or redundant dataset from the Datasets window:

- Right click on a dataset you want to remove. The context sensitive menu appears.
- Select **Delete** menu item. The selected dataset disappears from the **Datasets** window.

To top

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9.4.1. Design Stack Models View

The **Design Stack Models** view (available by selecting in the main menu bar **Window -> Show view -> Other** then under **PCB Layer Stack** node) presents a list of design stack models (see Figure 1). The first column presents the row element - design stack model. Other columns contain data of the particular design stack model. The default column perspective contains the following columns:

- **Id** - an identifier of a design stack model.
- **Name** - specifies the word or group of words by which the design stack model (in the role of product) is referred to.
- **Id Owner** - the assigned person or organization owns the design stack model (in the role of product), and has final say over its disposition and any changes to it.
- **Creator** - the design stack model (in the role of product) has been created by the assigned person or organization.
- **Conductive layers** - the number of conductive layers in the design stack model.
- **Total Thickness** - the total thickness of the layers, which contain the current design stack model.

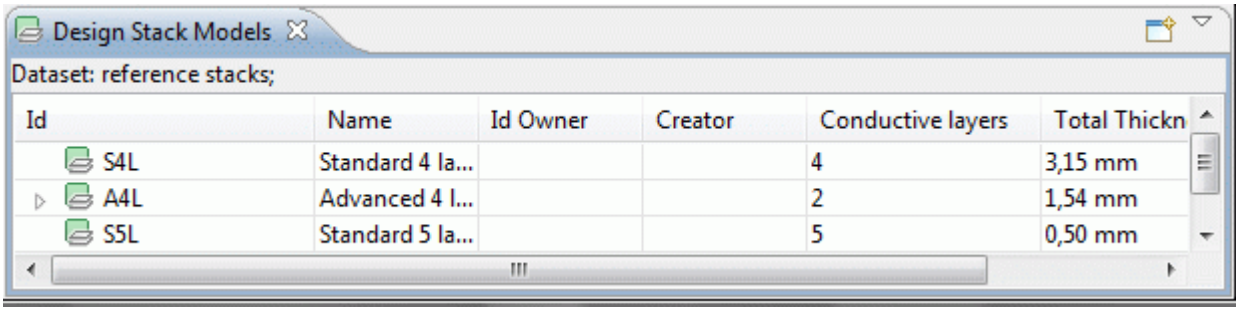

Figure 1. Design Stack Models view (default columns)

To top

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9.4.2. Creating Design Stack Model

Follow the routine to create a new design stack model in the **Design Stack Models** view table (see Figure 1):

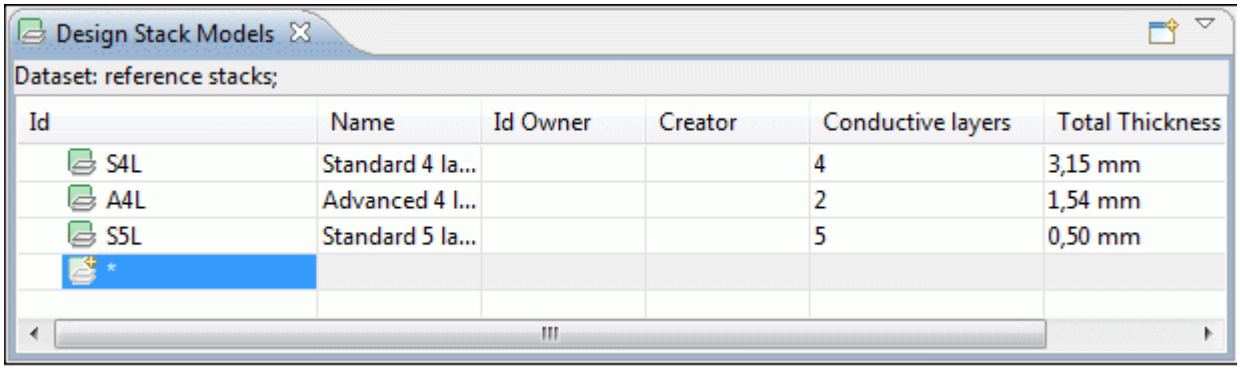

Figure 1. Design Stack Models view (with highlighted autorow)

- Select the autorow in the table (see Figure 1 highlighted cell in **Id** column).
- Click on the selected autorow to enter in to the edit mode.
- Enter an identifier for the new design stack model.
- Press **Enter** key. A new record appears in the **Design Stack Models** view table (see Figure 2).

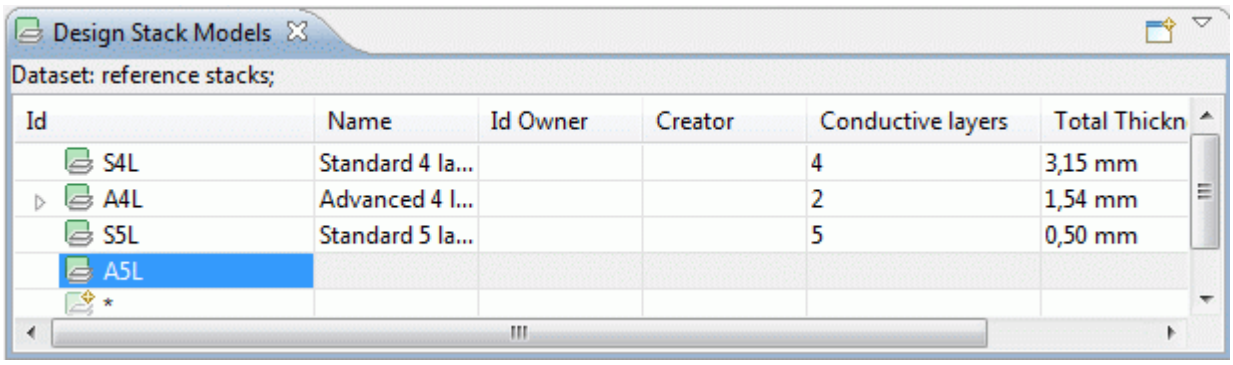

Figure 2. Created design stack model element in the table.

To top

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9.4.3. Editing Design Stack Model Data

Activate the **Design Stack Models** view. Select an applicable design stack model element in the table. You can edit the following:

- Enter the identifier into the **Id** column.
- Enter the text, which defines the design stack model (in the role of product) name into the **Name** column.
- Assign an organization in the **Id Owner** and **Creator** columns by dragging and dropping an applicable organization form the **Persons and Organizations** view.

You can not directly enter the value into the **Conductive layers** field. This attribute is derived and comes form the **Stratums** view. The value depends on the number of conductive layers entered in the **Stratums**

view for the particular design stack model element.
Also you can not edit the **Total Thickness** column value, as it depends on values entered in the **Stratums** view table (thickness of layers of the particular design stack model).

To top

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9.5.1. Documents

A *document* is a logical document that serves as the identifier for a container for some product data.
The **Documents** window presents a tree table with a hierarchy of documents, which are available for the selected dataset (see Figure 1). The default perspective contains the following columns:

- **Id** - an identifier of the document.
- **Name** - specifies the word or group of words by which the document is referred to.
- **Description** - specifies additional information about the document.

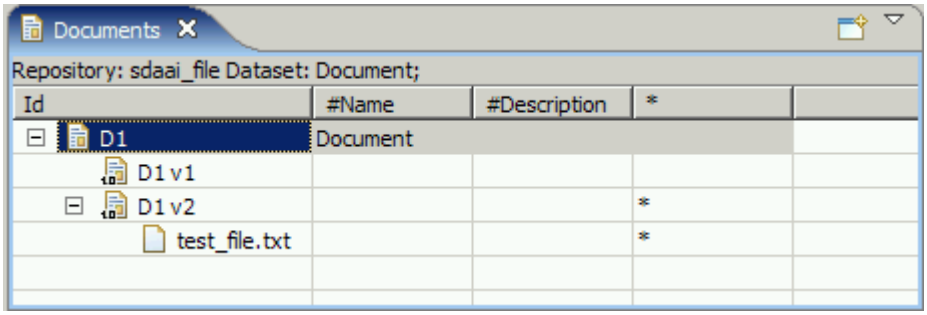

Figure 1. Documents window

To top

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9.5.2. Document Hierarchy

The **Id** column in the **Documents** window presents the hierarchy of documents (see Figure 1):

- **Root node** - documents, files, file references.
- **Second level node** - document versions.
- **Third level node** - associated files and/or file references.

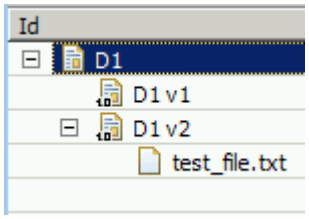

Figure 1. Document hierarchy

To top

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9.5.3. Creating New Document

Follow the routine to create a new document in the document hierarchy:

- Right click on an empty area of the Documents window.
- Select the **New Document** menu item in the appeared context sensitive menu (see Figure 1).

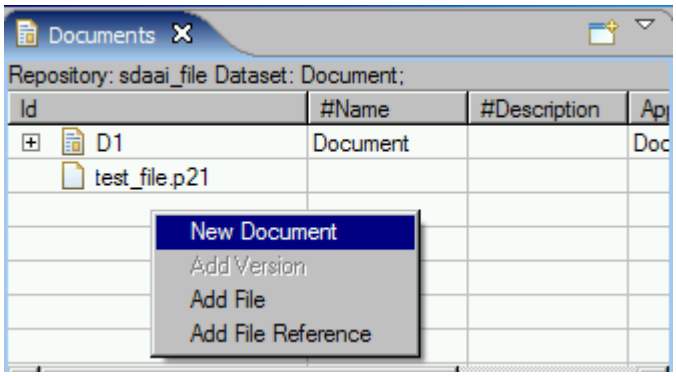

Figure 1. New document menu item

- The new document appears as the root node in the tree with a new version node (see Figure 2).

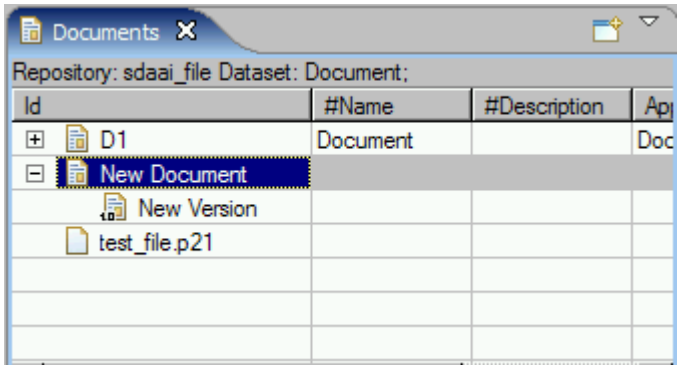

Figure 2. New document with version

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9.5.4. Adding Version

To add a new version of a document:

- Right click on an applicable document node in the tree.
- Select the **Add Version** menu item in the appeared context sensitivie menu (see Figure 1).

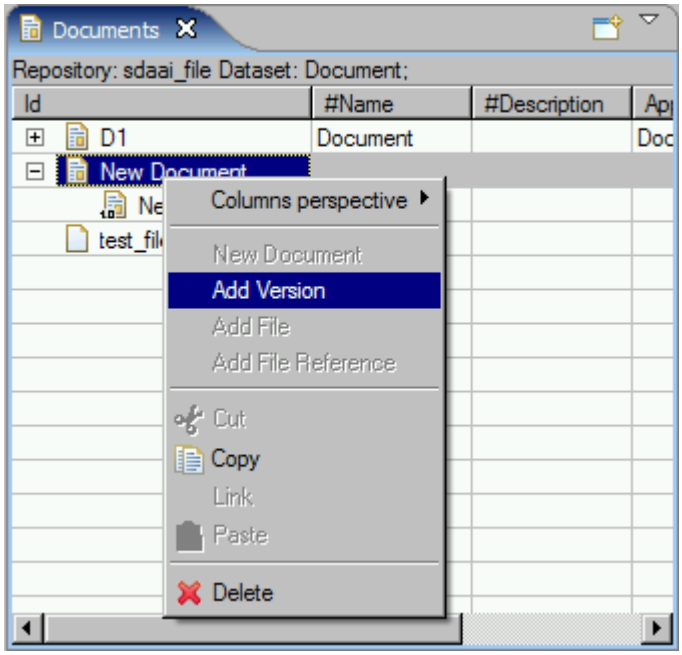

Figure 1. Add Version menu item

- The new version node appears under the selected document (see Figure 2).

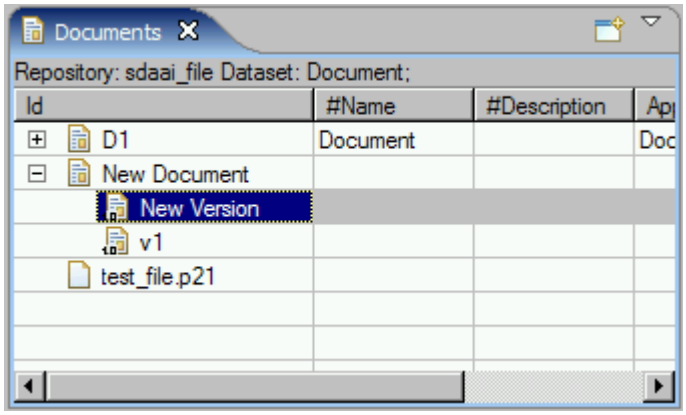

Figure 2. New version node in the tree

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9.5.5. Adding File

There is a possibility to add a file as a root element in the tree or assign a file to a particular document version.
To add a file:

- Right click on the empty area of the **Documents** window or on an applicable document version.
- Select the **Add File** menu item in the appeares context sensitive menu (see Figure 1).

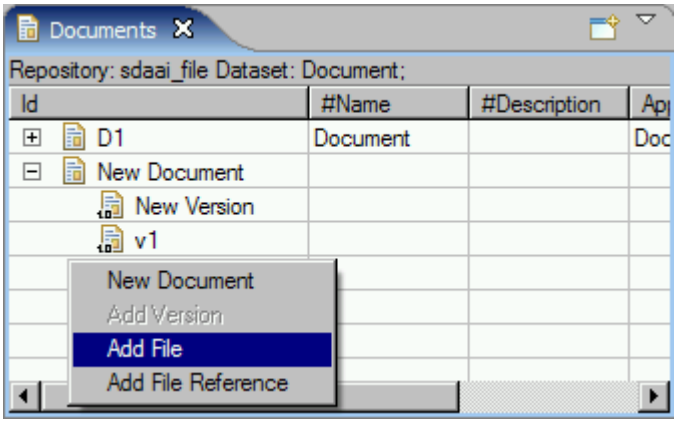

Figure 1. Add File menu item

- Select a file to be added in the folder system by using **Choose file** dialog window (see Figure 2).

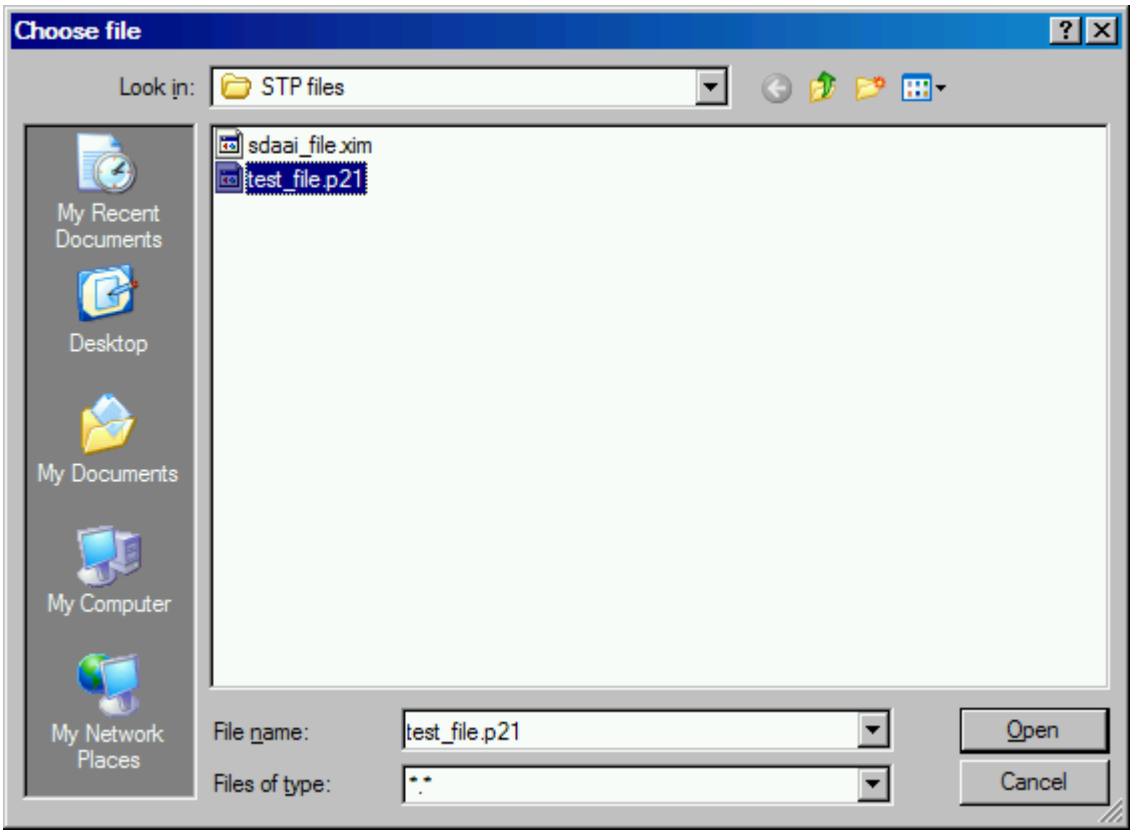

Figure 2. Choose file dialog window to select a file to be added

- The selected file appears in the tree (see Figure 3).

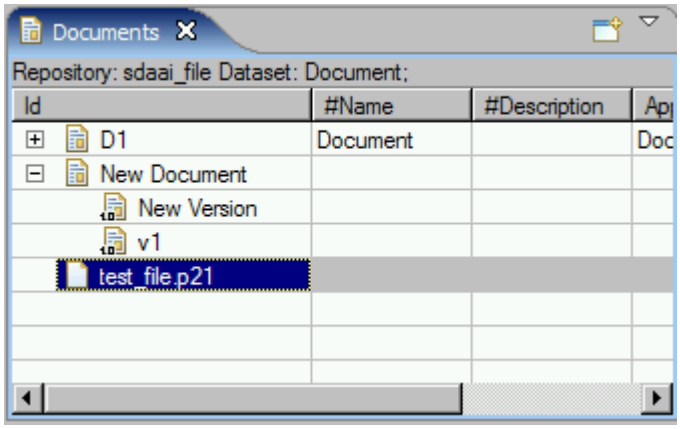

Figure 3. Added file as a root element in the tree

To top

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9.5.6. Adding File Reference

There is a possibility to add a file reference as a root element in the tree or assign file reference to a particular document version.
To add file reference:

- Right click on an empty area of the **Documents** window or on an applicable document version node.
- Select the **Add File Reference** menu item in the appeared context sensitive menu (see Figure 1).

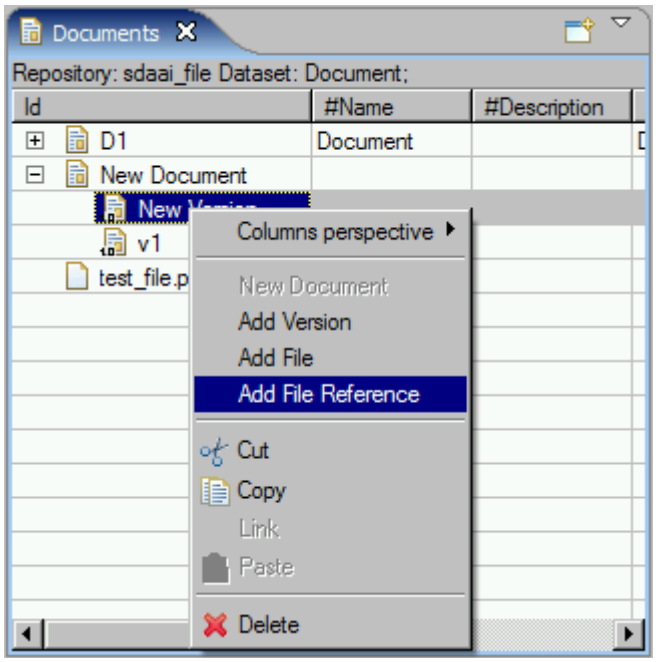

Figure 1. Add File Reference menu item

- Select the file to be added as a reference in the folder system by using **Choose file** dialog window (see Figure 2).

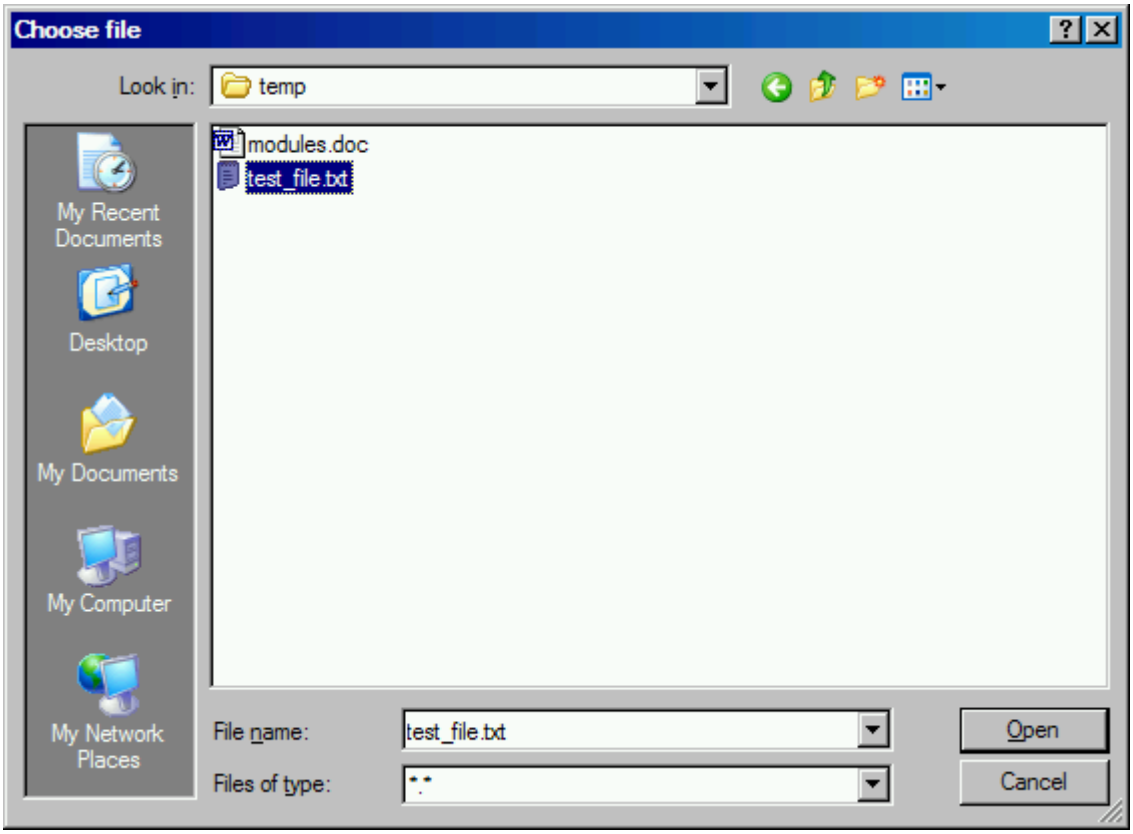

Figure 2. Choose file dialog window

- The associated file refernce appears in the tree (see Figure 3).

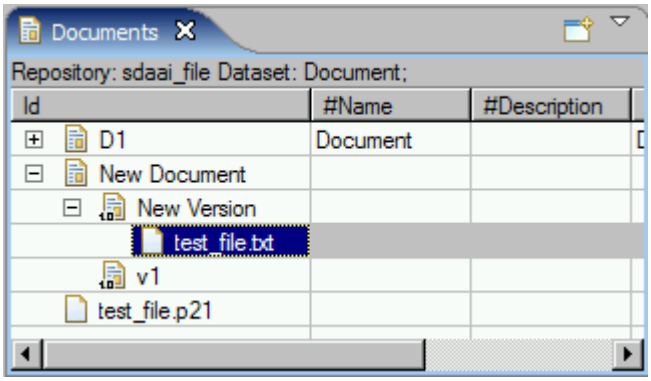

Figure 3. Added file refernece under the document version node

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9.6.1. FMECA View

The *Failure mode* specifies the manner by which a failure is observed; it generally describes the way the failure occurs.
The *Failure effect* specifies immediate consequences of a failure on operation, function or functionality, or status of some item.
The *Criticality analysis* is used to chart the probability of failure modes against the severity of their consequences.
The *Failure cause* are defects in design, process, quality, or part application, which are the underlying cause of the failure or which initiate a process which leads to failure.

The **FMECA (Failure Mode, Effects, and Criticality Analysis)** view presents a table which

provides the maintenance of FMECA data (see Figure 1). The first column presents a hierarchy:

- **Top node** - a product part, for which the failure mode object may be associated as a child.
- **Second level node** - failure mode, which can contain child failure modes and failure causes.
- **Third and lower level nodes** - associated failure modes and/or failure causes.

Note. The failure cause node can not have any child nodes.

Other table columns contain particular FMECA data for each failure mode and failure cause. The default column perspective contain the following columns:

- **Local Effect** - specifies the failure effect as it applies to the item under analysis.
- **Next Higher Assembly Effect** - specifies the failure effect as it applies at the next higher indenture level.
- **End Effect** - specifies the failure effect at the highest indenture level or total system.
- **Severity** - specifies the worst possible consequence of a failure classified by the degree of injury, property damage, system damage and mission loss that could occur.
- **Failure Mode Class** - specifies the class of the failure mode. Predefined values are 'Human error', 'Disassembly', 'External effect', 'Deterioration' and 'Lubrication failure'.
- **Failure Cause Class** - specifies the classification items of the failure cause. Predefined values are: 'Design failure', 'Weakness failure', 'Manufacturing failure', 'Aging failure', 'Misuse failure' and 'Mishandling failure'.
- **Preventive control (design provision)** - specifies the actions available or that can be taken to negate or reduce the effect of a failure on a system.
- **Probability** - specifies a probability of the failure cause.
- **Detection Rating** - specifies the detection of the condition which might lead to the failure.
- **Detection Method** - specifies the method by which a failure can be discovered by the system operator under normal system operation or by a maintenance crew carrying out a specific diagnostic action. The predefined values are: 'Alarm', 'Built in test', 'Non destructive examination', 'Vibration analysis', 'Particle analysis', 'Chemical analysis', 'Spectographic analysis', 'Visual', 'Audible' and 'Olfcatory'.
- **RPN** (Risk Priority Number) - provides an alternate evaluation approach to Criticality Analysis. The risk priority number provides a qualitative numerical estimate of design risk. RPN is defined as the product of three independently assessed factors: Probability, Detection Rating and Severity.

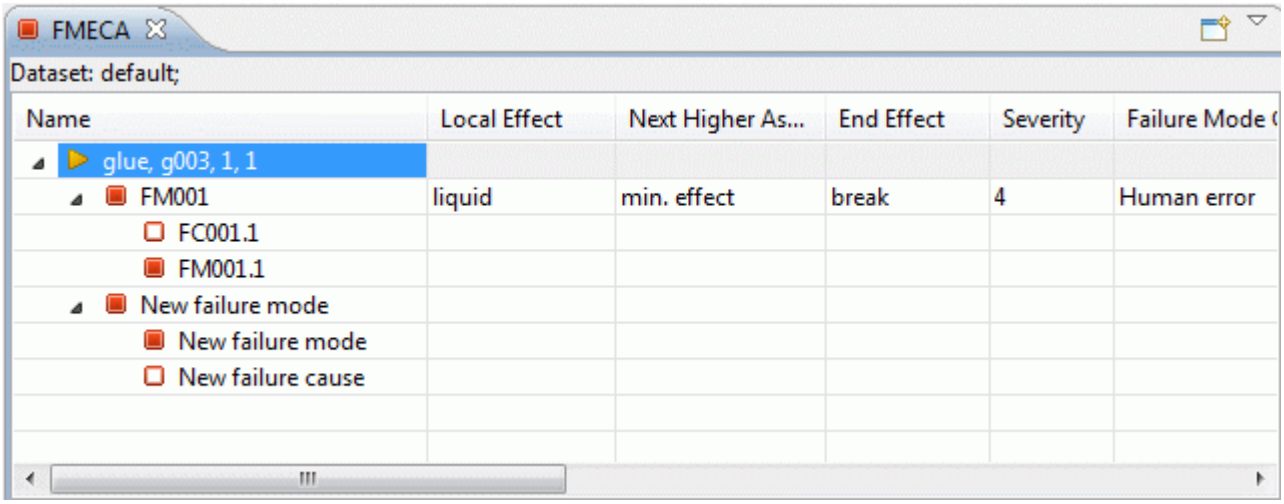

Figure 1. FMECA view

To top

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9.6.2. Creating Failure Mode

Follow the routine to create a new failure mode object in the **FMECA** view:

- Right click on a product part node to create a top level failure mode.

Or

- Right click on an applicable failure mode node in the tree to create a child failure mode object.
- Select the **New failure mode** menu item in the appeared context sensitive menu (see Figure 1).

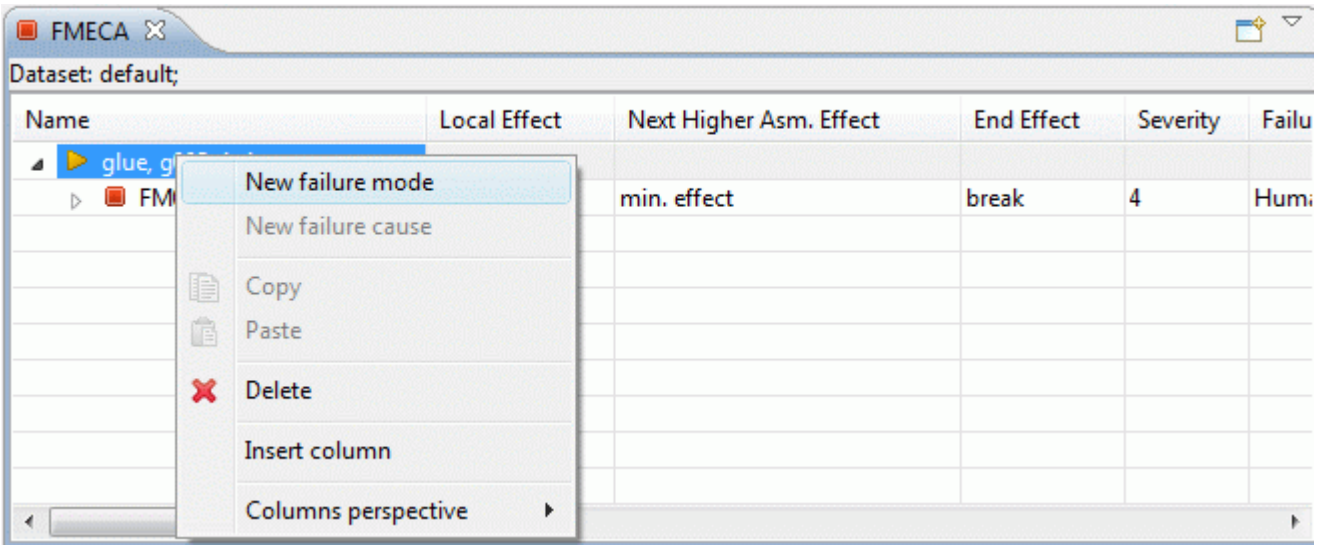

Figure 1. Context sensitive menu with selected New failure mode menu item

The created failure mode object, named 'New failure mode', appears under the selected node (see Figure 2).

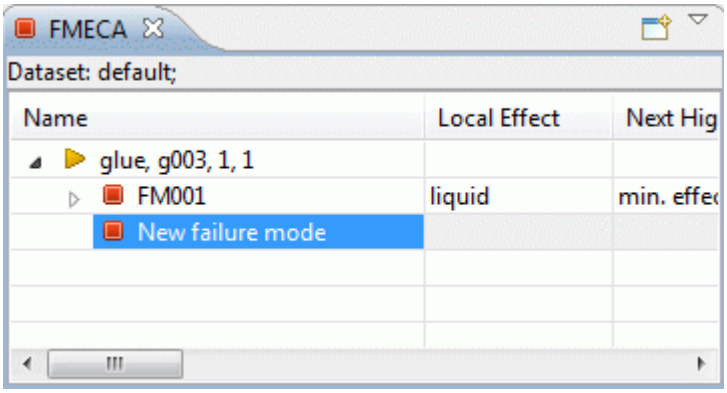

Figure 2. Created failure mode (highlighted)

To top

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9.6.3. Creating Failure Cause

Follow the routine to create a new failure cause object in the **FMECA** view:

- Right click on an applicable failure mode node.
- Select the **New failure cause** menu item in the appeared context sensitive menu (see Figure 1).

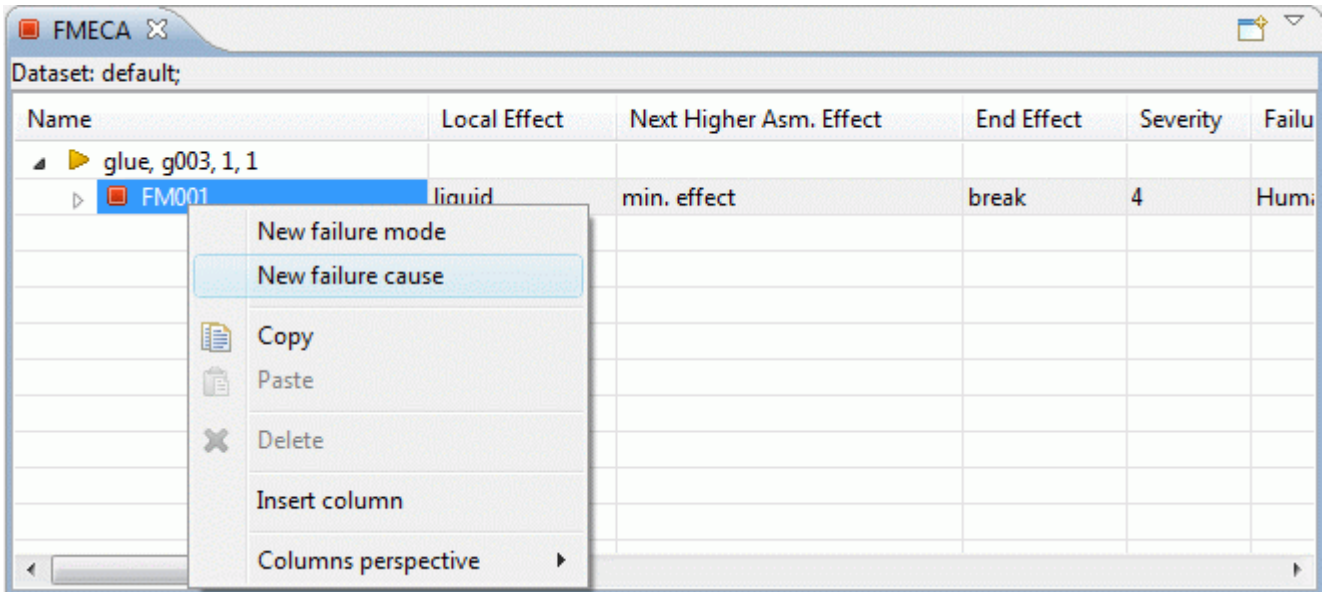

Figure 1. Context sensitive menu with selected New failure cause menu item

- The created failure cause object, named 'New failure cause', appears under the selected failure mode node (see Figure 2).

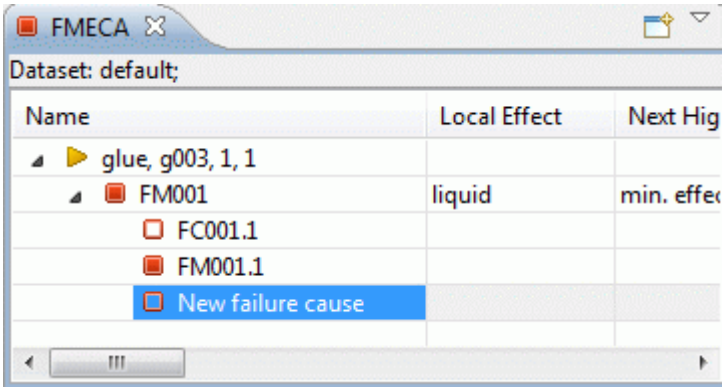

Figure 2. Created failure cause (highlighted)

To top

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9.6.4. Editing Failure Mode Data

Activate the **FMECA** view. Select an applicable failure mode element in the tree table. You can edit the following:

- Enter a new text or edit the present text in the **Local effect** column text field.
- Enter a new text or edit the present text in the **Next higher assembly Effect** column text field.

- Enter a new text or edit the present text in the **End effect** column text field.
- Select an applicable number value from the combo box list in the **Severity** column.
- Select an applicable classification name value from the combo box list in the **Failure Mode Class** column.
- Select an applicable number value from the combo box list in the **Detection Rating** column.

To top

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9.6.5. Editing Failure Cause Data

Activate the **FMECA** view. Select an applicable failure cause element in the tree table. You can edit the following:

- Enter a new text or edit the present text in the **Local effect** column text field.
- Enter a new text or edit the present text in the **Next higher assembly Effect** column text field.
- Enter a new text or edit the present text in the **End effect** column text field.
- Select an applicable classification name from the combo box list in the **Failure Cause Class** column.
- Enter a new text or edit the present text in the **Preventive Control (design provision)** column text field.
- Select an applicable number value from the combo box list in the **Probability** column.
- Select an applicable method name from the combo box list in the **Detection Method** column.

To top

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9.6.6. Removing

Follow the routine to remove a redundant or incorrect failure mode and/or failure cause from the **FMECA** view tree table:

- Right click on an applicable element in the **Name** column.
- Select **Delete** menu item in the appeared context sensitive menu.

The selected element is removed from the tree table.

Note. If the selected element contains the child elements, they will be removed together with the selected element.

To top

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9.7.1. GD&T View

GD&T stands for geometric dimensioning and tolerancing. The principles of this methodology are defined in:

- ISO 110 Geometrical Product Specifications (GPS) - Geometrical tolerancing - Tolerances of form, orientation, location and run-out;
- ISO 5459 Technical drawings - Geometrical tolerancing - Datums and datum-systems for geometrical tolerances;
- ASME Y14.5 Geometric Dimensioning and Tolerancing;
- and others.

ISO and ASME standards are similar, but with specific differences a designer has to take care of. The **GD&T** view is supporting the functionality defined in the referenced standards on the level of semantic information that is computer sensible. In addition presentation information attached to the 3D model is generated according to either:

- ISO 16792 Technical product documentation - Digital product definition data practices, or
- ASME Y14.41 Technical product documentation - Digital product definition data practices.

The initial release of this GD&T functionality does not support all details of referenced standards.

The content of the **GD&T** view (see Figure 1), like the **3D View**, depends on the selection in the **Parts** view (selected part version). If no part version is selected in the **Parts** view, the **GD&T** view shows an empty page with a message "There is no selected part".

Table of GD&T elements

The big table in the middle lists the GD&T elements that contribute to the complete GD&T specification. The following element types are available:

- shape aspect - these are wrapper objects for geometric elements and by default they are not visible.
- datum feature - a surface component used to established a datum.
- datum target - an element to establish the datum reference plane for manufacturing and inspection operations.
- geometric tolerance - presents acceptable variations (maximum and minimum limits) of size, form, profile, orientation, location, and runout of a feature.
- dimension with or without tolerances.

The content of the lower part of the view depends on selection in the table. In case an element is selected in the table, the detailed information of this element is presented in a kind of a dialogue and a user is enabled to update data. If some item is selected in the context sensitive menu, the particular dialogue appears on the screen to create a new element.

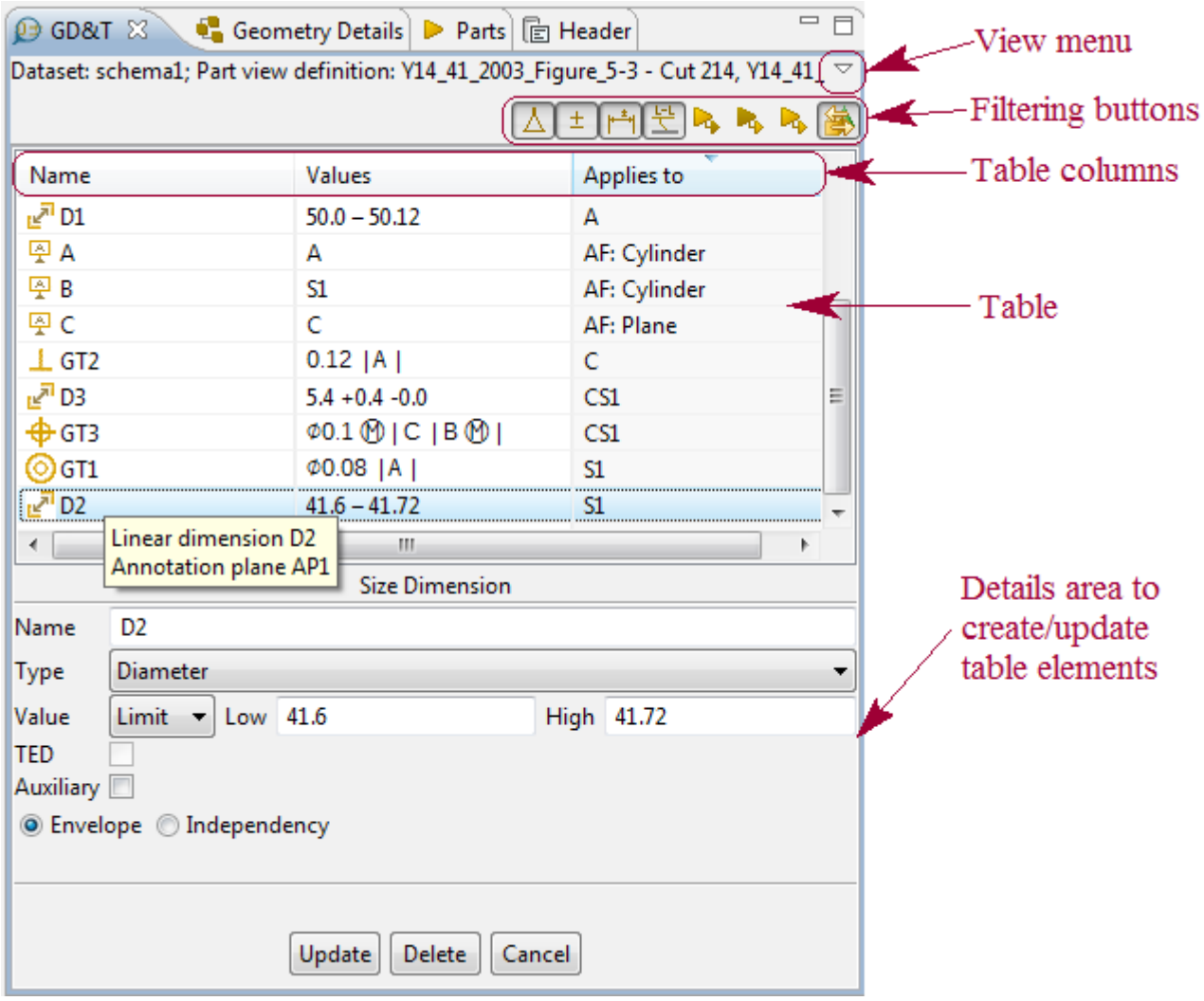

Figure 1. GD&T view table and wizard in details area to update selected entry

Table filters

The upper part of the **GD&T** view contains a line of filtering buttons that control the kind of elements to be displayed in the table.
The following filtering is available for the table elements:

- **Datums** - show datums, datum features, datum targets and other datum elements;
- **Tolerances** - show tolerances;
- **Dimensions** - show dimensions;
- **Annotation planes** - show annotation planes;
- **Derived features** - show shape aspects, which are derived (for example: centre axis, apex, offset and etc.);
- **Integral features** - show shape aspects, which are integral.
  Note. Some composite shape aspects are displayed under this filter (namely those that are composed only of integral shape aspects).
- **Other features** - show all shape aspects, that are not derived and not integral;
- **Follow 3D** - switch this button *On* to follow actions, made in **3D View** (selection in **3D View** is represented in **GD&T** view).

View Menu

The **GD&T** view menu in the right upper corner of the view contains the menu items to change units, dimensioning standard and settings.

Units

All GD&T specifications have to be done in the same unit. The unit combo box identifies whether the unit is **mm** or **inch**.
By default units for length dimensions and tolerances are derived from GD&T preferences. All displayed lengths are of the choose unit. Units can be changed directly in the **GD&T** view by selecting an applicable value from the combo box, which is situated in the upper part of the **GD&T** view. The selected unit (currently active unit) is saved together with all changes after the file is closed.

GD&T Standard

ISO TC10/213 and ASME have different rules on meaning of GD&T and how to apply it. This combo box specifies to which of these GD&T standard families the design applies (**ASME** or **ISO**).

GD&T Settings

The default settings for GD&T are defined in GD&T preferences (see Preferences -> GD&T). The styling related settings are stored together with each annotation. The other settings together with the scaling value are stored with the *Part*. They can be changed via the **GD&T** view menu:

- Select the **Settings...** item in the view menu. The **GD&T Settings** dialogue window is prompted on the screen (see Figure 2):
  - **Annotation size scale** in the **GD&T** view is a transformation that enlarges (increases) or shrinks (diminishes) annotation objects (text height, curve width, terminator symbol size and etc.) by a scale factor that is the same in all directions. See Preferences -> GD&T -> Annotation Styles for default values of annotation objects.
    To scale annotation objects enter an annotation size scale value into the **Annotation size scale** text field.
  - **Number of decimal places for GD&T** - specify the precision for values of GD&T elements (dimensions and geometric tolerances) by entering an applicable value into the number field.
  - **Show trailing zeroes for GD&T** - select this option to show trailing zeros for values of GD&T elements in the **GD&T** view table (not in dialogues).
  - **Number of decimal places for geometry** - specify the precision for values of construction geometry elements, by entering an applicable value into the number field.
- Press **OK** button to confirm your selection.

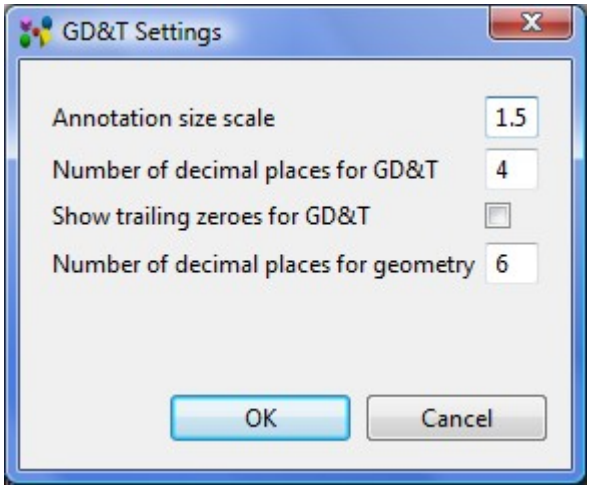

Figure 2. GD&T Settings wizard

Note. To apply new settings for a particular element, select the element in the table and press **Update** button in the **GD&T** view details area. The same result can be achieved by selecting annotation in the **3D View** a moving it to any direction.

Table columns

The columns of this table present the following information:

- **Name** - name GD&T element (flyby text displays the type of the GD&T element);
- **Values** - the content of this column depends on the entry type. There can be present tolerance information, dimension value, datum and etc.
- **Applies to** - shape aspect to which the GD&T element applies.

Sorting

The sorting of table elements can be performed by clicking on an applicable column title.

Context sensitive menu

In case that one or several GD&T elements are selected in the main table in **GD&T** the following context menu entries may show up (depending on other constraints):

- **Delete** - to remove the selected GD&T element from the table.
- **New Composite Shape Aspect** - to create a new composite shape aspect entry. **Note.** This menu item is enabled if more than one shape element is selected in the table.
- **New Constructive Geometry**- to create a new constructive geometry entry for the selected shape aspect.
- **New Datum Feature** - to create a new datum feature for the selected shape aspect.
- **New Dimension** - to create a new dimension for the selected shape aspect.
- **New Geometric Tolerance** - to create a new geometric tolerance entry.
- **New Text Annotation** - to create a 3D annotation with a free text (one or several lines).

If some of the menu entries are no available for the selected element(s), the particular entry is disabled .

Create new GD&T element

To create a new GD&T element:

- Select one or several GD&T elements in the main table. **Note.** To create a simple construction geometry element (line, plane) there is no need to select an element in the table.
- Right click and select a menu item to invoke particular dialogue.
- Enter values into the dialogue fields, which appears in the lower part of the **GD&T** view, and press **Create** button.

Note. IDA-STEP presents possibility to create annotations not only for a single peace part, but also for assemblies.

Update GD&T element

In case some information for a GD&T element is incorrect, select the element from the table and enter

values into the appeared dialogue in the lower part of the **GD&T** view. After all required fields are filled, press **Update** button.

Remove GD&T element

There is a possibility to remove an incorrect or redundant GD&T element from the table list. **Note.** The removal is available only for elements, which have no associations assigned.

- Right click on a GD&T element.
- Select **Delete** menu item in the appeared context sensitive menu.
  Note. The **Delete** menu is disabled for element, which can not be removed - i.e. is used by other elements. The removal is possible only after all users of the selected GD&T element are deleted.

Or

- Select an element in the **GD&T** view table. The wizard to update the element properties appears in the details area.
- Press **Delete** button to remove the element.

Precision for values in GD&T

The precision of a value describes the number of digits that are used to express that value. To set an applicable precision for GD&T values, enter number of **Decimal places** in the GD&T preferences page (see Preferences -> GDT).

To top

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9.7.2. Shape Aspects

A *shape aspect* is an element of the shape of a product. In case some shape aspects exit in **GD&T** view, the user is enabled to create/update an integral shape aspect, a derived shape aspect (construction geometry elements) and other shape aspects (that are built from appropriate shape aspects).
The following data are presented for a shape aspect in the **GD&T** view table:

- **Name** - a name of the shape aspect. **Note.** The fly by text displays a type of a shape aspect (shape aspect, constructive geometry, centre axis and etc.)
- **Values** - names of shape aspects, the selected shape aspect is associated to (in case of composite or derived shape aspects);
- **Applies to** - a product shape of which this entry is an aspect.

In case a shape aspect (integral, composite, derived or other shape aspect) entry is selected in the table of the **GD&T** view, the **Shape aspect** dialogue appears under the table (see Figure 1). Ensure that a particular filter button is turned *On* in the **GD&T** view: **Integral features**, **Derived features** and/or **Other features**.
Follow the routine to edit shape aspect data:

- Edit or enter a new name for a shape aspect.
- Press **Update** button to confirm changes.

Note. Use the **Cancel** button to close the dialogue. Use the **Delete** button to remove the element. If the selected shape aspect is in use by other GD&T elements, the **Delete** button is disabled.

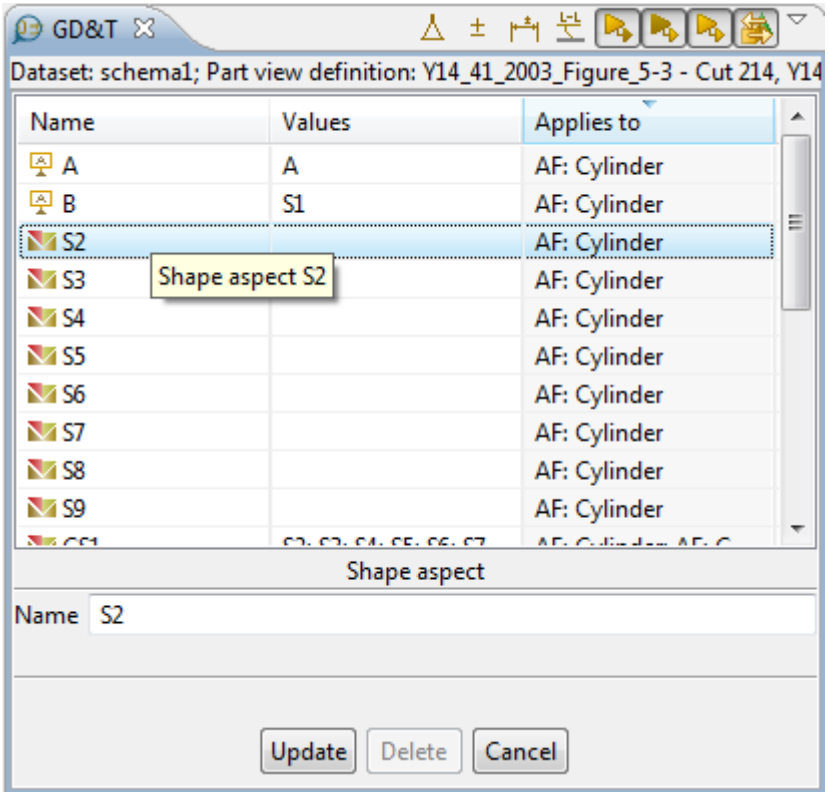

Figure 1. Selected shape aspect in table and Shape aspect dialogue under table

To top

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9.7.2.1. Composite Shape Aspect

A *Composite Shape Aspect* in IDA-STEP is a shape aspect, that associates aspects of the product shape for a specific purpose.

New composite shape aspect

Follow the routine to create a composite shape aspect in the **GD&T** view table:

- Select two or more shape aspect entries in the table. The message string 'Multiple objects selected' appears under the table.
- Right click and select **New Composite Shape Aspect** menu item in the appeared context sensitive menu (see Figure 1).

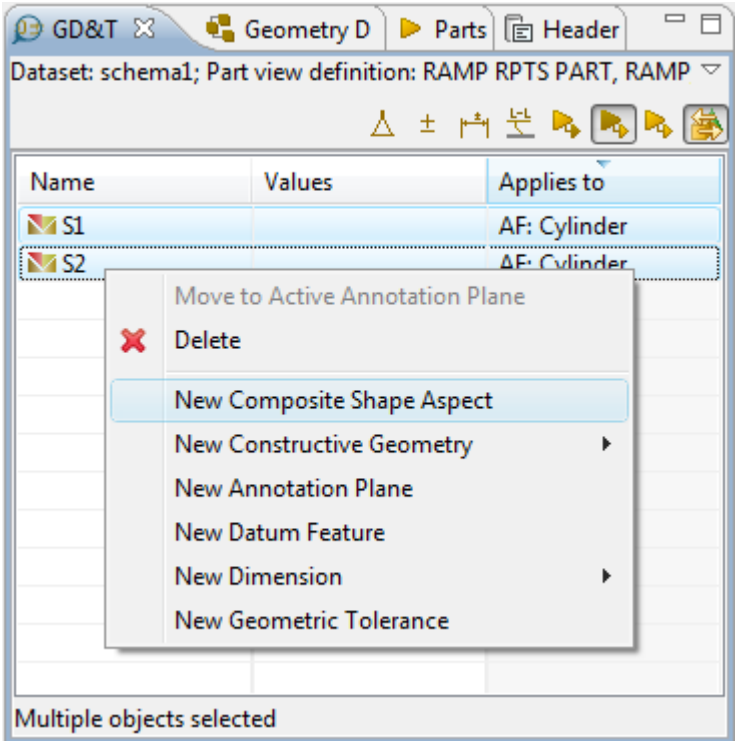

Figure 1. Selected two shape aspects and New Composite Shape Aspect menu item

A new composite shape aspect appears in the table (see Figure 2). The name for this composite shape aspect is set automatically (i.e. 'CS1') and displayed in the **Name** column. The names of shape aspects, which were selected for the creation of the composite shape aspect are presented in the **Values** column. The selection of the composite shape aspect is presented in the **3D View** by highlighting the particular geometry object.

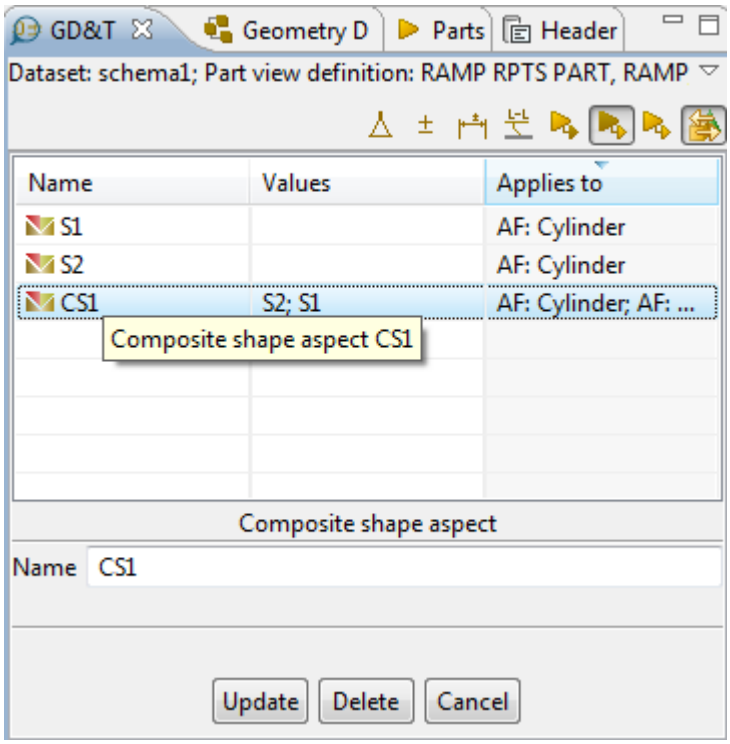

Figure 2. Created composite shape aspect (entry in table) and Shape aspect dialogue (details area under table)

To Top

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9.7.2.2. Constructive Geometry

In this topic you will find a description how to create explicit construction geometry elements (point, line, plane) and derived construction geometry elements for selected shape elements in the **GD&T** view.
Note. To view the created explicit construction geometry element, ensure that the **Other features** filter button is turned *On*, in case of derived construction geometry element, the **Derived features** filtering button should be turned *On*.
When creating a new point/line/plane or a derived shape element in the **GD&T** view, a new constructive geometry model element is created in the **Geometry Details** view, under the active construction geometry model (see Figure 1).

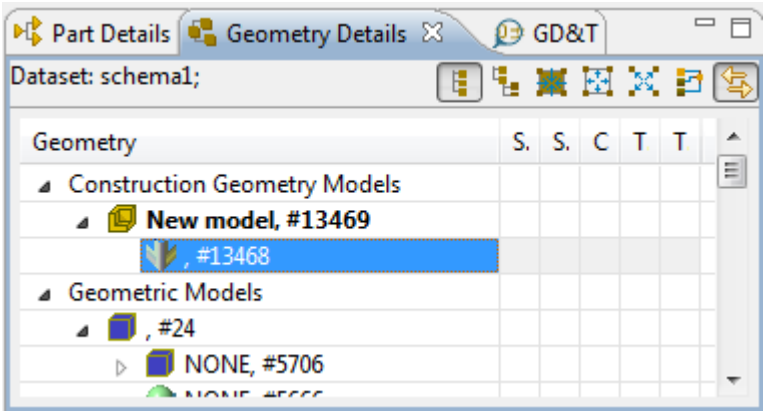

Figure 1. Geometry details view with construction geometry model element

New point

Follow the routine to create a *point* using explicit entered geometric values for a construction geometry model:

- Right click on the **GD&T** view table area.
- Select the **New Constructive Geometry -> Point** item in the appeared context sensitive menu.
- Specify parameters for the point in the **Point** wizard, which appears under the table (see Figure 2):

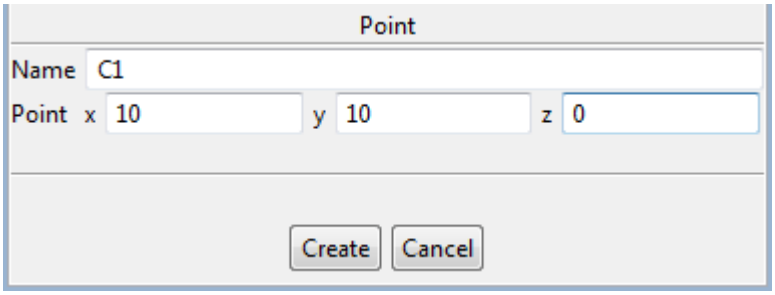

Figure 2. Point wizard with specified parameters for construction geometry element - point

- The name for a point (C with a number) is automatically added to the **Name** field. This default name can be edited by entering a new text into the **Name** field.
- Enter **x**, **y** and **z** values into the **Point** text fields to specify coordinates of the point to be created. **Note.** In case some point on edge or surface was selected in the 3D View, the **x**, **y** and **z** values are filled automatically.
- Press the **Create** button to confirm choices and create a point.

The created point appears in the table of the **GD&T** view and under the particular construction geometry model in the **Geometry Details** view. In case the point element is selected in the table, the **Point** wizard appears under the table (see Figure 3). The point parameters can be updated by entering new values and pressing the **Update** button. The point can be removed by pressing **Delete** button. In case the

point has an association with some other elements, the **Delete** button is disabled.

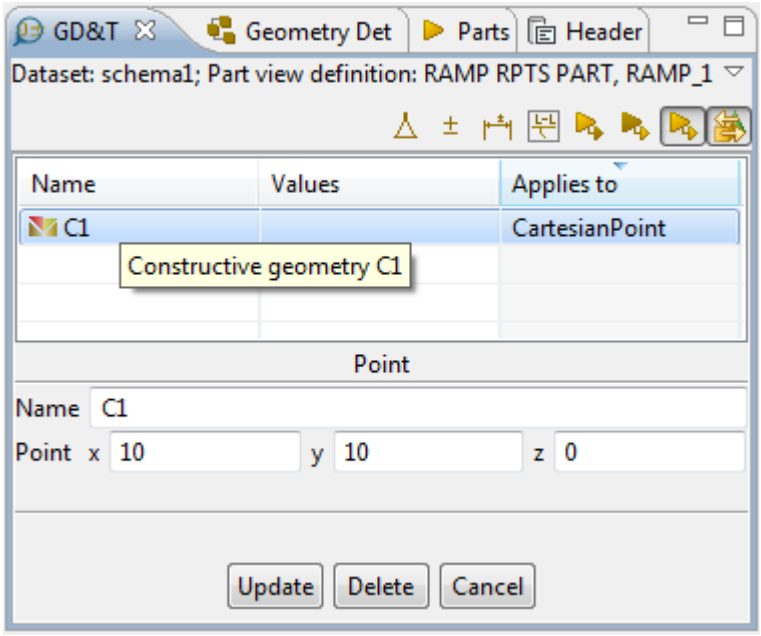

Figure 3. Created point ('C1') in table, Point details in wizard under table

New line

Follow the routine to create a *line* using explicit entered geometric values for a construction geometry model:

- Right click on the **GD&T** view table area.
- Select the **New Constructive Geometry -> Line** item in the appeared context sensitive menu.
- Specify parameters for the line in the **Line** wizard, which appears under the table (see Figure 4):

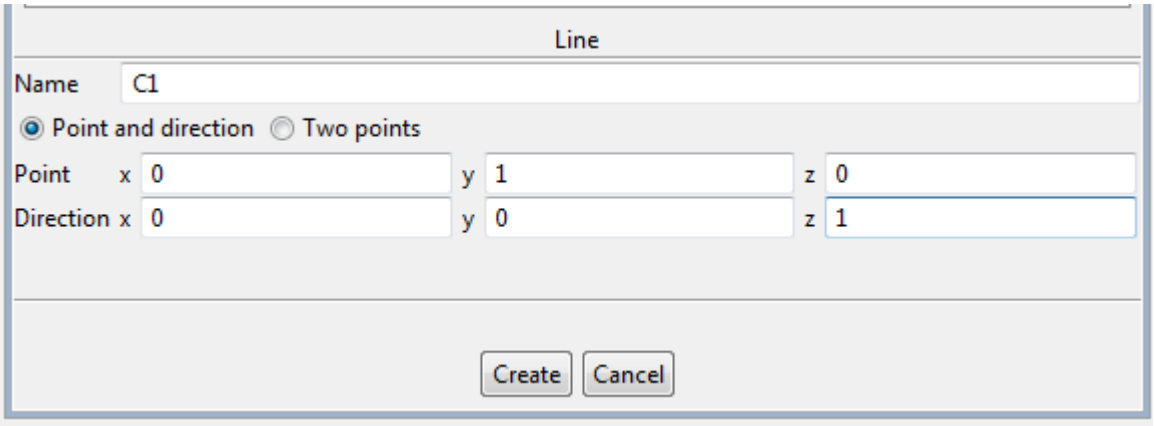

Figure 4. Line wizard with specified parameters for construction geometry element - line

- The name for a line (C with a number) is automatically added to the **Name** field. This default name can be edited by entering a new text into the **Name** field.
- Specify, how the line is defined by selecting particular radio button:
  - Select **Point and direction** to define line by specifying one point and direction:
    - Enter **x**, **y** and **z** values into the **Point** text fields.
    - Enter **x**, **y** and **z** values into the **Direction** text fields.

  Or

  - Select **Two points** to define the line by specifying two points:
    - Enter **x**, **y** and **z** values for the **First** point.

        - Enter **x**, **y** and **z** values for the **Second** point.
    - Press the **Create** button to confirm choices and create a line.

The created line appears in the table of the **GD&T** view and under the particular construction geometry model in the **Geometry Details** view. In case the line element is selected in the table, the **Line** wizard appears under the table. The line parameters can be updated by entering new values and pressing the **Update** button. The line can be removed by pressing **Delete** button. In case the line has an association with some other elements, the **Delete** button is disabled.

New plane

Follow the routine to create a *plane* using explicit entered geometric values for a construction geometry model:

- Right click on the **GD&T** view table area.
- Select the **New Construction Geometry -> Plane** item in the appeared context sensitive menu.
- Specify parameters for the plane in the **Plane** wizard, which appears under the table (see Figure 5):

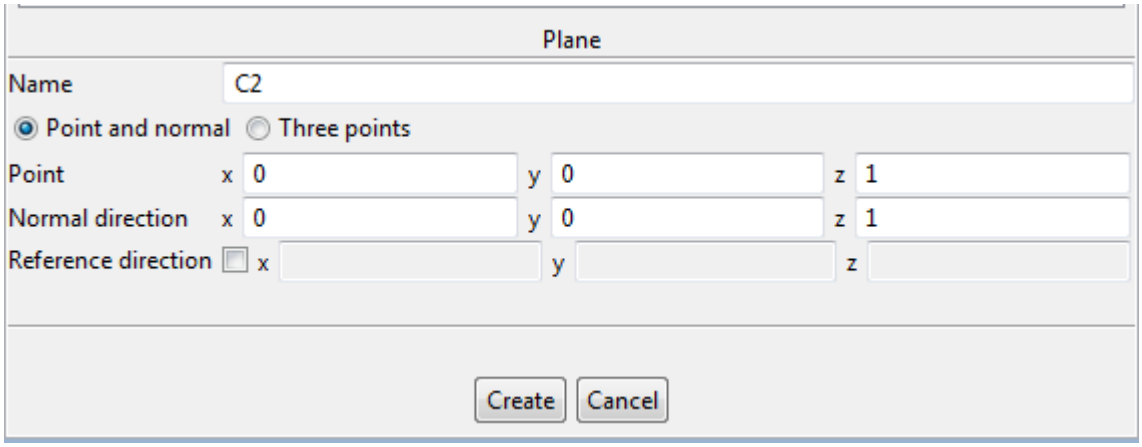

Figure 5. Plane wizard

- The name for a plane (C with a number) is automatically added to the **Name** field. This default name can be edited by entering a new text into the **Name** field.
- Specify, how the plane is defined by selecting particular radio button:
    - Select **Point and normal** to create plane by specifying a point on plane and two vectors:
        - Enter **x**, **y** and **z** values for the **Point**.
        - Enter **x**, **y** and **z** values for the **Normal direction**.
        - Select (optionally) the **Reference direction** option to specify **x**, **y** and **z** values for the second vector.

    Or

    - Select **Three points** radio button to create a plane by specifying three points:
        - Enter **x, y and z** values for the **First** point.
        - Enter **x, y and z** values for the **Second** point.
        - Enter **x, y and z** values for the **Third** point.
- Press **Create** button to confirm choices and create a plane.

The created plane appears in the table of the **GD&T** view and under the particular construction geometry model in the **Geometry Details** view. In case the plane element is selected in the table, the **Plane** wizard appears under the table. The plane parameters can be updated by entering new values and pressing the **Update** button. The plane can be removed by pressing **Delete** button. In case the plane has

an association with some other elements, the **Delete** button is disabled.

Create derived shape elements

Follow the routine to create a derived shape aspect in the **GD&T** view table:

- Ensure that **Derived features** and **Integral features** filtering buttons are turned *On*.
- Select an applicable shape aspect in the table. **Note.** If no shape aspects are available in the table, the creation of a derived shape aspect is not possible.
- Right click and select **New Constructive Geometry** menu item in the appeared context sensitive menu. A sub menu appears on the screen, which content depends on the shape aspect was selected previously. Select one of available sub menu entries, such as Apex, Symmetry axis, symmetry plane and etc. **Note.** In our example the selected shape aspect is *cylinder*, so the available derived shape aspect is a 'Symmetry axis'. A new derived shape aspect appears in the table (see Figure 6). The name for this derived shape aspect is set automatically (i.e. 'DS1') and displayed in the **Category** column. A name of a shape aspect, which was selected for the creation of the derived shape aspect is presented in the **Values** column.

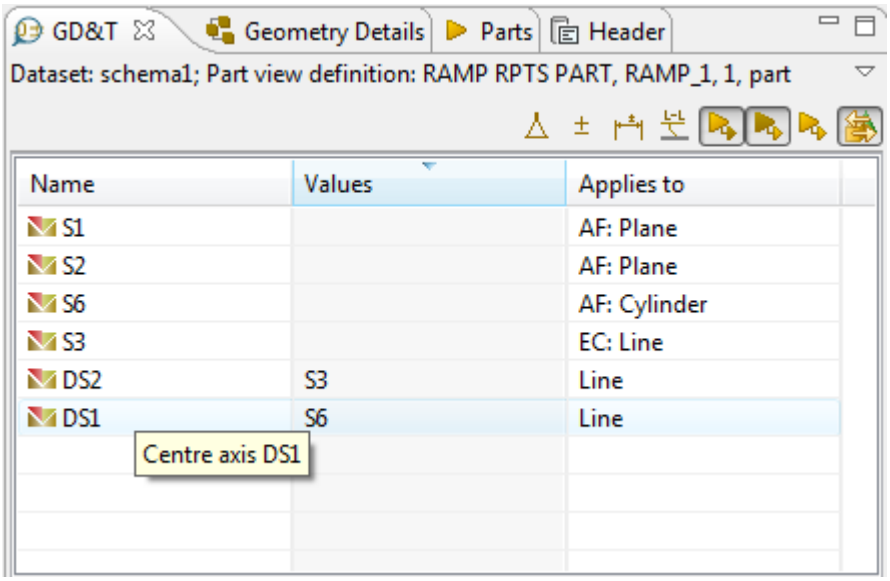

Figure 6. Created derived shape aspect

- Select the derived shape aspect entry in the table ('DS1') to display it highlighted in the **3D View** together with the associated shape aspect ('S6') (see Figure 7).

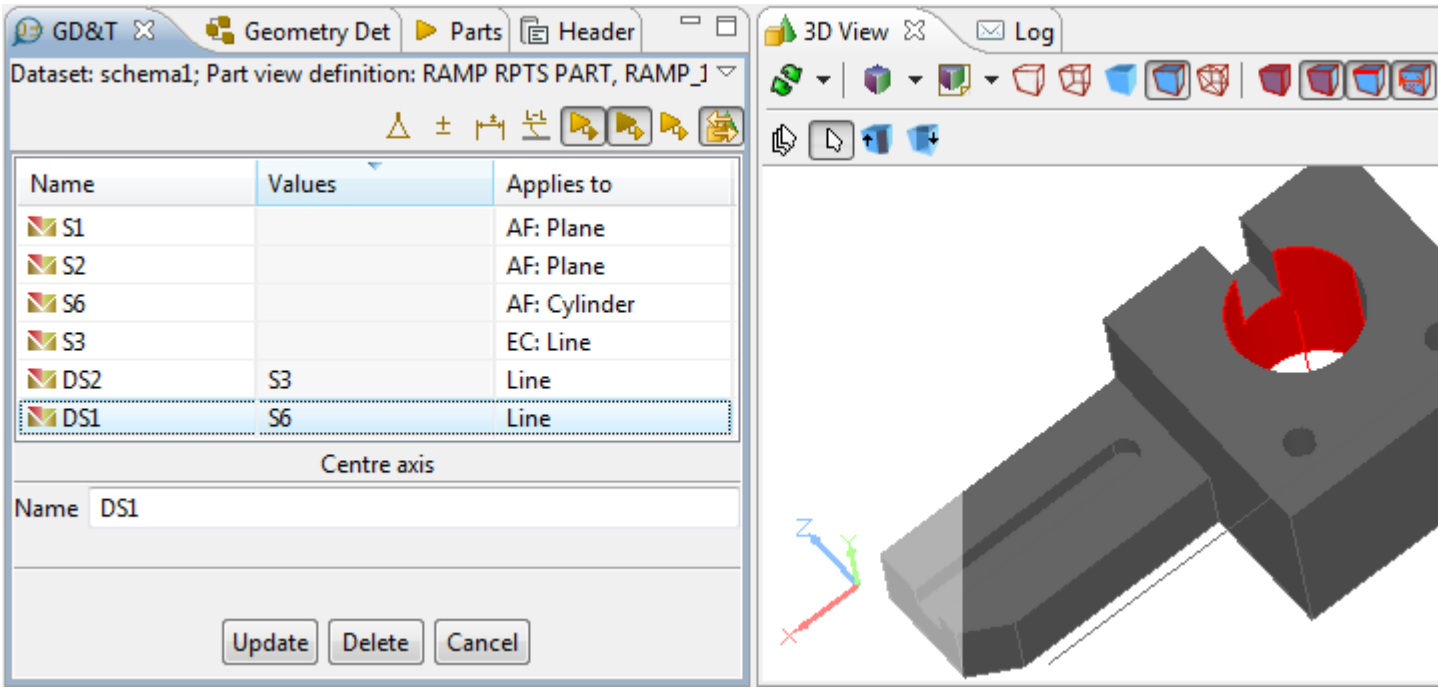

Figure 7. Selected derived shape aspect ('DS1') is highlighted (black line) in 3D View with associated shape aspect ('S6' - highlighted in red colour)

See also:

- Create Apex
- Create Symmetry Axis
- Create Symmetry Plane
- Create Symmetry Point
- Create Common Line
- Create Common Plane
- Create Intersection
- Create Offset
- Create Perpendicular Line
- Create Perpendicular Plane
- Create Tangent Plane

To top

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9.7.2.2.1. Create Apex

Follow the routine to create an apex element in the **GD&T** view:

- Select a shape aspect of type **cone** in the table to create a cartesian *point*, which is an apex of the selected cone.
- Right click and select **New Constructive Geometry -> Apex** item in the appeared context sensitive menu (see Figure 1).

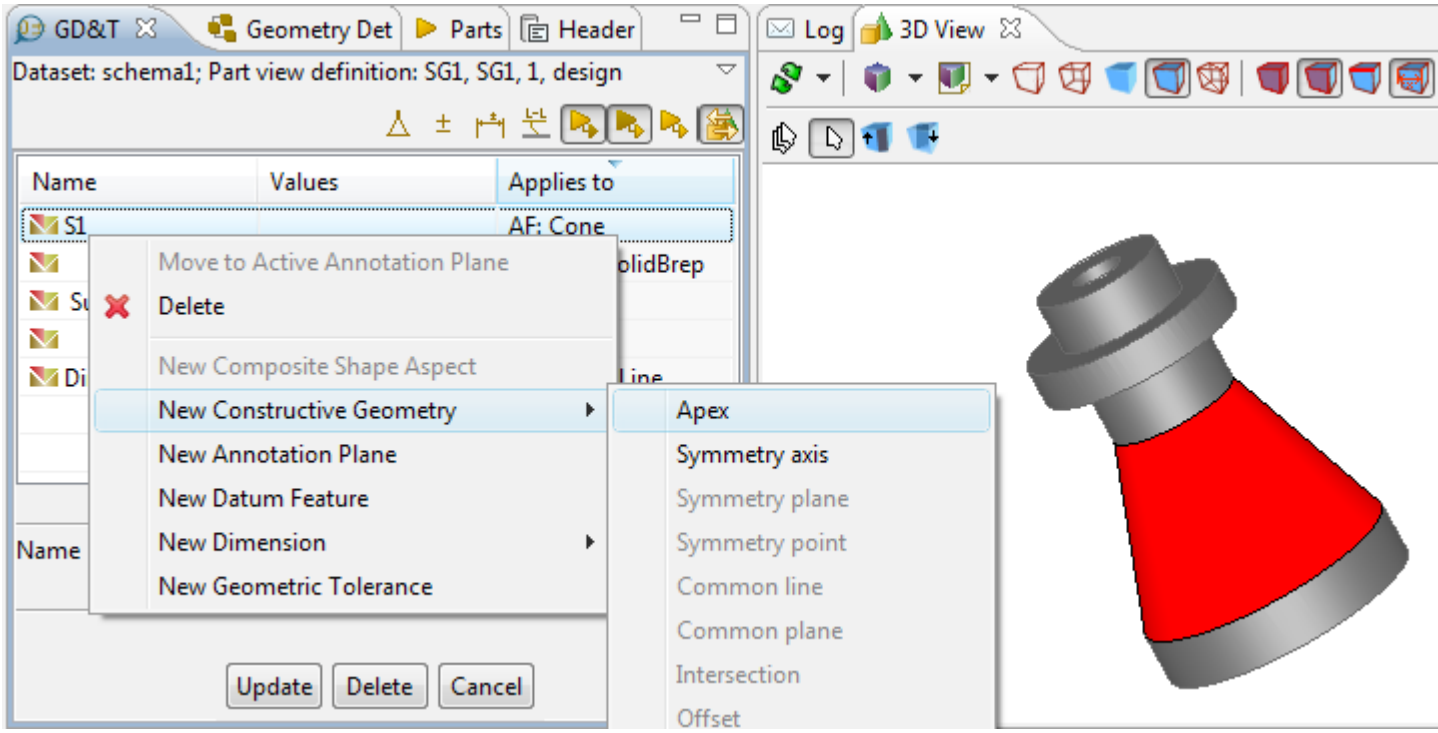

Figure 1. Selected cone shape aspect, menu item to create apex and cone geometry presentation in 3D View

The apex shape aspect with the default name DS<number>appears in the table after menu item is selected. Select the created item in the table to view details and highlight it (together with referenced shape aspects) in the **3D View** (i.e. 'DS1', see Figure 2).

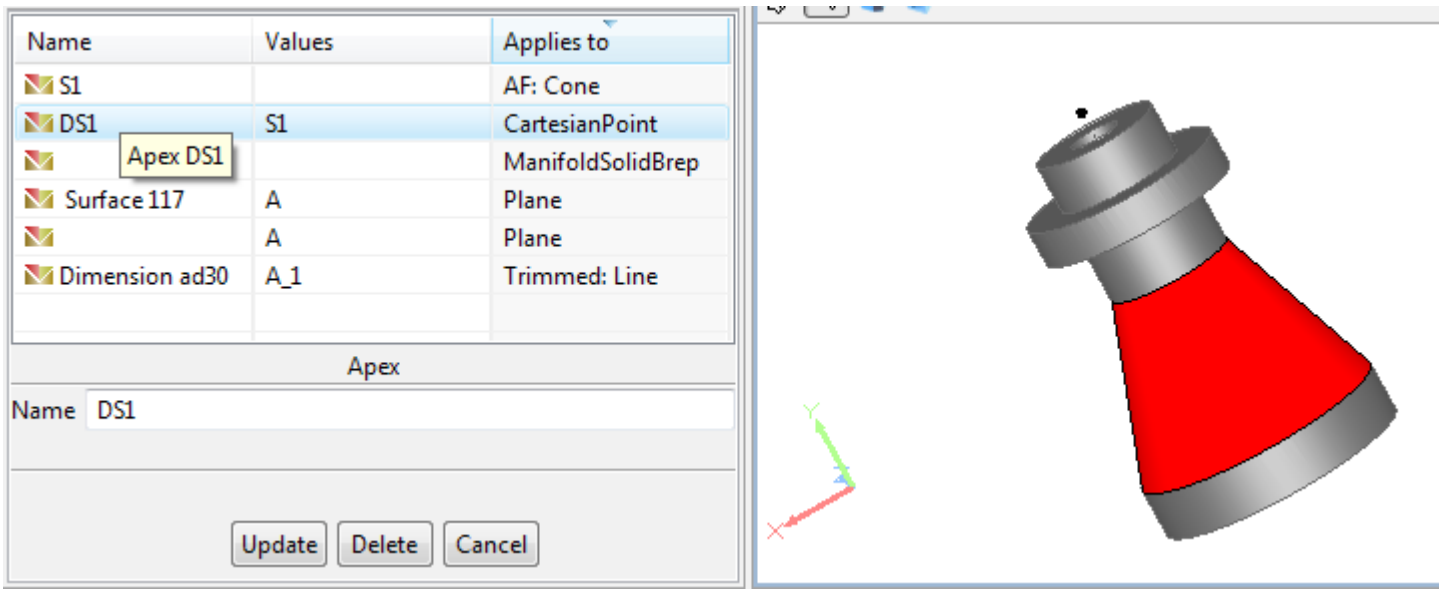

Figure 2. Created apex entry in GD&T table ('DS1')

To top

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9.7.2.2.2. Create Symmetry Axis

Follow the routine to create a symmetry axis in the **GD&T** view:

- Select a shape aspect:

- **cylinder** - to create a line, which is a center line of the selected cylinder.
- **cone** - to create a line, which is a center line of the selected cone.
- **toroid** - to create a line, which is a center line of the selected toroid.
- **revolution** - to create a line, which is a center line of the selected revolution.

- Right click and select **New Constructive Geometry -> Symmetry axis** item in the appeared context sensitive menu (see Figure 1).

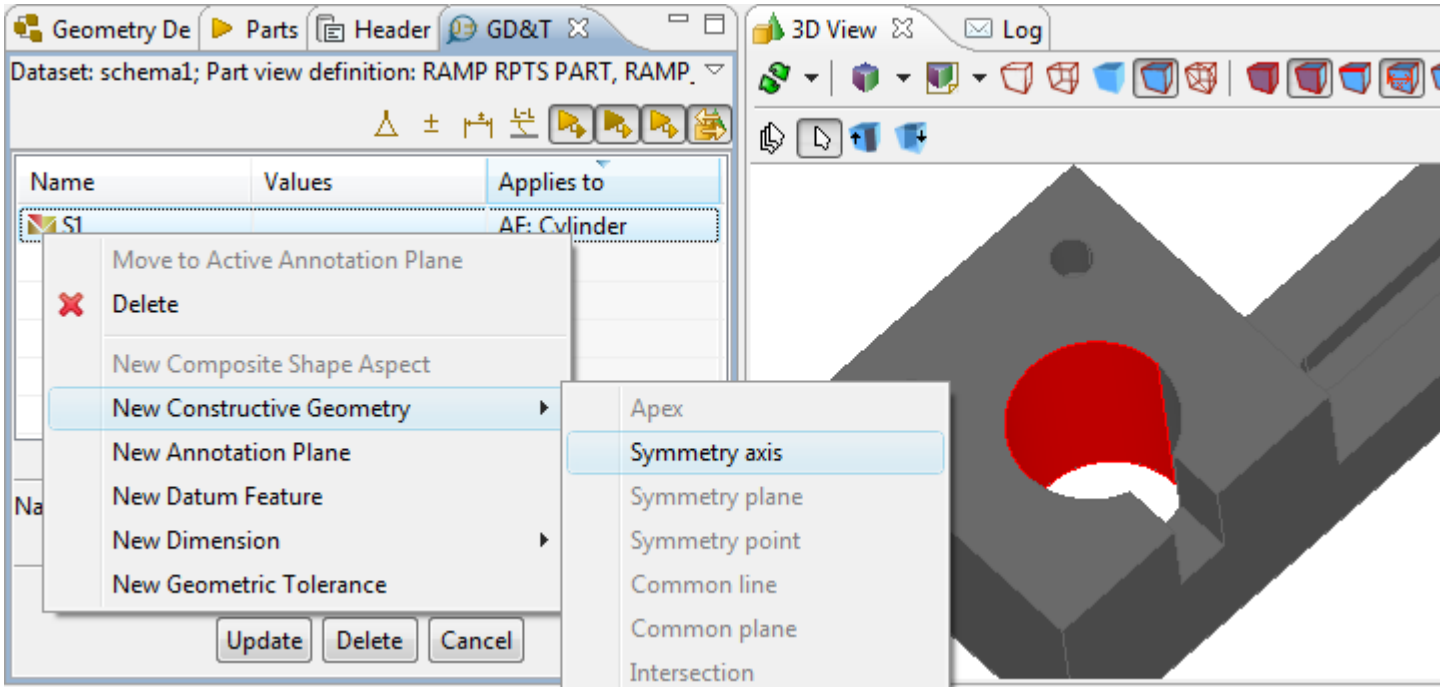

Figure 1. Selected cylinder shape aspect, menu item to create symmetry axis and cylinder geometry presentation in 3D View

The centre axis entry with the default name DS<number> (i.e. 'DS1') appears in the table after menu item is selected. Select the created item in the table to view details and highlight it (together with referenced shape aspects) in the **3D View** (i.e. 'DS1', see Figure 2).

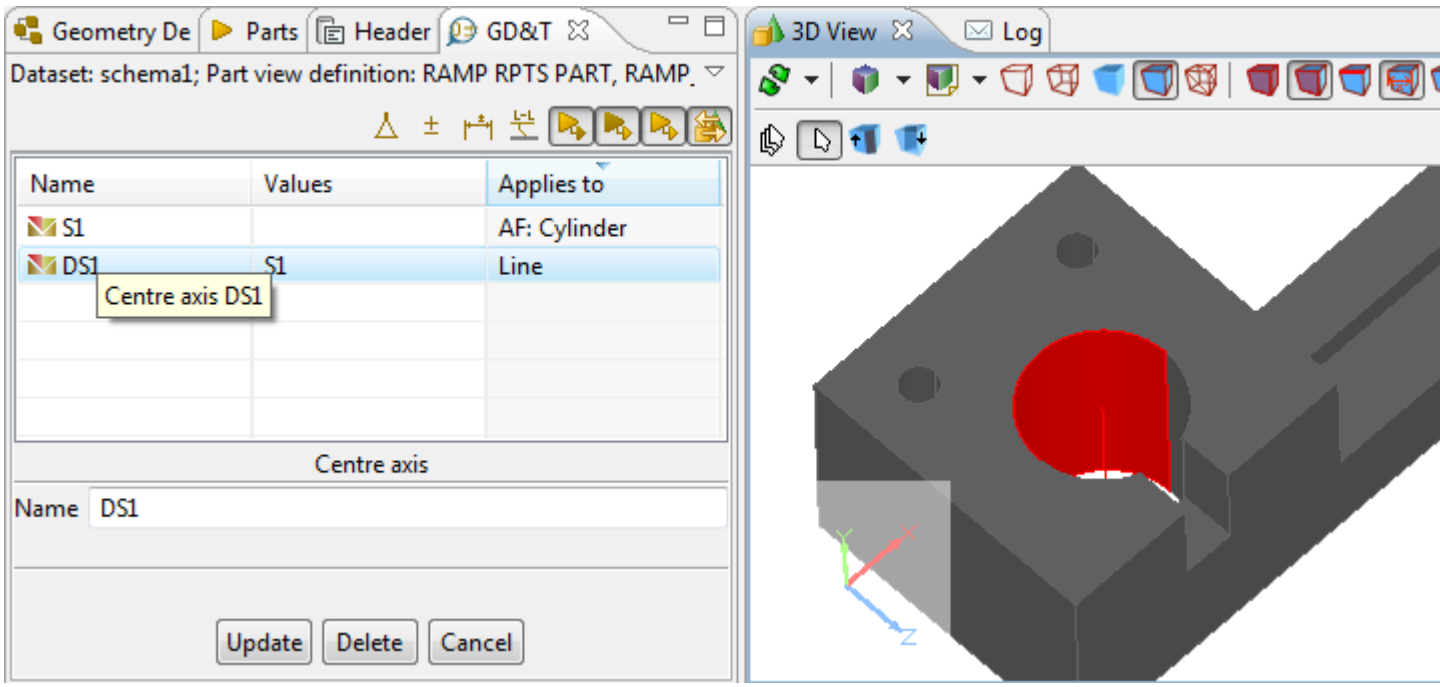

Figure 2. Created symmetry axis in GD&T view (table and details area) and in 3D View.

To top

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9.7.2.2.3. Create Symmetry Plane

Follow the routine to create a symmetry plane in the **GD&T** view:

- Select two shape aspects of type **plane** to create a *plane*, which is a supposed mirror for the selected planes. Each pair of points (one point on one plane, other - on the second) have the symmetrical relationship with a respect to the plane, which is called symmetry plane.
- Right click and select **New Constructive Geometry -> Symmetry plane** item in the appeared context sensitive menu (see Figure 1).

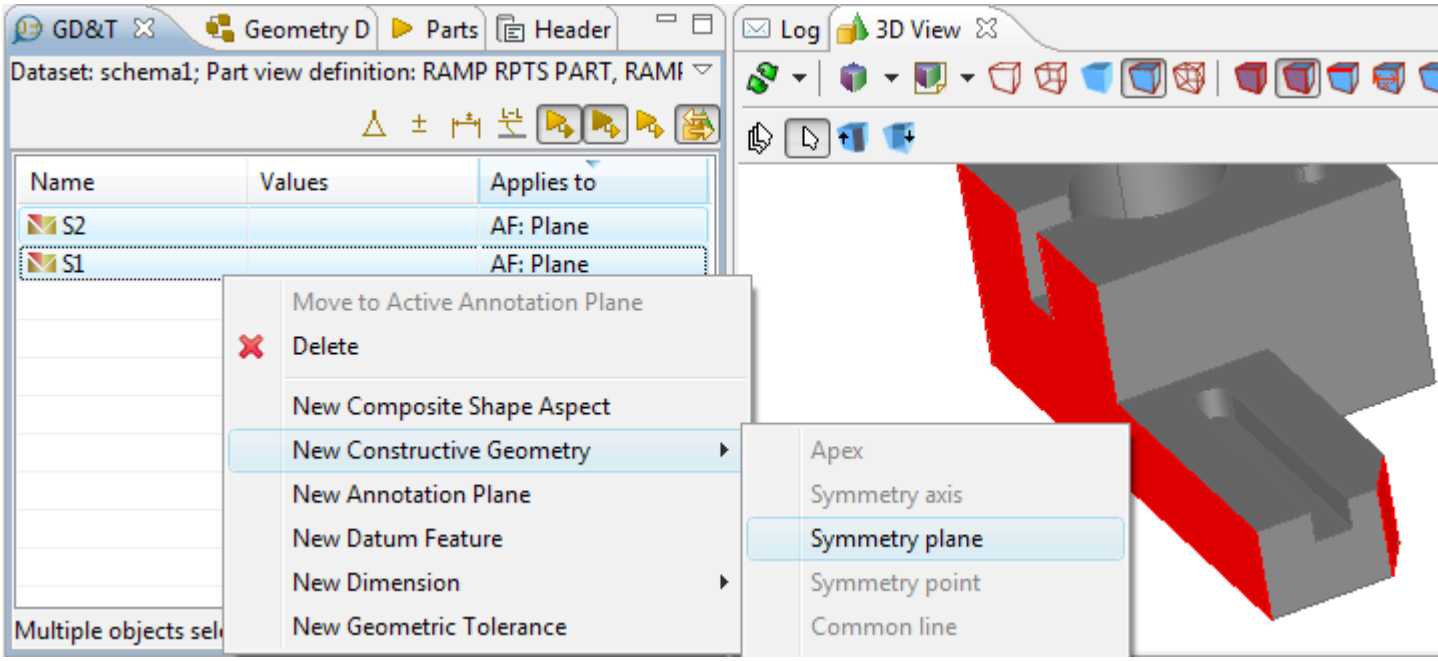

Figure 1. Selected items: two plane items, menu item to create symmetry axis in GD&T view and two planes in 3D View

The centre plane entry with the default name DS<number> appears in the table after menu item is selected. Select the created item in the table to view details and highlight it (together with referenced shape aspects) in the **3D View** (i.e. 'DS1', see Figure 2).

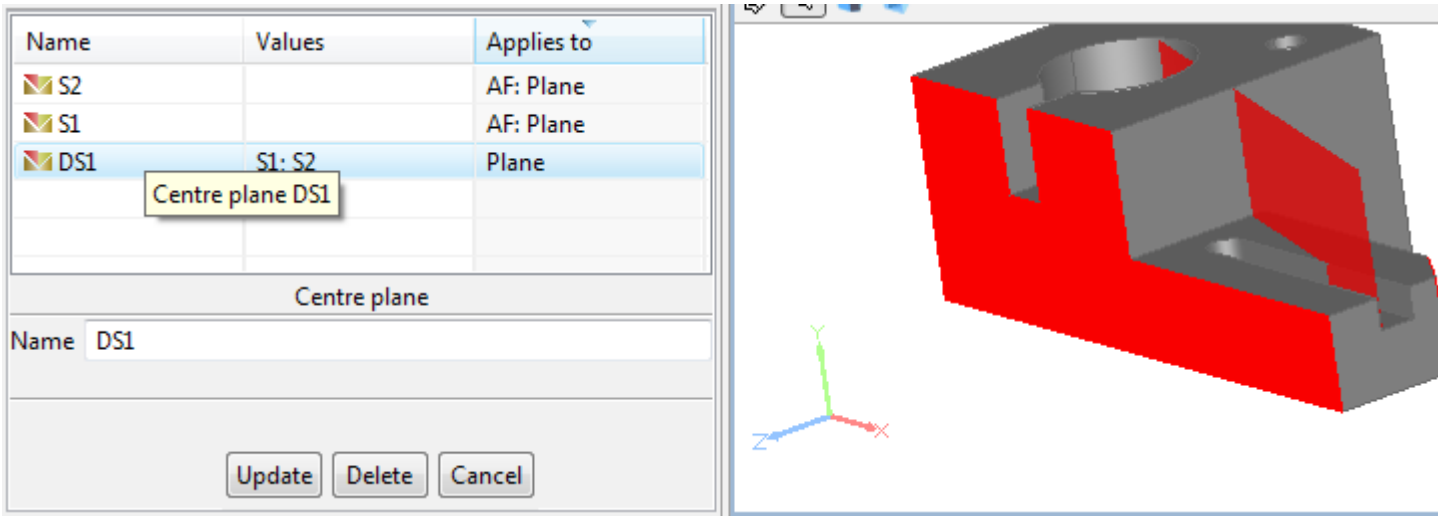

Figure 2. Created symmetry plane in GD&T view (table and details area) and in 3D View

To top

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9.7.2.2.4. Create Symmetry Point

Follow the routine to create a symmetry point:

- Select a shape aspects:
  - **circle** - to create a Cartesian *point*, which is a center point for the selected circle.
  - **sphere** - to create a Cartesian *point*, which is a center point of the selected sphere.
- Right click and select **New Constructive Geometry -> Symmetry point** item in the appeared context sensitive menu (see Figure 1).

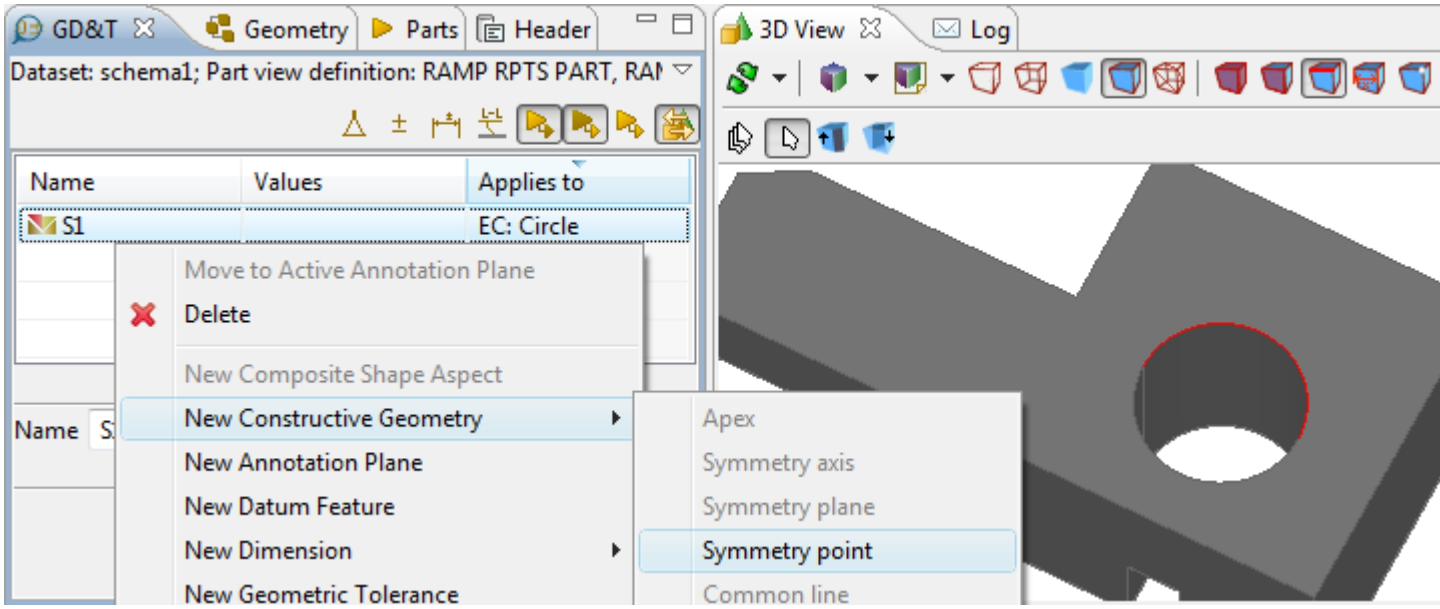

Figure 1. Selected items: circle item, menu item to create symmetry point in GD&T view and circle in 3D View

The centre point entry with the default name DS<number> appears in the table after menu item is selected. Select the created item in the table to view details and highlight it (together with referenced shape aspects) in the **3D View** (i.e. 'DS1', see Figure 2).

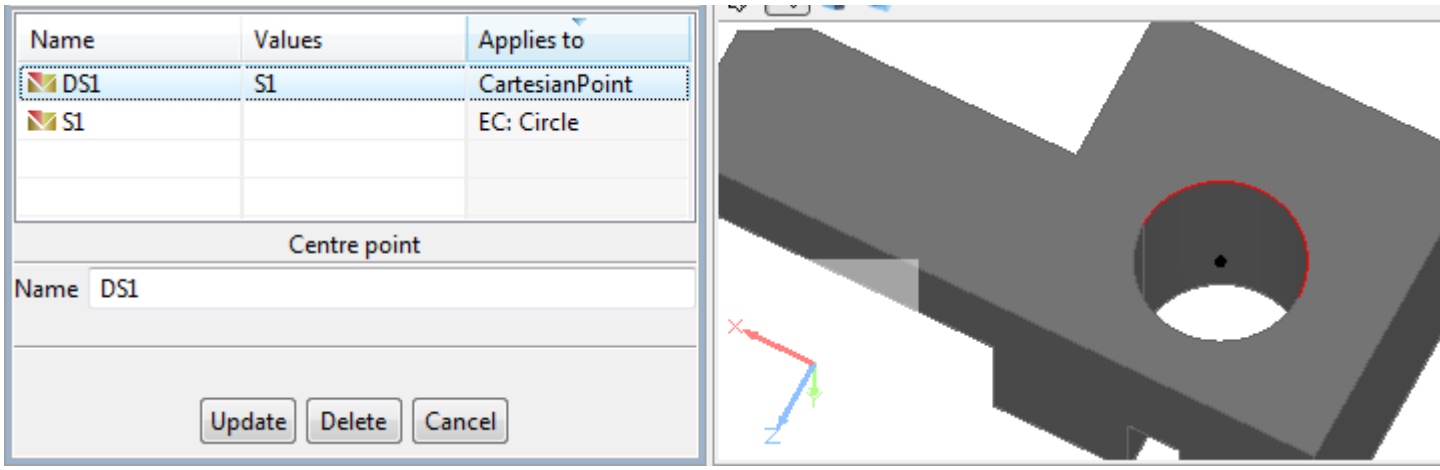

Figure 2. Created symmetry point in GD&T view (table and details area) and in 3D View.

To top

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9.7.2.2.5. Create Common Line

Follow the routine to create a common line in the **GD&T** view:

- Select two shape aspects of type **points** (vertex point, centre point and etc.) to create a *line*, which goes through the selected points.
- Right click and select **Create Constructive Geometry -> Common Line** item in the appeared context sensitive menu (see Figure 1).

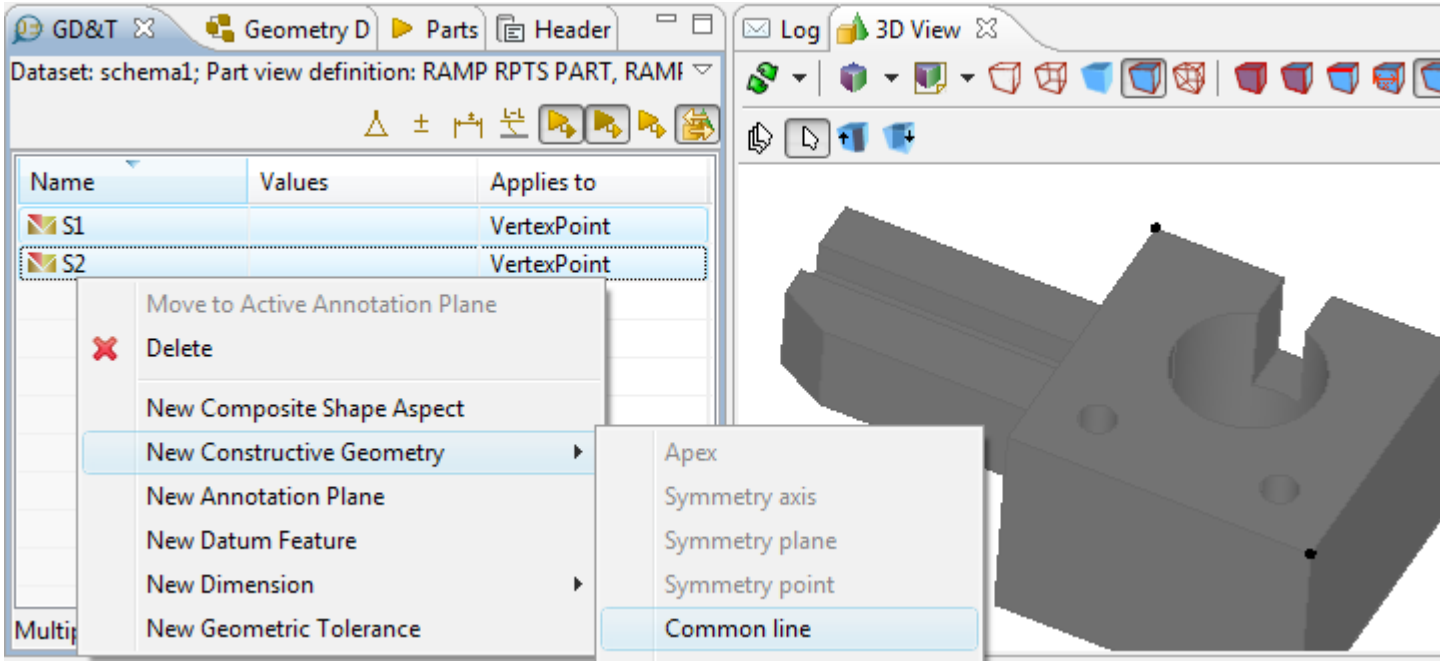

Figure 1. Selected items: two vertex point items, menu item to create common line in GD&T view and two points in 3D View

The geometric alignment entry with the default name DS<number> appears in the table after menu item is selected. Select the created item in the table to view details and highlight it (together with referenced shape aspects) in the **3D View** (i.e. 'DS1', see Figure 2).

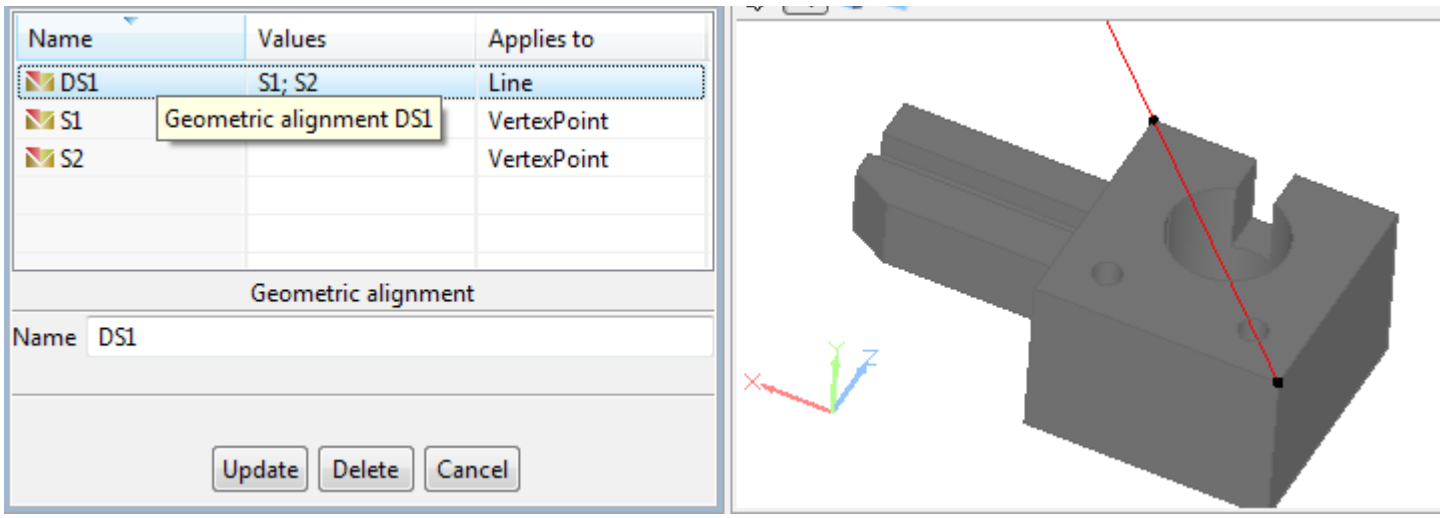

Figure 2. Created common line in GD&T view (table and details area) and in 3D View

To top

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9.7.2.2.6. Create Common Plane

Follow the routine to create a common plane in the **GD&T** view:

- Select two shape aspects:
  - two **lines** - to create a *plane*, which goes through the selected lines.
  - **line** and **point** - to create a *plane*, which goes through the selected line and the specified point.
- Right click and select **Create Constructive Geometry -> Common Plane** item in the appeared context sensitive menu (see Figure 1).

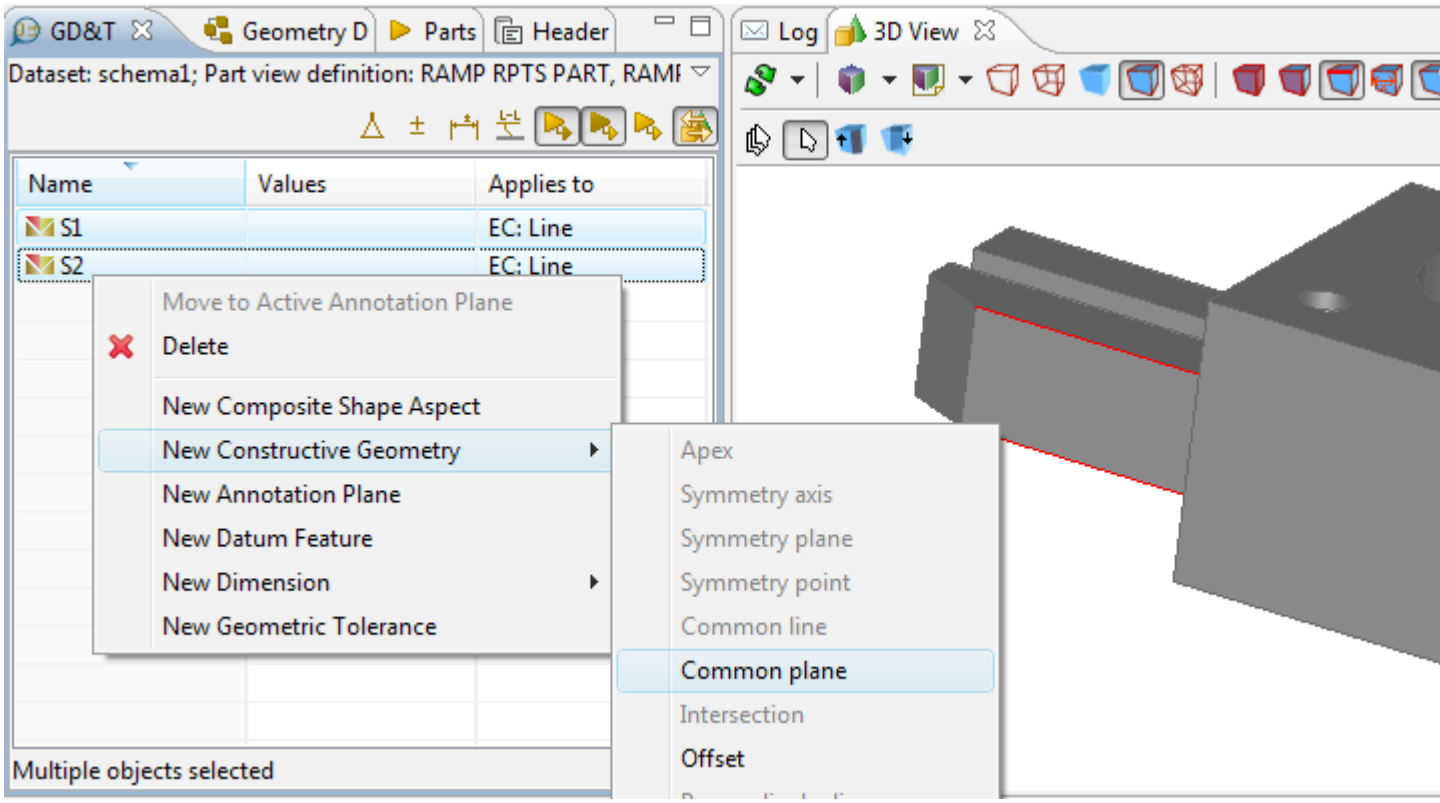

Figure 1. Selected items: two lines, menu item to create common plane in GD&T view and two lines in 3D View

The geometric alignment entry with the default name DS<number> appears in the table after menu item is selected. Select the created item in the table to view details and highlight it (together with referenced shape aspects) in the **3D View** (i.e. 'DS1', see Figure 2).

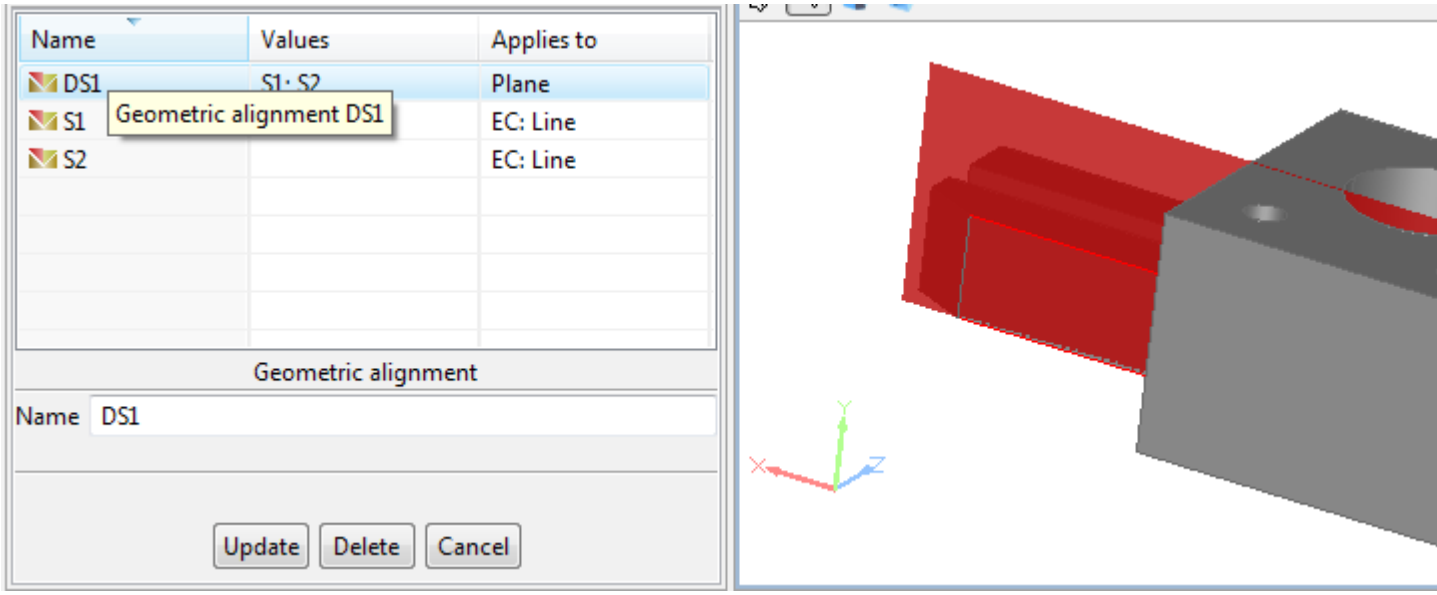

Figure 2. Created common plane in GD&T view (table and details area) and in 3D View

To top

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9.7.2.2.7. Create Intersection

Follow the routine to create an intersection of two elements in the **GD&T** view:

- Select two shape aspects:
  - two **lines** - to create a Cartesian *point*, which is an intersection of two non parallel lines.
  - two **planes** - to create a *line*, which is an intersection of two non parallel planes.
  - **plane** and **cylinder** - to create a *circle*, an *ellipse* or a *line*:
    - in case the selected plane goes across the cylinder, an intersection is a *circle* or an *ellipse*.
    - in case the selected plane goes along the cylinder, an intersection is a *line* (the result can be one or two lines).
  - **plane** and **line** - to create a Cartesian *point*, which is an intersection of non parallel objects - line and plane.
  - **line** and **cylinder** - to create a Cartesian *point*, which is an intersection of non parallel objects - line and cylinder.
  - **line** and **sphere** - to create a Cartesian *point*, which is an intersection of line and sphere.
  - **plane** and **sphere** - to create a *circle* or a Cartesian *point*:
    - in case the selected plane crosses the sphere, an intersection is a *circle*.
    - in case the selected plane is a tangent plane for the selected sphere, an intersection is a Cartesian *point*.
- Right click and select **Create Constructive Geometry -> Intersection** item in the appeared context sensitive menu (see Figure 1).

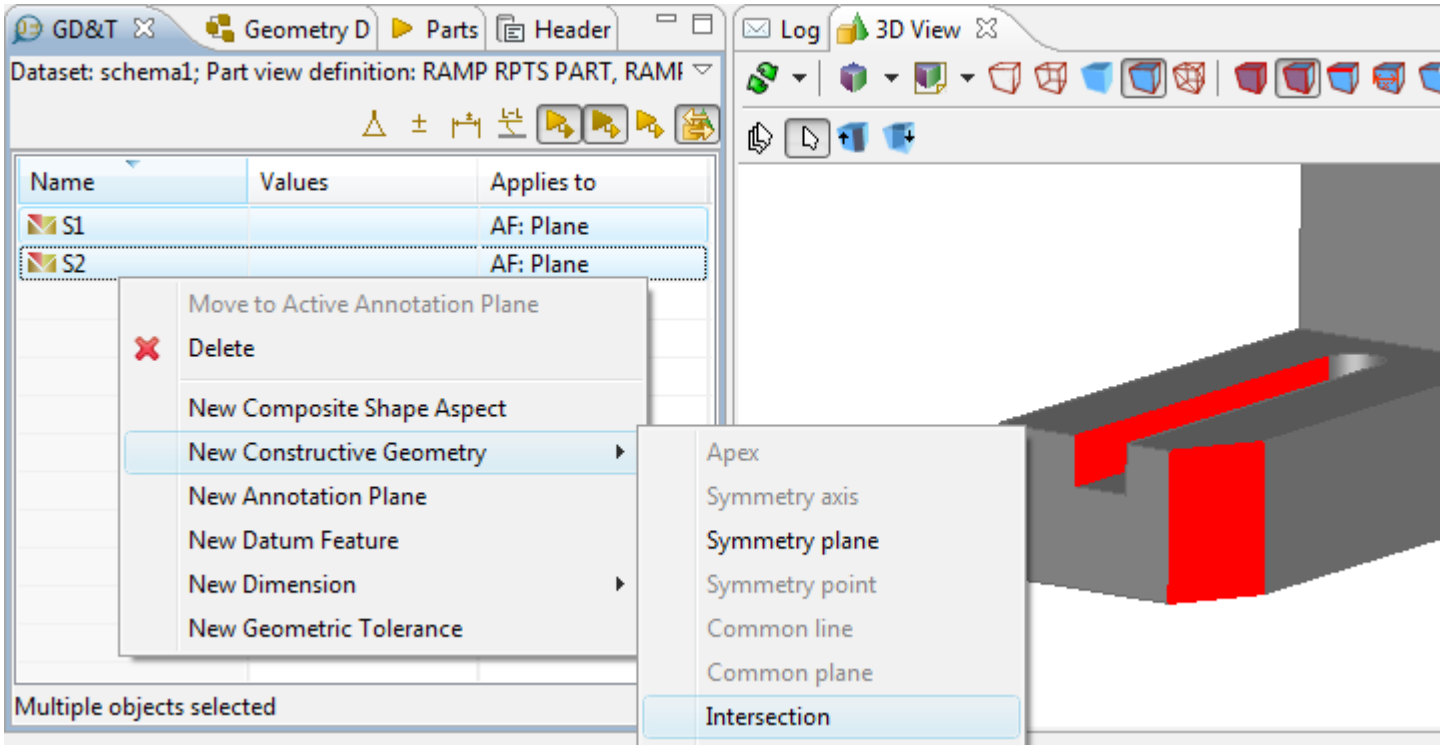

Figure 1. Selected items: plane and cylinder, menu item to create intersection in GD&T view and plane and cylinder in 3D View

The intersection entry with the default name DS<number> appears in the table after menu item is selected. Select the created item in the table to view details and highlight it (together with referenced shape aspects) in the **3D View** (i.e. 'DS1', see Figure 2).

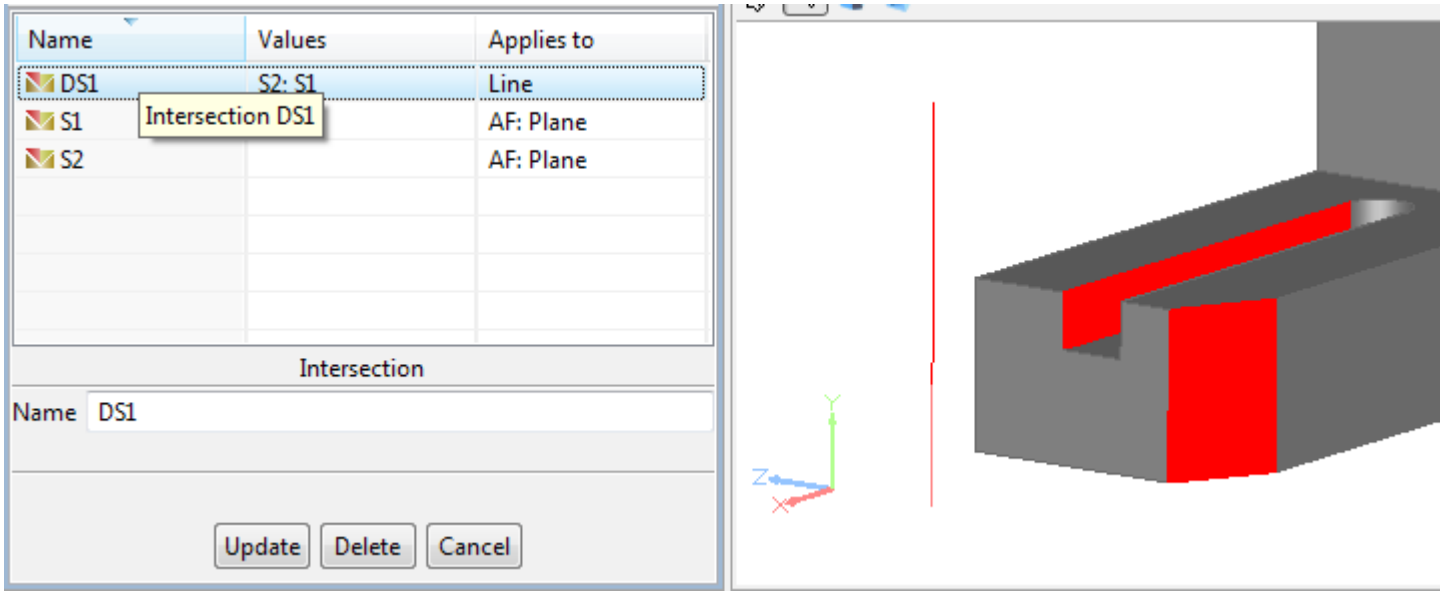

Figure 2. Created imaginary intersection (*Line*) in GD&T view (table and details area) and in 3D View

To top

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9.7.2.2.8. Create Offset

Follow the routine to create an offset in the **GD&T** view:

- Select a shape aspect (for example *point, line* or *plane*).
- Right click and select **Create Constructive Geometry -> Offset** item in the appeared context sensitive menu (see Figure 1).

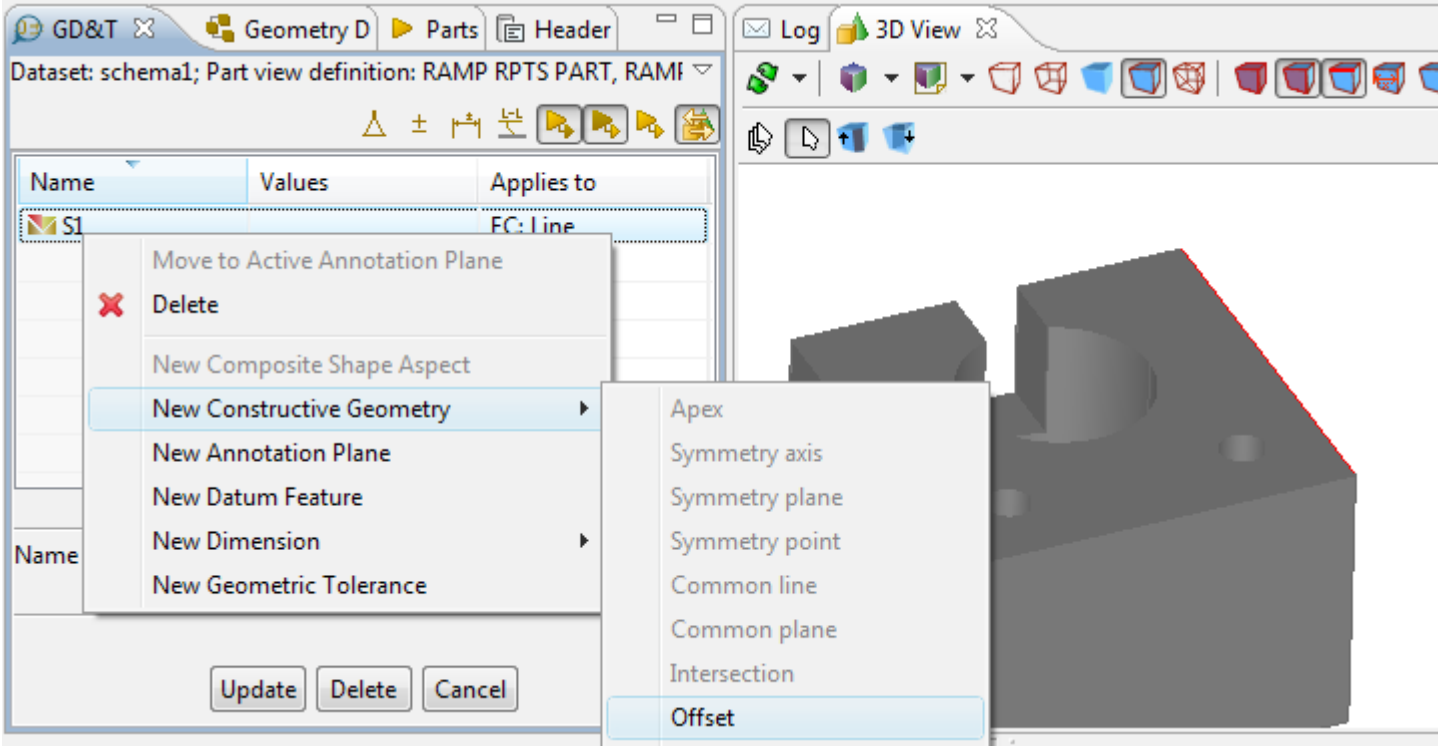

Figure 1. Selected items: line, menu item to create offset in GD&T view and line in 3D View

- The **Offset** wizard dialogue appears in the details area of the **GD&T** view. Enter the offset parameters into the wizard fields (see Figure 2):

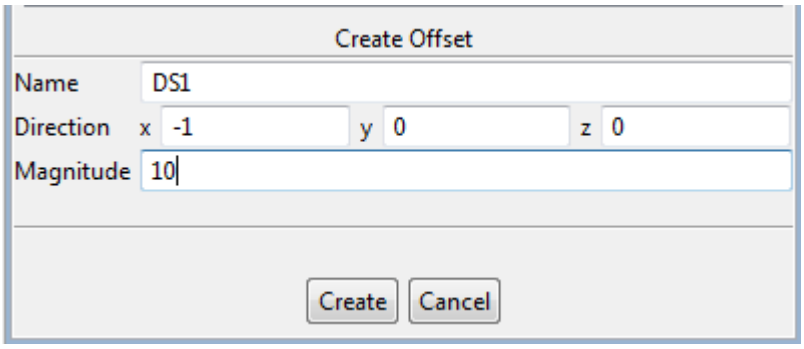

Figure 2. Offset wizard dialogue

  - **Name** field is filled automatically with the default value for derived shapes (DS and number). You can enter or edit the name of the offset if needed.
  - Enter **Direction** values into the **x**, **y** and **z** fields.
  - Enter the **Magnitude** value into the text field.
  - Press the **Create** button to confirm your selection.

The offset entry appears in the table with the default name DS<number>. Select the created item in the table to view details and highlight it (together with referenced shape aspects) in the **3D View** (i.e. 'DS1', see Figure 3).

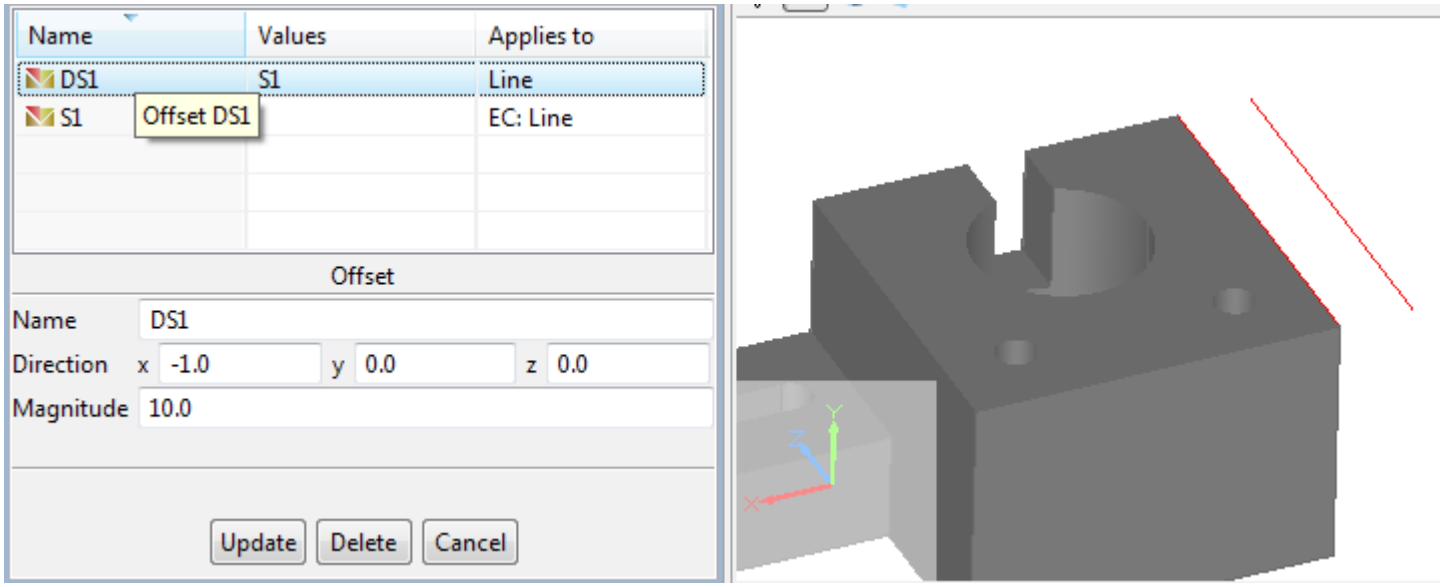

Figure 3. Created line offset in GD&T view (table and details area) and in 3D View

To top

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9.7.2.2.9. Create Perpendicular Line

Follow the routine to create a perpendicular line in the **GD&T** view:

- Select two shape aspects:
  - **plane** and **point** - to create a *line*, which is perpendicular to the selected plane and goes through the specified point.
  - **sphere** and **point** - to create a *line*, which is a sphere center axis through the specified point.
  - **cylinder** ant **point** - to create a *line*, which is perpendicular to the center axis of the selected cylinder and goes through the specified point.
  - **cone** and **point** - to create a *line*, which is perpendicular to the surface of the selected cone at the specified point.
- Right click and select **Create Constructive Geometry -> Perpendicular line** item in the appeared context sensitive menu (see Figure 1).

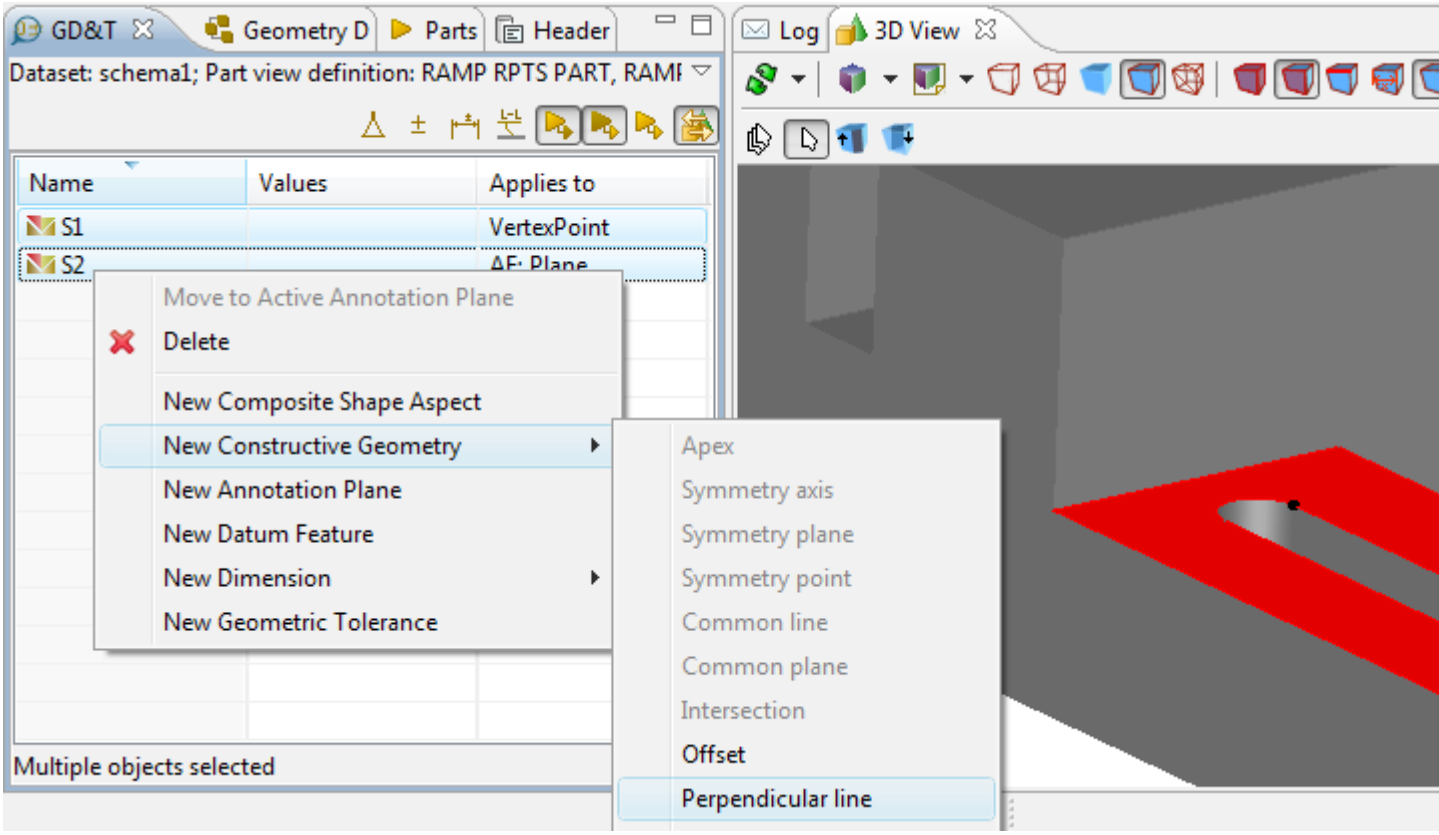

Figure 1. Selected items: plane and point, menu item to create perpendicular line in GD&T view and surface and point in 3D View

The perpendicular entry with the default name DS<number> appears in the table after menu item is selected. Select the created item in the table to view details and highlight it (together with referenced shape aspects) in the **3D View** (i.e. 'DS1', see Figure 2).

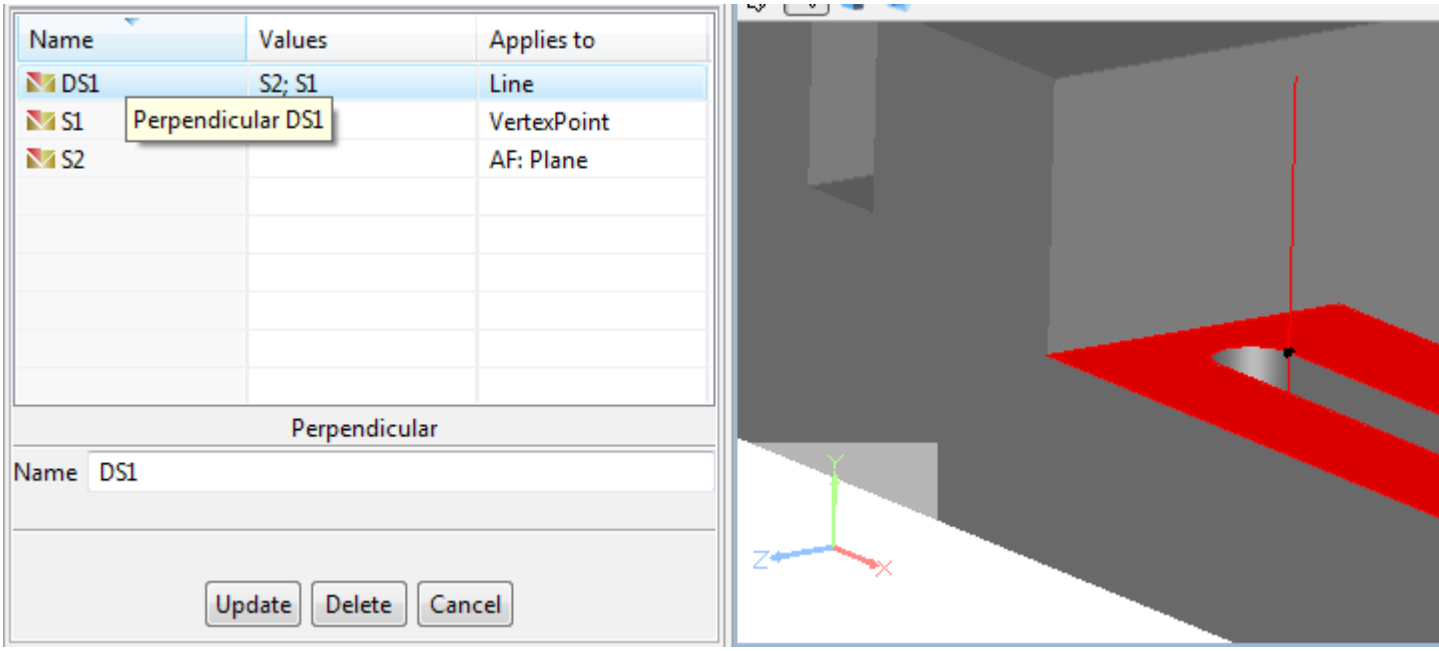

Figure 2. Created perpendicular line in GD&T view (table and details area) and in 3D View

To top

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9.7.2.2.10. Create Perpendicular Plane

Follow the routine to create a perpendicular plane in the **GD&T** view:

- Select two shape aspects:
  - **plane** and **line** - to create a *plane*, which is perpendicular to the selected plane and goes through the specified line.
  - **line** and **point** - to create a *plane*, which is perpendicular to the selected line and goes through the specified point.
  - **cylinder** and **point** - to create a *plane*, which is perpendicular to the center axis of the selected cylinder and goes through the specified point.
- Right click and select **Create Constructive Geometry -> Perpendicular plane** item in the appeared context sensitive menu (see Figure 1).

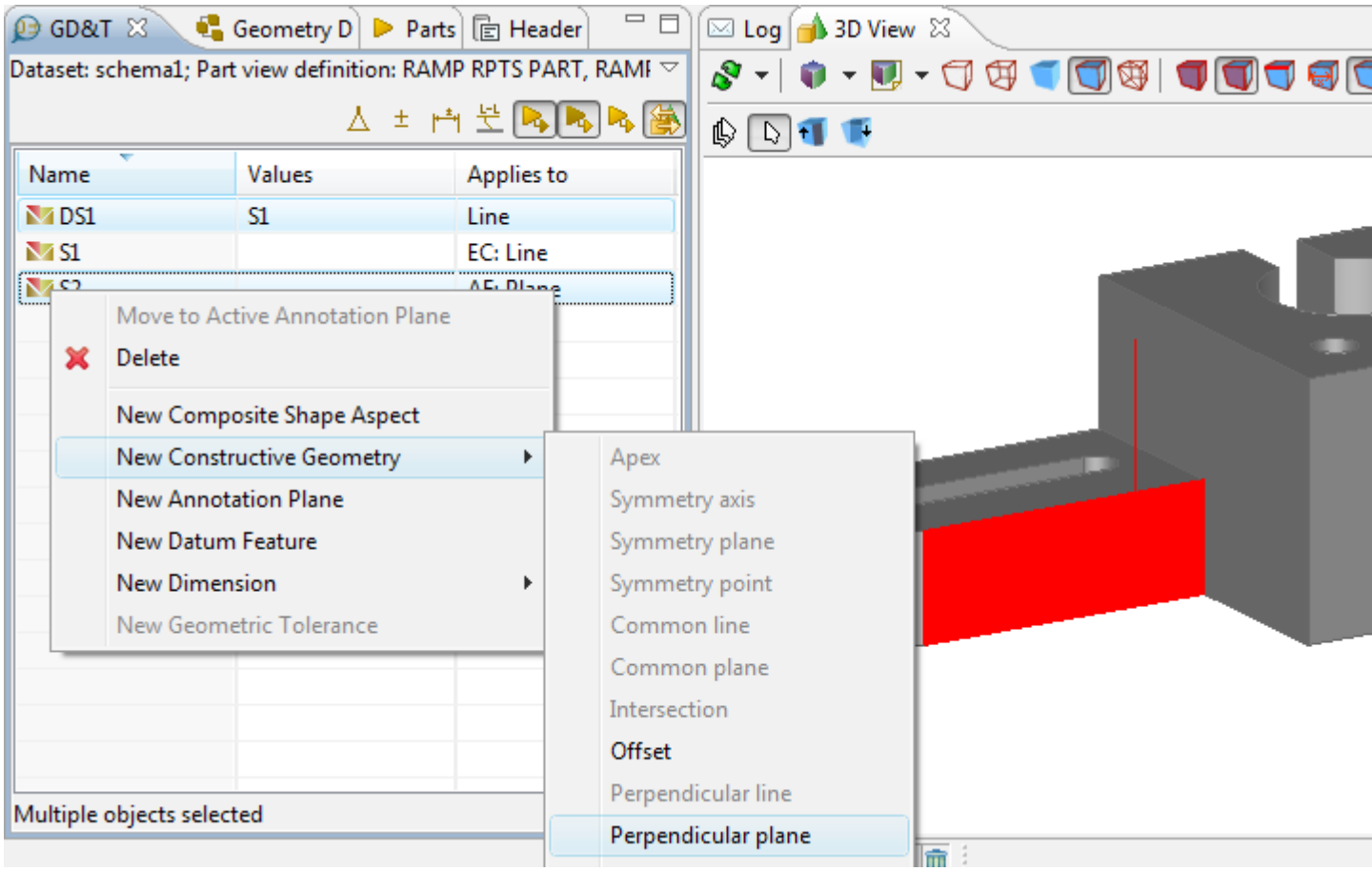

Figure 1. Selected items: plane and line, menu item to create perpendicular plane in GD&T view and surface and line in 3D View

The perpendicular entry with the default name DS<number> appears in the table after menu item is selected. Select the created item in the table to view details and highlight it (together with referenced shape aspects) in the **3D View** (i.e. 'DS1', see Figure 2).

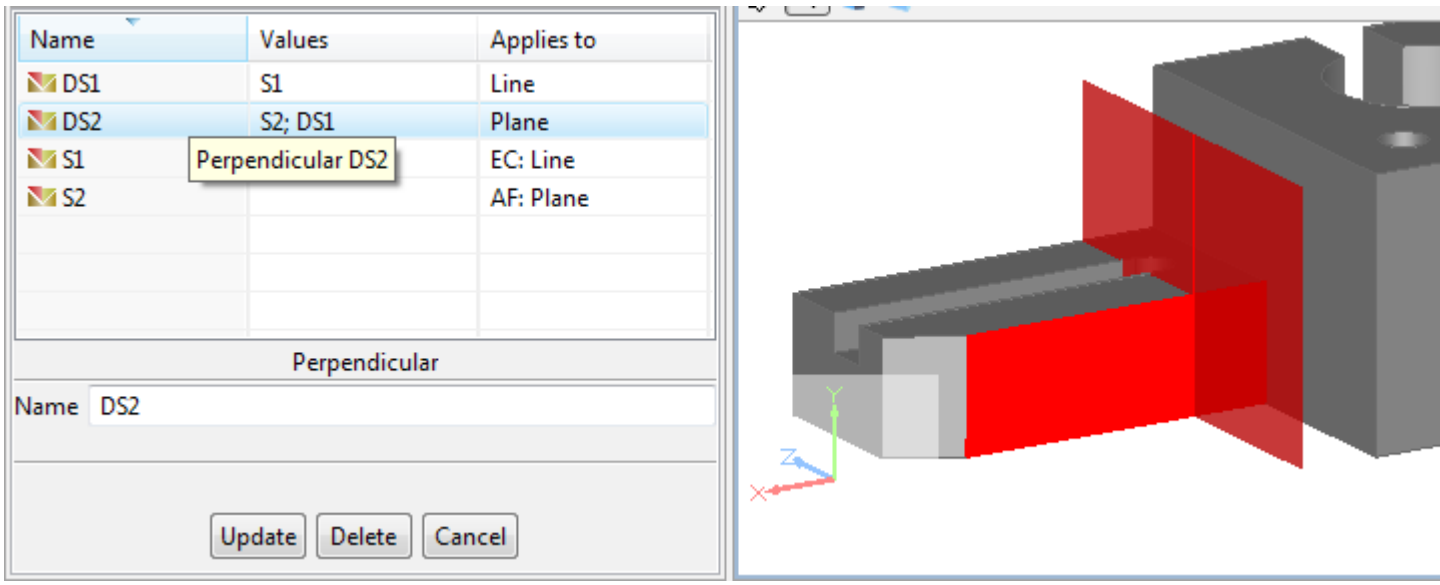

Figure 2. Created perpendicular plane in GD&T view (table and details area) and in 3D View

To top

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9.7.2.2.11. Create Extension

Follow the routine to create an extension in the **GD&T** view:

- Select a shape aspect of type **plane** to create a *plane*, which is an extension of the selected plane.
- Right click and select **Create Constructive Geometry -> Extension** item in the appeared context sensitive menu (see Figure 1).

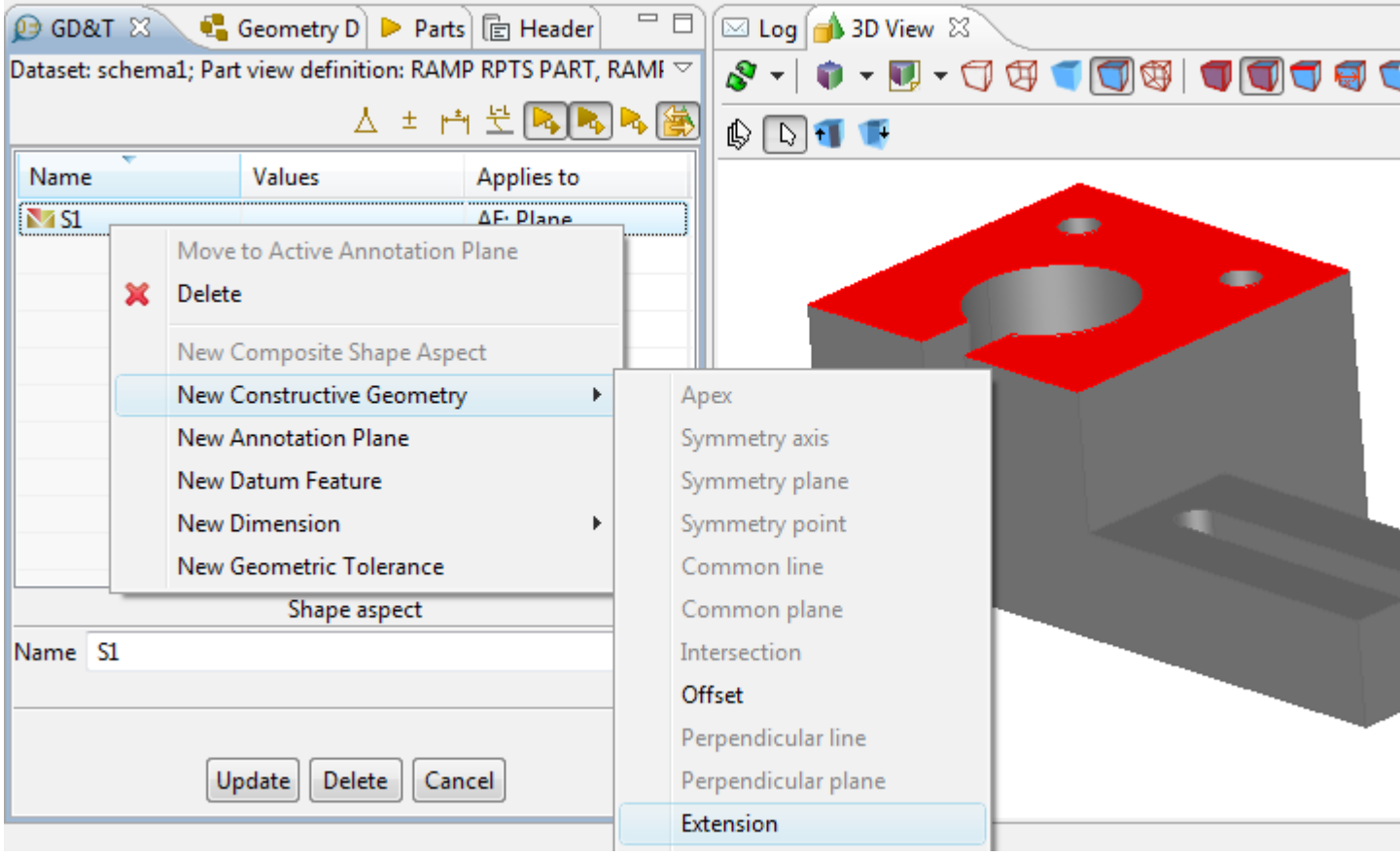

Figure 1. Selected items: plane, menu item to create extension in GD&T view and surface in 3D View

The extension entry with the default name DS<number> appears in the table after menu item is selected. Select the created item in the table to view details and highlight it (together with referenced shape aspects) in the **3D View** (i.e. 'DS1', see Figure 2).

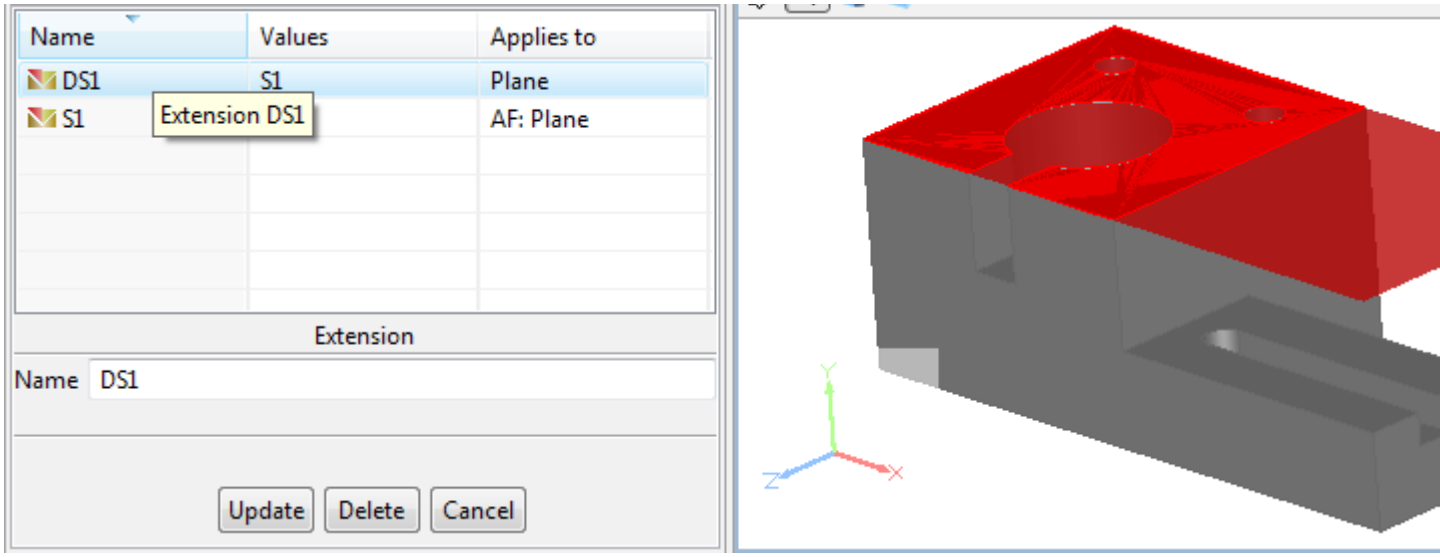

Figure 2. Created extension in GD&T view (table and details area) and in 3D View

To top

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9.7.2.2.12. Create Tangent Plane

Follow the routine to create a tangent plane:

- Activate the **3D View**. Ensure, that need selection mode buttons are switched *On*.
- Select two shape aspects in the **3D View (**ensure, if need selection mode buttons are switched *On***)**:
  - **cylinder** and **point** (point on surface) - to create a *plane* that touches the selected cylinder at the specified point.
  - **cone** and **point** (point on surface) - to create a *plane* that touches the selected cone surface at the specified point.
  - **sphere** and **point** (point on surface) - to create a *plane* that touches the selected sphere at the specified point.
- Right click and select **Create Constructive Geometry -> Tangent plane** item in the appeared context sensitive menu (see Figure 1).

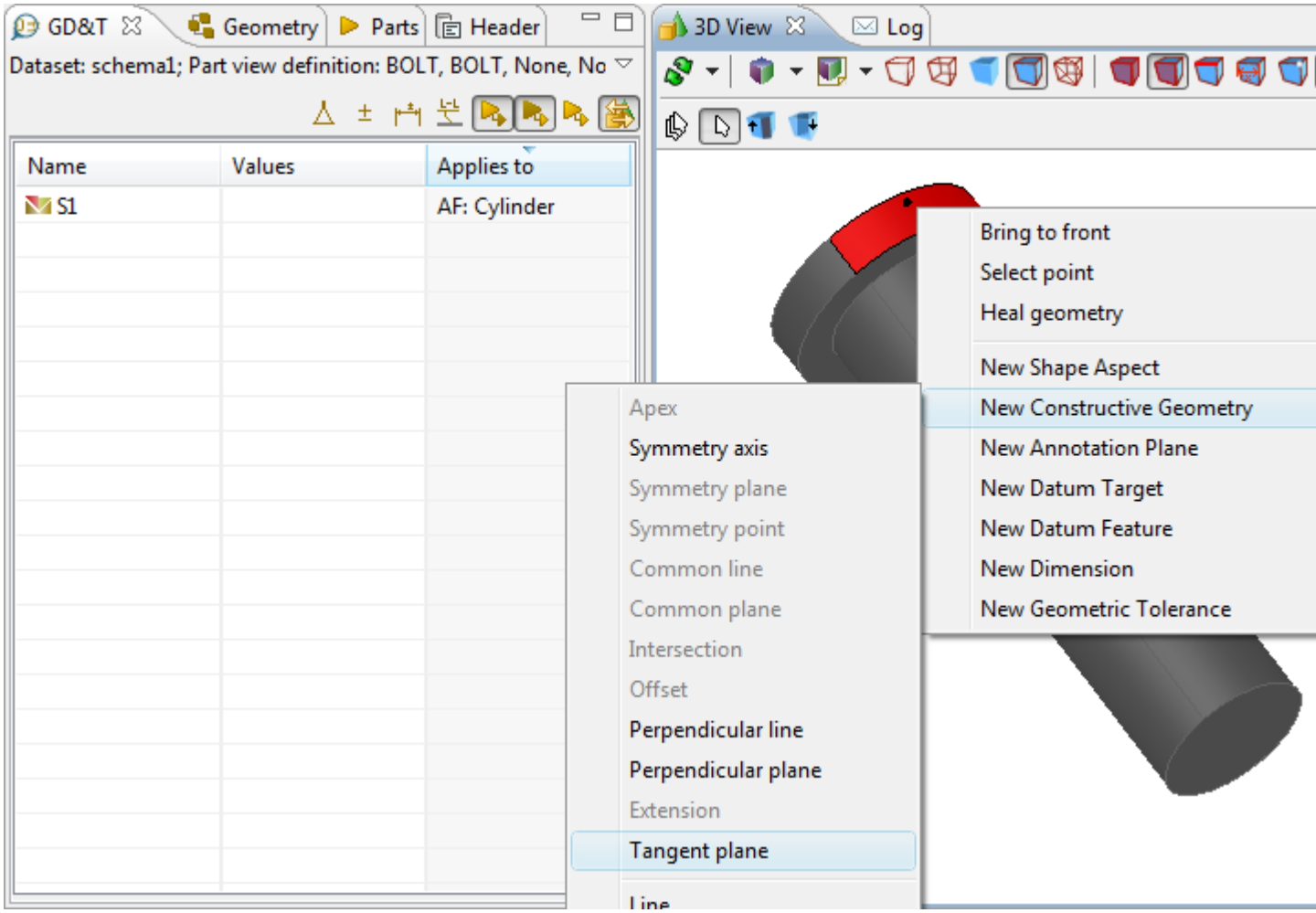

Figure 1. Selected items: cylinder and point, menu item to create tangent plane in GD&T view and cylinder surface and point in 3D View

The tangent entry with the default name DS<number> appears in the table after menu item is selected. Select the created item in the **GD&T** view table to view details and highlight it in the **3D View** (i.e. 'DS1', see Figure 2).

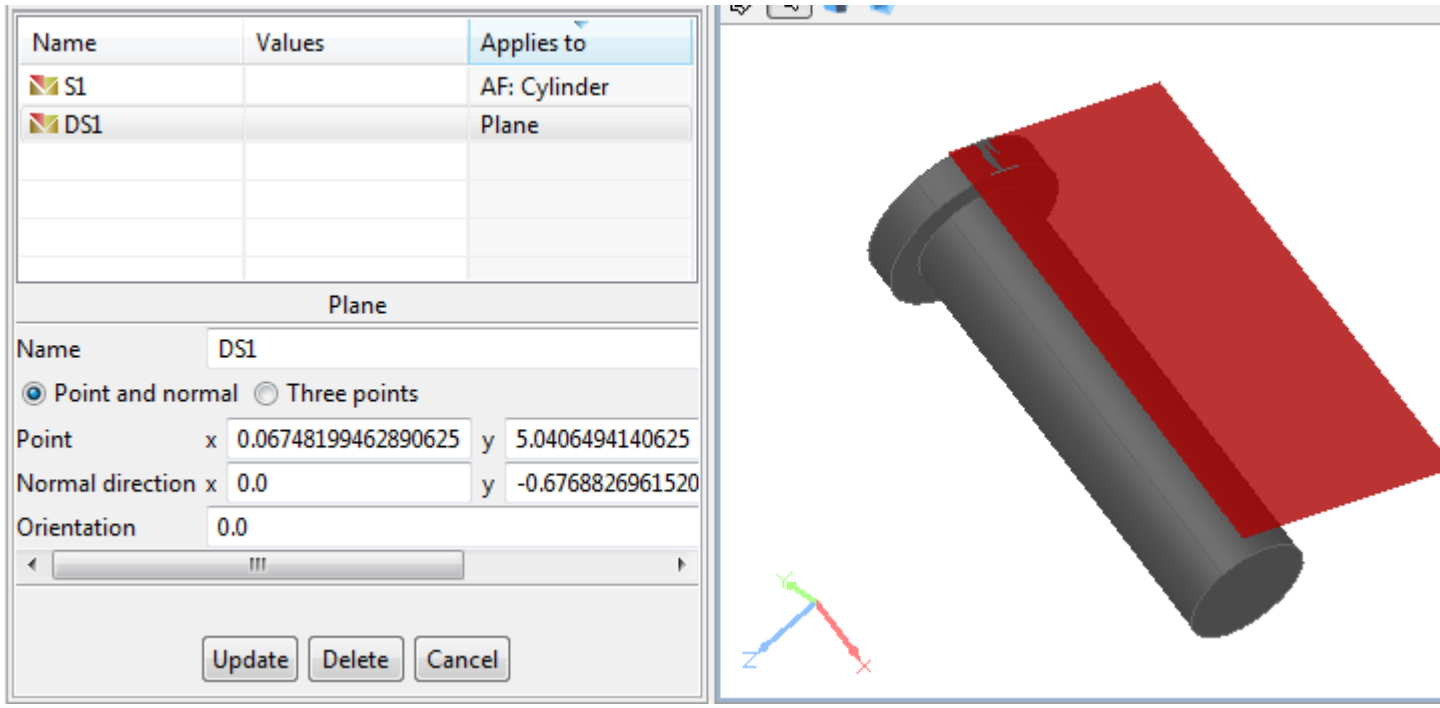

Figure 2. Created tangent plane in GD&T view (table and details area) and in 3D View

To top

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9.7.3. Annotation Plane

An *Annotation Plane* is a conceptual plane containing annotations.
The **GD&T** view displays annotation planes of active **Style/Annotation Model** (which can be selected in the **Geometry Details** view) and they are filtered with a **Shape Aspects** filter.
A user has a possibility to create, modify and delete an annotation plane, move selected annotation items to the active annotation plane.

Create annotation plane

Follow the routine to create an annotation plane in the **GD&T** view table:

- Select a shape aspect in the table. In our example a planar shape aspect (*Plane*) is selected.
  Note. It is not the required selection. In case no shape aspect is selected, more parameters have to be entered manually.
- Right click and select the **New Annotation Plane** item in the appeared context sensitive menu.
- The **Annotation Plane** wizard dialogue appears in the details area of the **GD&T** view (see Annotation Plane Wizard).

The created annotation plane entry appears in the **GD&T** view table (see Figure 1). The created annotation plane is marked (displayed in bold) as a currently active annotation plane.

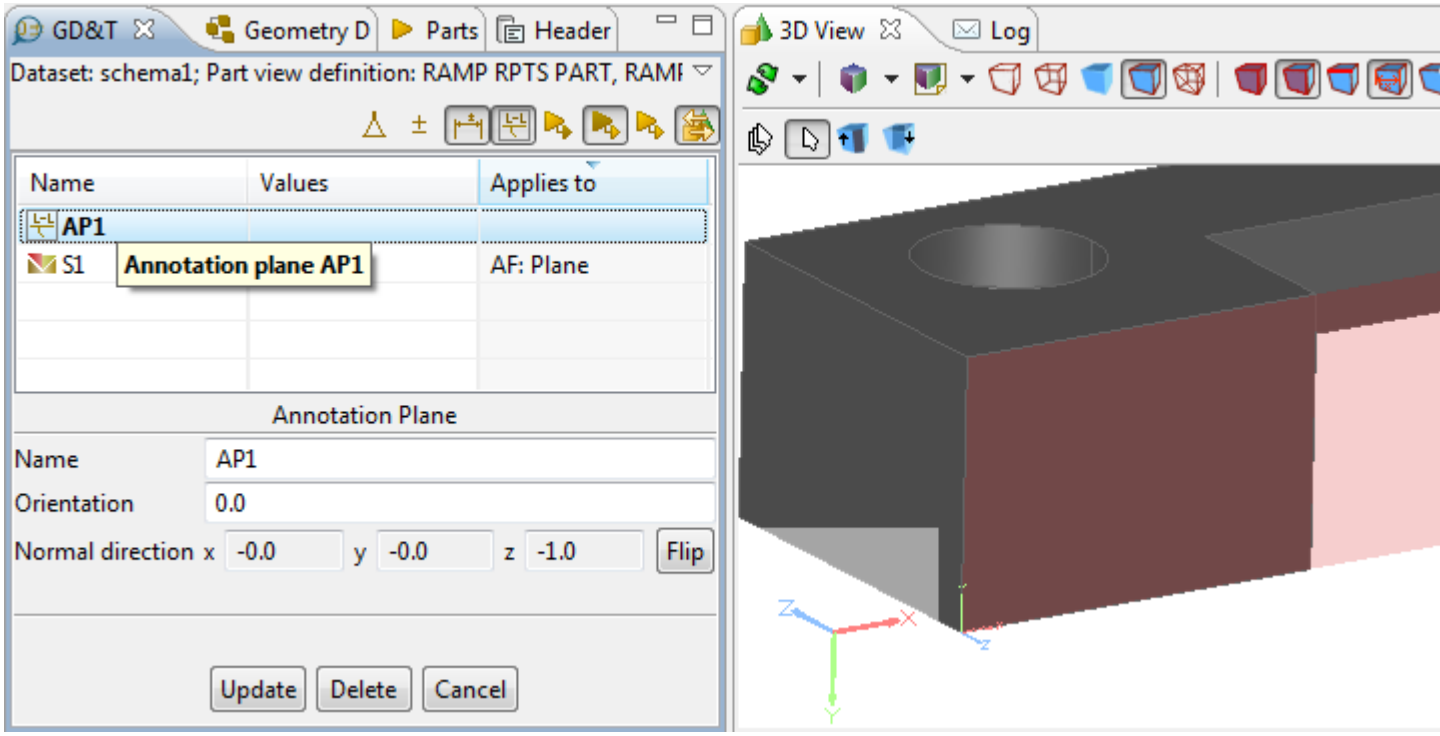

Figure 1. Created annotation plane (highlighted entry in GD&T table) and in 3D View

Let's introduce, how the population of an annotation plane depends on the selection:

1. Selection is a single point and any number (zero as well) of faces/edges. In this case the proposed annotation plane is one going through that point with normal (0, 0, 1). In other words *Point* is set to the selected point, *Normal direction* is set to (0, 0, 1) and *Orientation* is set to zero.
2. Selection is three different points not on the same line and any number (zero as well) of faces/edges. In this case the proposed annotation plane goes through all the selected points with

normal being a cross product between two vectors: the first point to the second point, the first point to the third point. In other words *Point* is set to the first selected point. *Normal direction* is set to a cross product between two vectors: the first point to the second point, the first point to the third point. *Orientation* is set to an angle between two vectors: the first point to the second point, the first point to the third point. You could also say that orientation is along a vector from the first point to the second point.

3. Selection is a single advanced face with a plane as an underlying geometry. In this case tan annotation plane will be the same as the selected plane and with the same normal as the selected advanced face.
4. Selection is a single plane. In this case an annotation plane will be the same as the selected plane.

Annotation Plane Wizard

In this topic, you will find a description of the **Annotation Plane** wizard content (see Figure 2) and how to enter parameters:

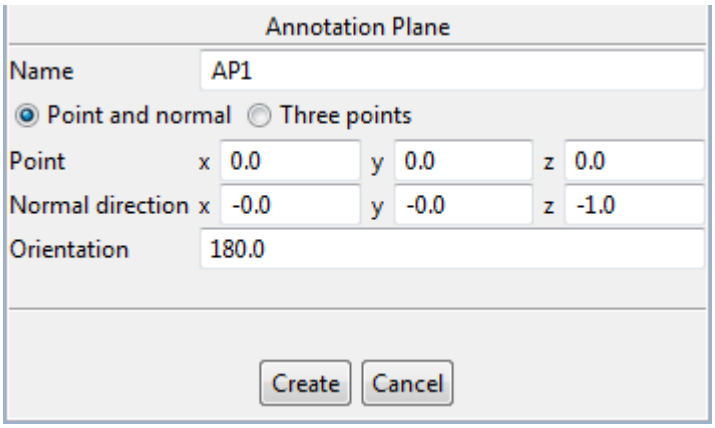

Figure 2. Annotation Plane wizard dialogue in GD&T view

- The **Name** field presents a default name of the annotation plane to be created (i.e., 'AP1', 'AP2' and etc.). Planes, that are used for annotations have unique names within all the **Style/Annotation** and **Geometric Models** of one design (**Geometry Details** view).
  To change a default value - enter a new or edit the currently entered name.
  Note. If a manually entered name already exist in the currently active Style/Annotation Model, the error message appears at the bottom section of the wizard dialogue.
- Select one of the radio buttons to define, how the annotation plane should be defined:
  - Point and normal, or
  - Three points.
- In case the **Point and normal** radio is selected:
  - Enter coordinates of a **Point** into **x**, **y** and **z** field.
  - Enter coordinates of a **Normal direction** into **x**, **y** and **z** fields.
  - Enter a value of the annotation plane orientation angle (degrees) into the **Orientation** field.
    The **Orientation** of the annotation plane defines an orientation of the text of annotation items, associated to this annotation plane. The annotation text reading direction is according X coordinate axis.
    To apply the changed orientation for annotation item, you have to update each annotation item (i.e. select annotation item in the **GD&T** view table and press the **Update** button in the annotation item wizard, which appears in the details area).
    Note. If a planar surface is selected, then the **Point**, **Normal direction** coordinates and **Orientation** angle value are added automatically.
- In case the **Three points** radio is selected:
  - Enter coordinates of a **First** point into the **x**, **y** and **z** fields.
  - Enter coordinates of a **Second** point into the **x**, **y** and **z** fields.
  - Enter coordinates of a **Third** point into the **x**, **y** and **z** fields.

Note. If a planar surface is selected, then the **First**, **Second** and **Third** point coordinates are added automatically.

- Press the **Create** button to create an annotation plane.

Update annotation plane data

To update parameters of an annotation plane:

- Select an annotation plane entry in the **GD&T** view table. The **Annotation Plane** wizard dialogue appears under the table in the details area (see Figure 3).
- Update needed annotation plane data:
  - Edit or enter a new annotation name into the Name field.
  - Edit or enter a new value of the annotation plane orientation angle (degrees) into the **Orientation** field.
  - The **Normal direction** fields are disabled for editing. To flip the direction of the annotation plane, press the **Flip** button.
- Press the **Update** button to confirm changes.

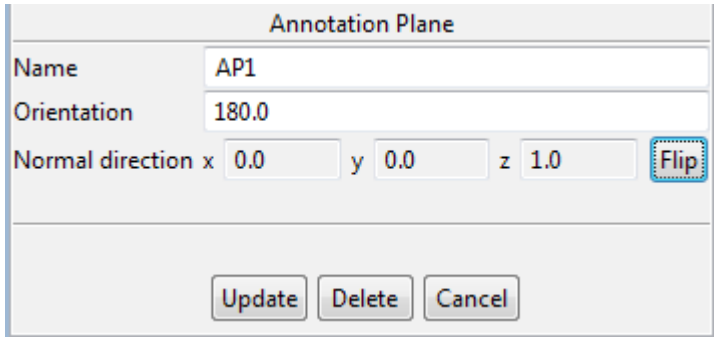

Figure 3. Annotation plane wizard dialogue to update annotation plane with pressed Flip button

Remove annotation plane

To remove an annotation plane:

- Press **Delete** button in the **Annotation Plane** wizard page.

Or

- Select an applicable annotation plane entry in the **GD&T** view table.
- Right click and select the **Delete** item in the appeared context sensitive menu.
  Note. In the case that there are annotations or callous on the annotation plane the user is asked on whether or not proceed with deletion

If some annotation item was situated on the recently removed annotation plane, the warning icon is added to the particular annotation item entry icon (see Figure 4). Fly over the icon with a mouse to see the warning text.

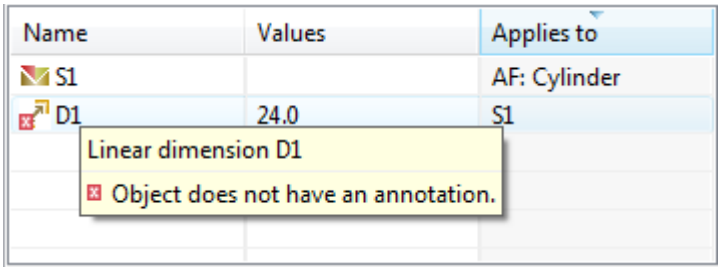

Figure 4. Dimension icon with the warning icon and tool tip with the warning text.

Active annotation plane

The currently active annotation plane entry is displayed by using bold text font in the **GD&T** view table.
To change the active annotation plane:

- Select an applicable annotation plane entry in the **GD&T** view table.
- Right click and select **Activate** item in the appeared context sensitive menu.

Move to active annotation plane

Annotation items, presented in the **GD&T** view table can be associated to various annotation planes. There is a possibility to move the annotation item to the active annotation plane:

- Select the annotation item in the **GD&T** view table.
- Right click and select the **Move to active annotation plane** item in the appeared context sensitive menu.

To top

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9.7.4. Datum Feature

A *Datum feature* is the actual surface component to establish a datum (points, lines, surfaces, planes and etc., from which a location or geometric relationships of other features may be established or related).

Create datum feature

Follow the routine to create a datum feature entry in the **GD&T** view table for selected shape aspect (for example *plane*):

- Right click and select **New Datum Feature** item in the appeared context sensitive menu (see Figure 1).
- The **Create Datum Feature** wizard dialogue appears in the **GD&T** view details area. How to manipulate with the wizard, see Create Datum Feature Wizard.

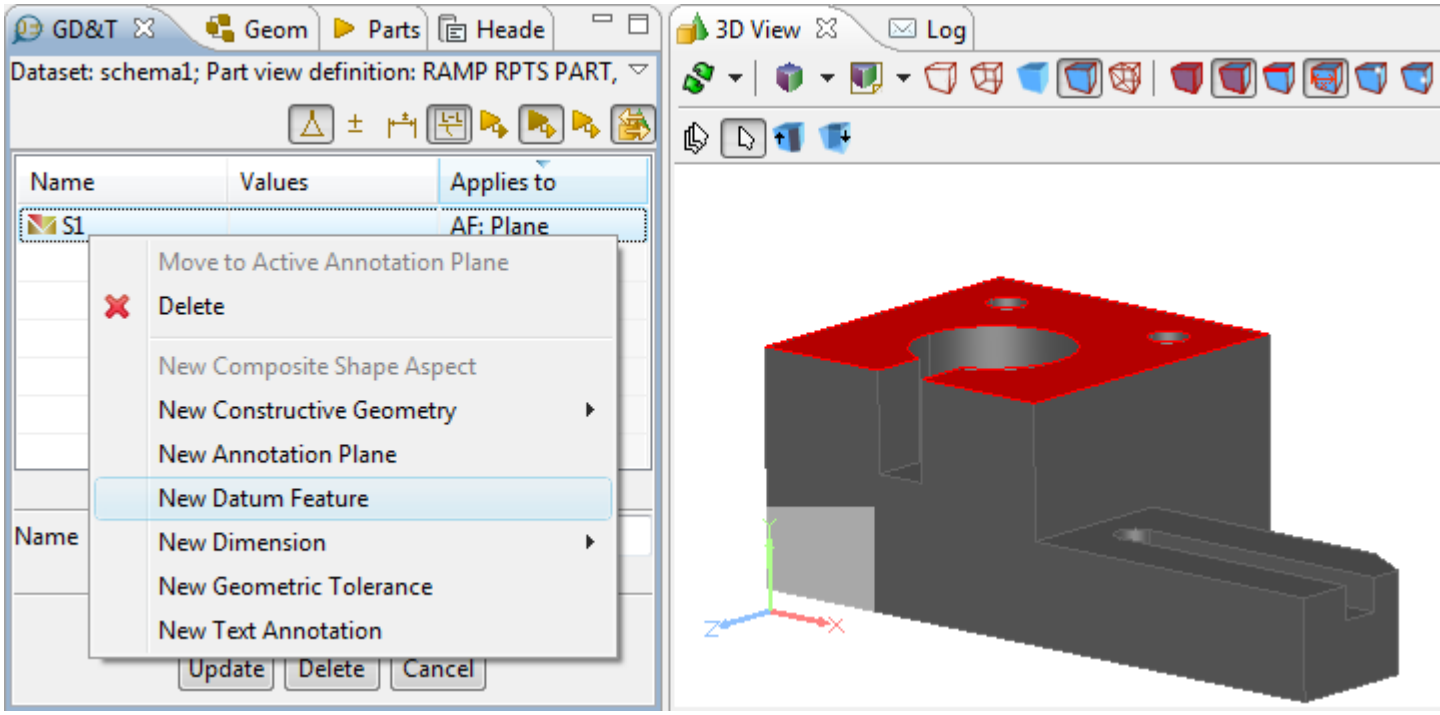

Figure 1. Shape aspect S1 in GD&T table to be defined as a datum feature A (wizard dialogue) and highlighted surface in 3D View

A new entry of datum feature appears in the **GD&T** view table (see Figure 2):

- the identification of a datum feature is presented in the **Name** column,
- the shape aspect the datum feature was made from (i.e. *S1*) is displayed in the **Values** column,
- the geometry element the created shape aspect is applied to (i.e. *plane*) is defined in the **Applies** to column.

Fly over the datum feature icon to view a tool tip, which contains additional information:

- The text *Datum feature* with the datum feature identification value, and
- The annotation plane the created datum feature is currently associated to.

Note. A new annotation plane (i.e. *AP1*) is created together with a new datum feature (i.e. *A*), in case no annotation plane were previously created.

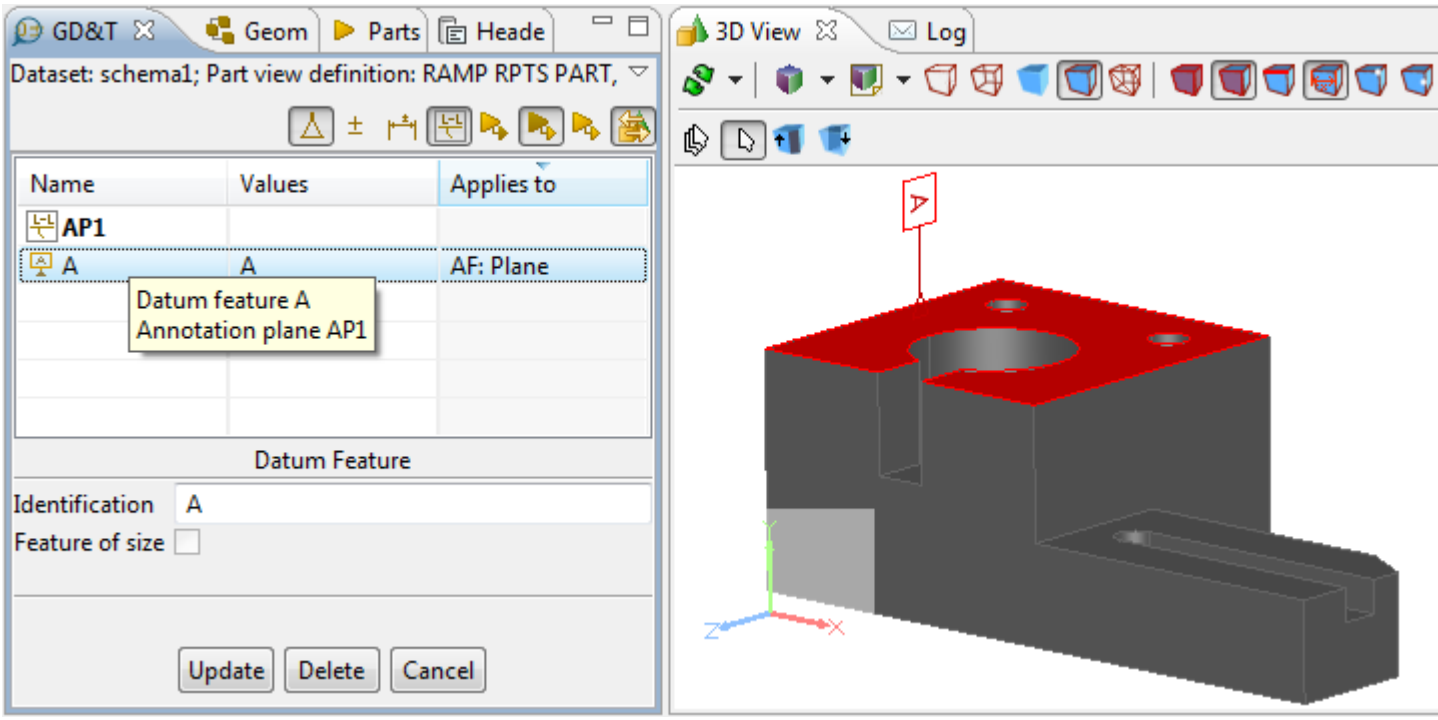

Figure 2. Created datum feature A in GD&T view (instead of shape aspect S1) and in 3D view (datum feature symbol)

Create Datum Feature Wizard

The **Create Datum Feature** wizard helps to create a datum feature for selected element (see Figure 3):

- The default identification label is automatically added into the **Identification** text field. To change the value - enter a new text, identifying the datum feature.
  In our example we have selected shape aspect *S1* (in the **GD&T** view table), which will be defined as a datum feature with the identification *A*.
- **Feature of size** check box. Select this check box if the datum feature is a feature of size. **Note.** The **Feature of size** check box is enabled for selection only if the symmetry element can be created for the selected geometric feature (i.e. for cylinder, cone, circle, two planes and etc.). In case there are selected a few geometric features and each of them are symmetrical, the dialogue still allows user to set a **Feature of size** flag.
- Press the **Create** button to create a new datum feature.

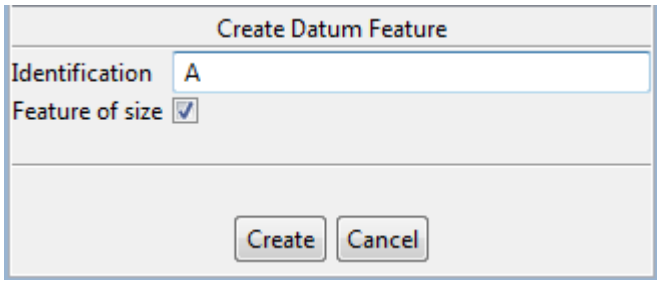

Figure 3. Create Datum Feature wizard page

Update datum feature

To update a datum feature:

- Select a datum feature entry in the **GD&T** view table. The **Datum Feature** wizard dialogue appears in the details area.
- Enter a new identifier into the **Identification** text field.

- The **Feature of size** check box is enabled for edition only if the selected feature can be a feature of size.
- Press the **Update** button to confirm your changes.

Delete datum feature

To remove a datum feature:

- Select the datum feature element in the **GD&T** view table.
- Press **Delete** button in the **GD&T** view details area.

Or

- Right click on the datum feature entry in the **GD&T** view table.
- Select **Delete** menu item in the appeared context sensitive menu.

To top

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9.7.5. Datum Target

A *Datum Target* is a specified point, line or area on a part that is used in establishing a datum.

The datum targets are displayed in the table list of the **GD&T** view if the filer button **Datums** is switched *On* (see Figure 1):

- **Name** column displays datum feature identification and number of the datum target. Fly by with a mouse over the **Name** cell to view a content of the tool tip, which contains an additional information about the selected datum target:
  - Full name of the table element - datum target, type of the target (in parenthesis) and a name of the target.
  - Annotation plane identification, which the selected target is associated to.
- **Values** column - value of a particular datum target (empty cell for Target Point, line length for Target Straight Line, diameter for Target Circle and Circular Curve and width x length for Target Rectangle).
- **Applies to** column - the datum identification, to which the target applies.

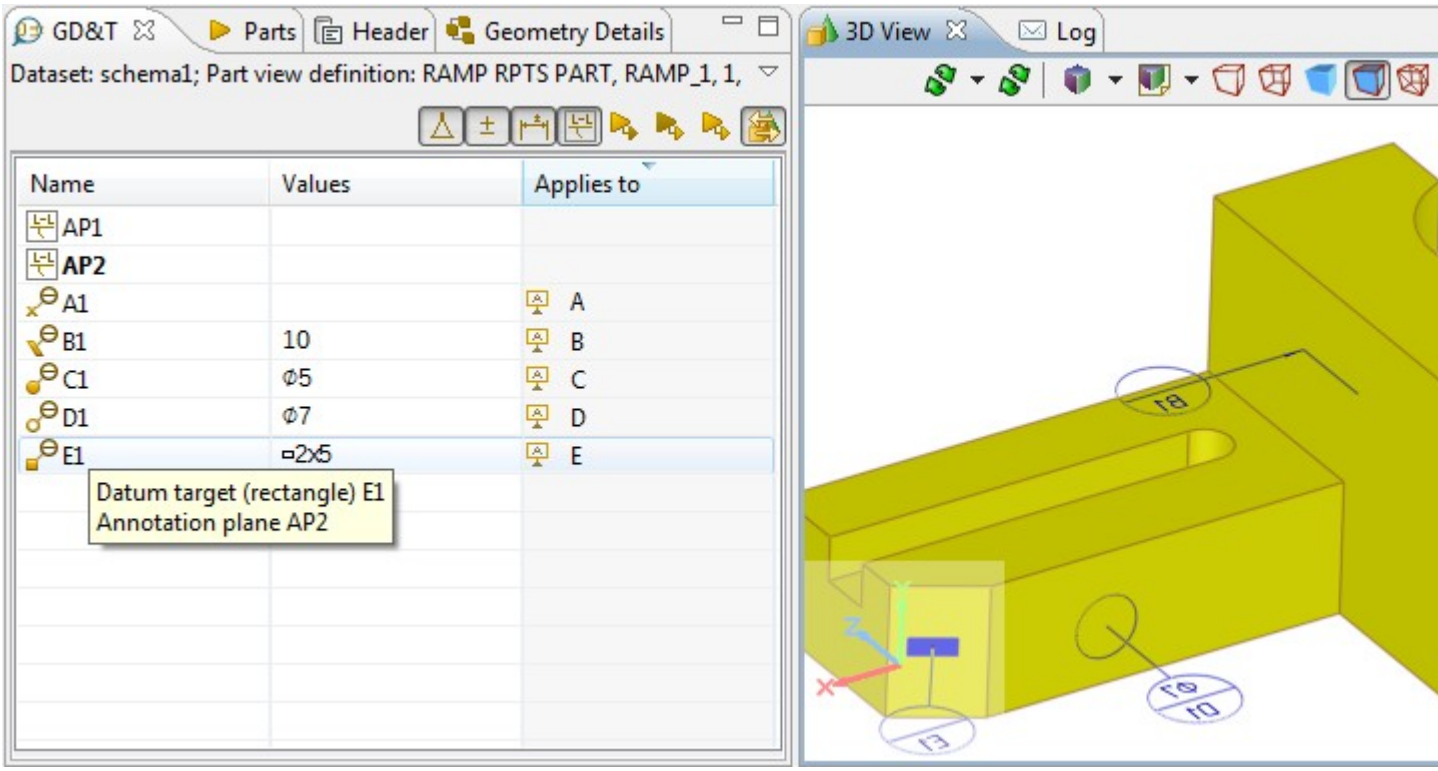

Figure 1. List of datum targets with tool tip displaying additional information about the datum target entry and datum target annotations in 3D View

Create Datum Target Wizards

In case some datum target creation was performed in the **3D View**, the particular datum target creation wizard is prompted in the **GD&T** view details area. The following wizards may appear:

- Create Target Point,
- Create Target Straight Line,
- Create Target Circle,
- Create Target Circular Curve,
- Create Target Rectangle.

For more information, how the created datum targets are presented on a geometry model in the **3D View**, see Datum Target.

Create Target Point Wizard

The **Create Target Point** wizard dialogue is displayed in case the **Datum Target -> Point Datum Target** menu item is selected in the **3D View** (see Figure 2):

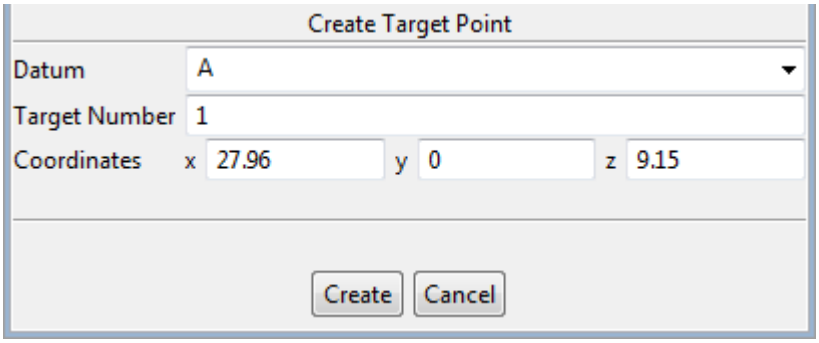

Figure 2. Create Target Point wizard dialogue

- The **Datum** field presents a default value for the datum the target point is associated to. To change a default value - enter a new datum identifier or select from the drop down list.
- The **Target Number** field present a default value of the target number. To change this value - enter a new number into the text field.
- The **Coordinates x**, **y**, **z** fields presents default coordinates of the selected point on the **3D View** geometry model. The precision for values, displayed in the field can be managed by changing decimal places value in the GD&T preferences (see Preferences -> GD&T).
  To change the default coordinate values - enter new coordinates into x, y, z fields.
- Press the **Create** button to create a datum target point.

The created target point appears in the table list of the **GD&T** (see Figure 1).

Create Target Straight Line Wizard

The **Create Target Straight line** wizard dialogue is displayed in case the **Datum Target -> Line Datum Target** menu item is selected (see Figure 3):

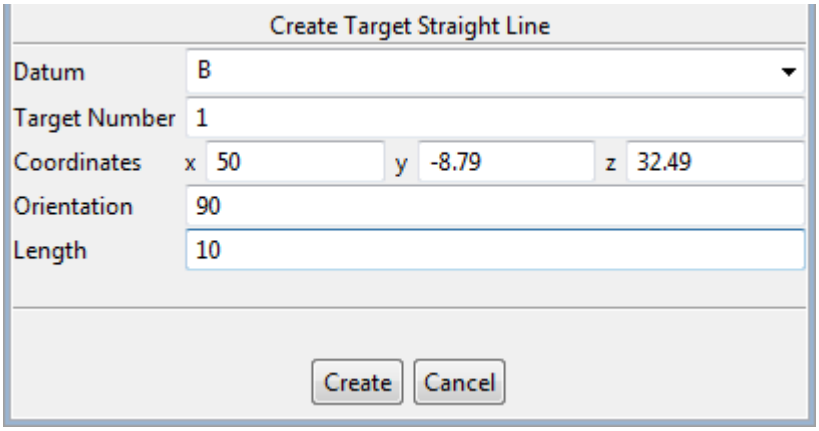

Figure 3. Create Target Straight Line wizard

- The **Datum** field presents a default value for the datum the target straight line is associated to. To change a default value - enter a new datum identifier or select from the drop down list.
- The **Target Number** field present a default value of the target number. To change this value - enter a new number into the text field.
- The **Coordinates x**, **y**, **z** fields presents default coordinates of the selected point (the start point of a target straight line) on the **3D View** geometry model. The precision for values, displayed in the field can be managed by changing decimal places value in the GD&T preferences (see Preferences -> GD&T).
  To change the default coordinate values - enter new coordinates into x, y, z fields.
- Enter the value of an angel (degrees) into the **Orientation** field.
- Enter the line length value into the **Length** field.
- Press the **Create** button to create a datum target straight line.

The created target straight line appears in the table list of the **GD&T** view (see Figure 1).

Create Target Circle Wizard

The **Create Target Circle** wizard dialogue is displayed in case the **Datum Target -> Circle Datum Target** menu item is selected (see Figure 4):

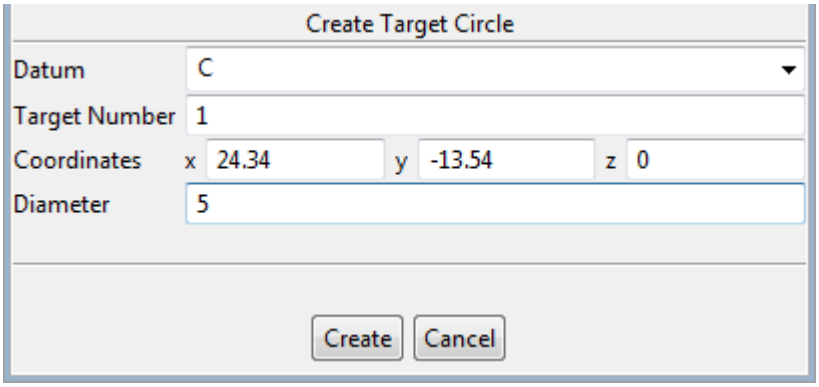

Figure 4. Create Target Circle wizard

- The **Datum** field presents a default value for the datum the target circle is associated to. To change a default value - enter a new datum identifier or select from the drop down list.
- The **Target Number** field present a default value of the target number. To change this value - enter a new number into the text field.
- The **Coordinates x**, **y**, **z** fields presents default coordinates of the selected point (the center point of a target circle) on the **3D View** geometry model. The precision for values, displayed in the field can be managed by changing decimal places value in the GD&T preferences (see Preferences -> GD&T).
  To change the default coordinate values - enter new coordinates into x, y, z fields.
- Enter the target circle diameter value into the **Diameter** text field.
- Press the **Create** button to create a datum target circle.

The created target circle appears in the table list of the **GD&T** view (see Figure 1).

Create Target Circular Curve Wizard

The **Create Target Circular Curve** wizard dialogue is displayed in case the **Datum Target -> Circular Curve Datum Target** menu item is selected. The wizard dialogue is identical to the Create Target Circle wizard dialogue.
The difference between created datum target is, that the Target Circle is the circle area and the Target Circular Curve - only the curve of a circle.

Create Target Rectangle Wizard

The **Create Target Rectangle** wizard dialogue is displayed in case the **Datum Target -> Rectangle Datum Target** menu item is selected (see Figure 5):

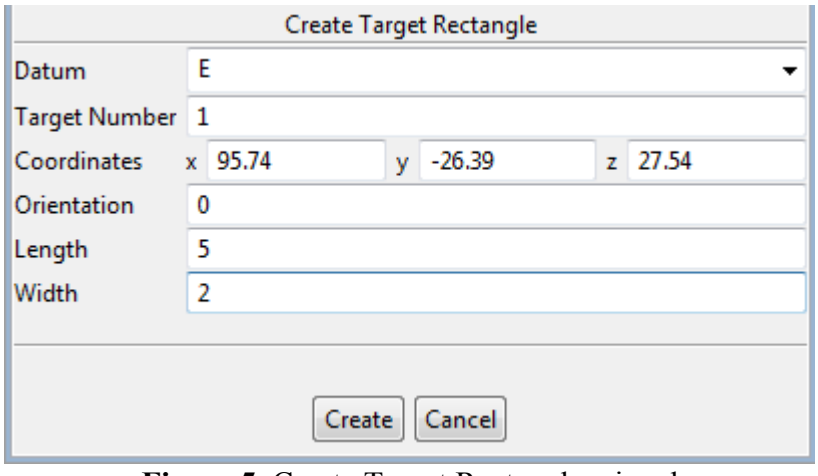

Figure 5. Create Target Rectangle wizard

- The **Datum** field presents a default value for the datum the target rectangle is associated to. To change a default value - enter a new datum identifier or select from the drop down list.
- The **Target Number** field present a default value of the target number. To change this value - enter a new number into the text field.
- The **Coordinates x**, **y**, **z** fields presents default coordinates of the selected point on the **3D View** geometry model. The precision for values, displayed in the field can be managed by changing decimal places value in the GD&T preferences (see Preferences -> GD&T).
  To change the default coordinate values - enter new coordinates into x, y, z fields.
- Enter the value of an angel (degrees) into the **Orientation** field.
- Enter the rectangle length value into the **Length** field.
- Enter the rectangle width value into the **Width** field.
- Press the **Create** button to create a datum target rectangle. The created target rectangle appears in the table list of the **GD&T** (see Figure 1).

Update datum target

To update parameters of a datum target:

- Select an applicable datum target entry in the **GD&T** view table. The particular wizard dialogue appears under the table in the details area.
- Update needed datum target data (see Create Datum Target Wizards for details) and press the **Update** button.

Remove datum target

To delete an incorrect or redundant datum target:

- Select a datum target in the **GD&T** view table.
- Press delete button in the **GD&T** view details area.

Or

- Right click and select **Delete** menu item in the appeared context sensitive menu.

To top

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9.7.6. Datum System

A *Datum system* is a group of two or more separate datums used as a combined reference for a toleranced feature. **Note.** A datum system in IDA-STEP may be comprised of a single datum.

How to start creation of a datum system, see Clause Create datum system.
After the creation, a new entry of the datum system appears in the **GD&T** view table (see Figure 1):

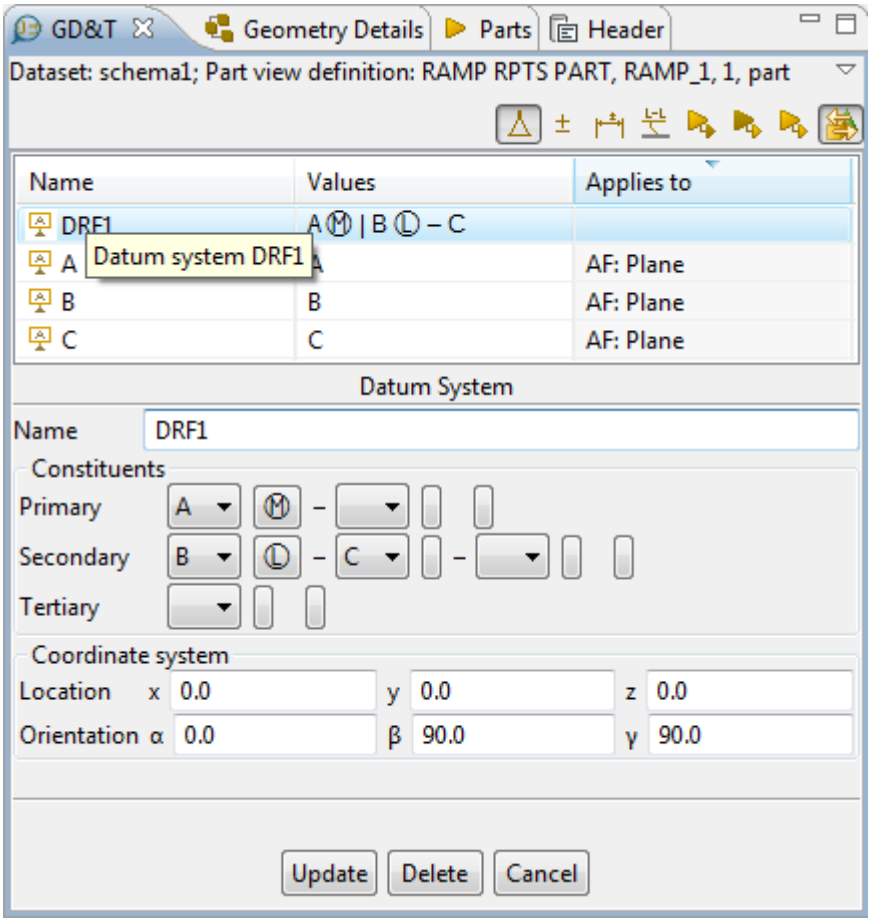


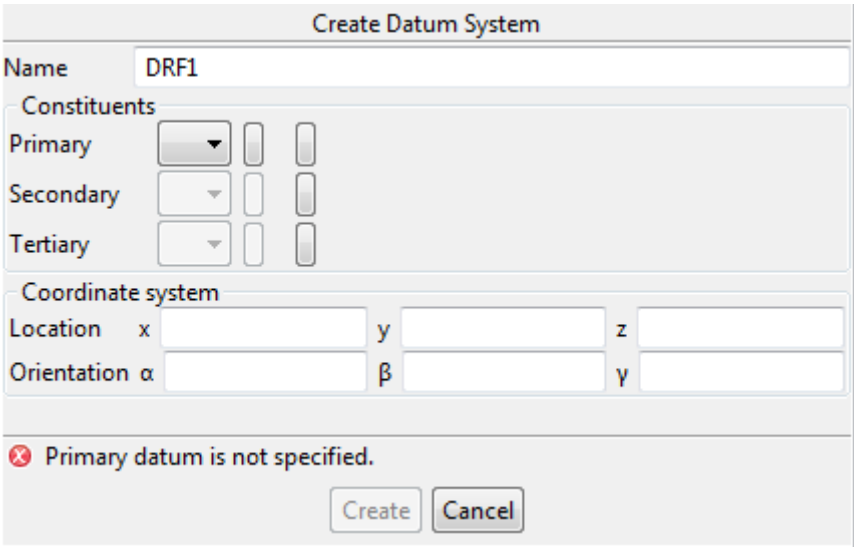
Figure 1. Created datum system (DRF1) in table and Datum System parameters in details area

- the identification of a datum system is presented in the **Name** column (i.e. 'DRF1'),
- the datum feature or set of datum features with applied modifiers (i.e. AⓂ|BⓁ - C), which are included in the created datum system, is displayed in the **Values** column.

Fly over the datum system icon to view a tool tip, which contains additional information (*Datum system* with the datum system identification value). If a datum system entry is selected in the **GD&T** view table, the **Datum System** window with the datum system parameters is presented in the details area.

Create Datum System Wizard

The **Create Datum System** wizard guides in the creation of a new datum system (see Figure 2):



a) b)

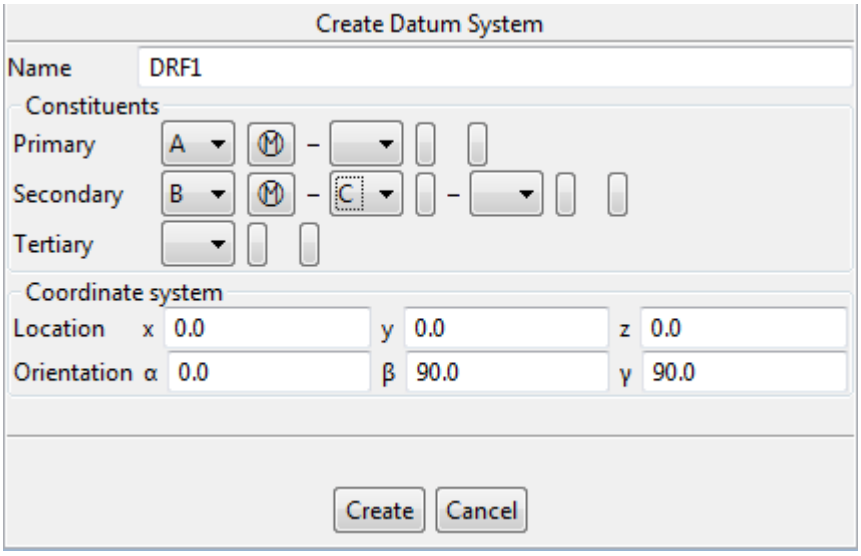


Figure 2. Create Datum System wizard page: a) datums not specified b) with specified datums

- The default identification label of the datum system to be created (i.e. '*DRF1*') is automatically added into the **Name** text field. To change the value - enter a new text, identifying the datum system.
- Specify constituents of the datum system by selecting datum features in the **Primary**, **Secondary** and/or **Tertiary** combo boxes:
    - Initially you can see only one combo box next to the **Primary**, **Secondary** and **Tertiary** labels (see Figure 2 a). After the value is selected in the first combo box, another one appears next the first one (see Figure 2 b). You can specify datums as many as needed in each constituent line.
    - You can specify a modifier for each datum feature by pressing on the small square next to the datum feature combo box. The **Modifiers** dialogue window appears on the screen (see Figure 3). The dialogue window contains two sections **Modifiers supported by ISO** and **Modifiers not supported by ISO**. You can collapse or expand those sections by using double arrows next to the section label or just by clicking on the label. To select an applicable modifiers, set a check mark in front of a needed modifier. **Note.** In some cases, i.e. *Modifiers not supported by ISO*, *Distance* modifier, additionally the number value should be specified in the text field.
      Press **OK** button to confirm selections and close the window. Selected modifiers appear next to the particular datum feature field.

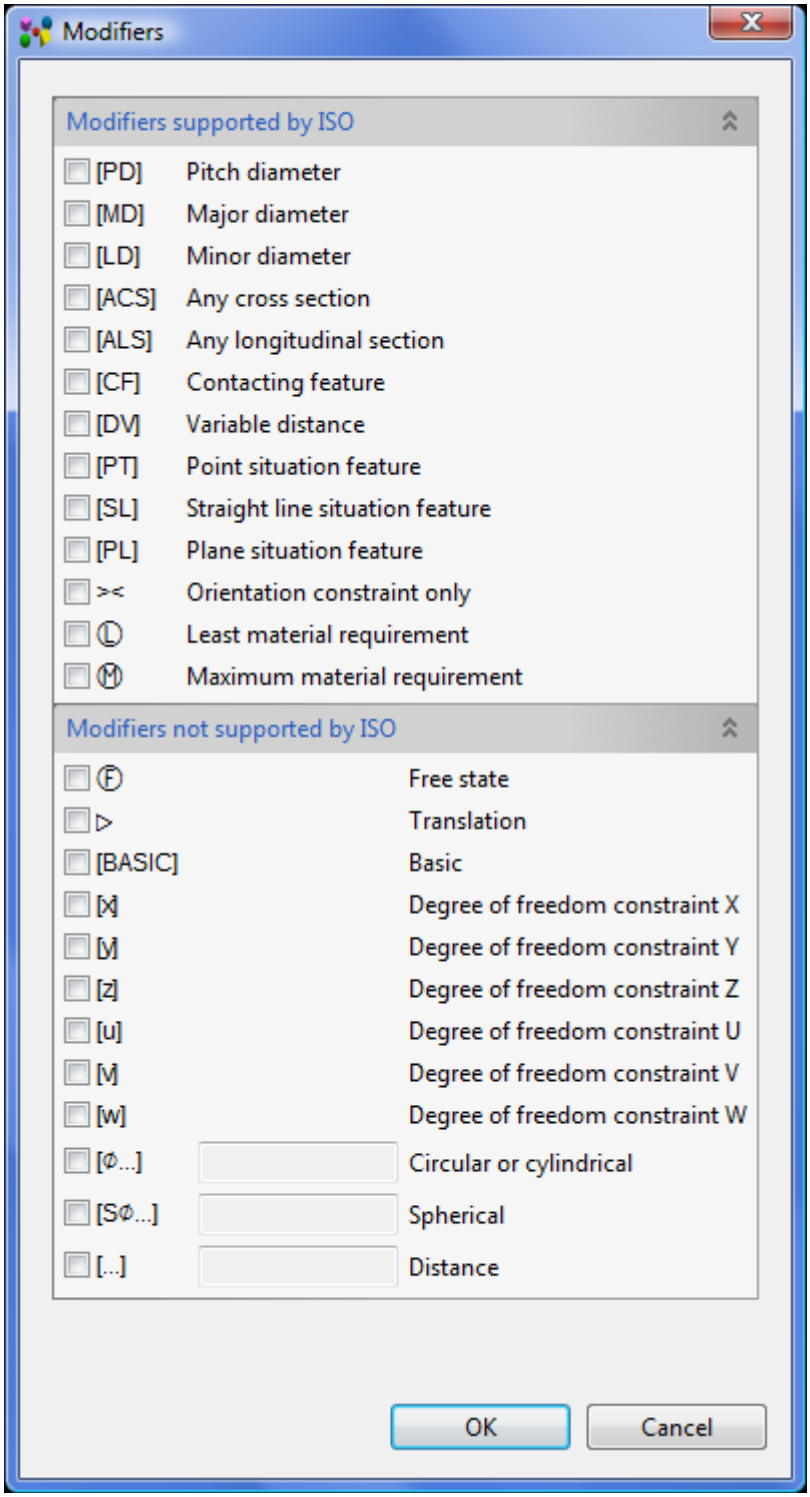


Figure 3. Modifiers dialogue window

- The **Coordinate system** values are entered automatically as soon as constituents of a datum system are selected. To change the coordinate system parameters:
    - Enter new location **x**, **y** and **z** values into particular fields next to the **Location** label.
    - Enter new orientation values into **α**, **β** and **γ** fields next to the **Orientation** label.
- Press **Create** button to create a datum system with specified parameters.

Update datum system

To update a datum feature:

- Select a datum system entry in the **GD&T** view table. The **Datum System** dialogue appears in

the details area (see Figure 1).
- Enter a new parameters for a datum system (see Create Datum System Wizard for details).
- Press **Update** button to confirm your changes.

Delete datum system

To remove a datum system:

- Select the datum system element in the **GD&T** view table.
- Press **Delete** button in the **GD&T** view details area.

Or

- Right click on the datum system entry in the **GD&T** view table.
- Select **Delete** menu item in the appeared context sensitive menu.

Note. If the selected datum system is used by any GD&T element, the delete action is not available (**Delete** button and **Delete** menu item are disabled).

To top

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9.7.7. Dimensions

IDA-STEP presents a possibility to define dimensions of selected shape aspects in the **GD&T** view:

- **Size dimension** - a specified value of a diameter (spherical diameter), radius, length directly related to the size of an object;
- **Distance dimension** - measure the distance between two points, measured in the default units;
- **Angular dimensions** - measure the angle between two lines, planes, cones, surfaces sharing a common end point called the vertex.

Create Dimension

To create a particular dimension (for example for a shape aspect of type *Circle*):

- Select a shape aspect of type *Cylinder* in the **GD&T** view table or a particular geometry element in the **3D View** geometry model. **Note.** In some cases, two shape aspects can be selected for creating a dimension (to measure a distance between two shape aspects)
- Right click and select **New Dimension -> Angular/Distance/Size** menu item in the appeared context sensitive menu (see Figure 1). **Note.** In case some dimensioning for the selected shape aspect is not available, the particular sub menu item is disabled.

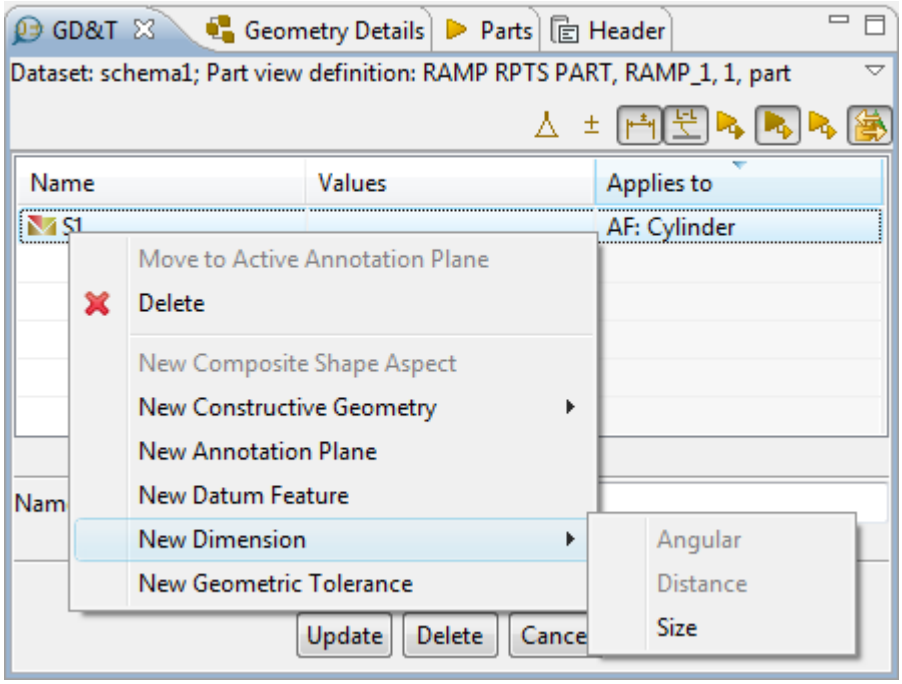

Figure 1. Selected shape aspect of type Circle, context sensitive menu New Dimension -> Angular/Distance/Size

After the menu item (**Angular**, **Distance**, or **Size**) is selected, the particular wizard dialogue, named according the selected dimension type, appears under the table (see Angular/Distance/Size Dimension Wizard). In our example only *Size dimension* is available for the selected shape aspect *Cylinder*.

The created dimension object appears in the table. The name of shape aspect, for which the dimension is created is defined in the **Applies to** column. In our example the created dimension is size dimension, cylinder diameter with plus minus toleranced value, applies to *S1* shape aspect (see Figure 2).
In case the dimension object is selected, the wizard is displayed under the table, where the value and other parameters can be updated for the selected dimension by pressing the **Update** button.

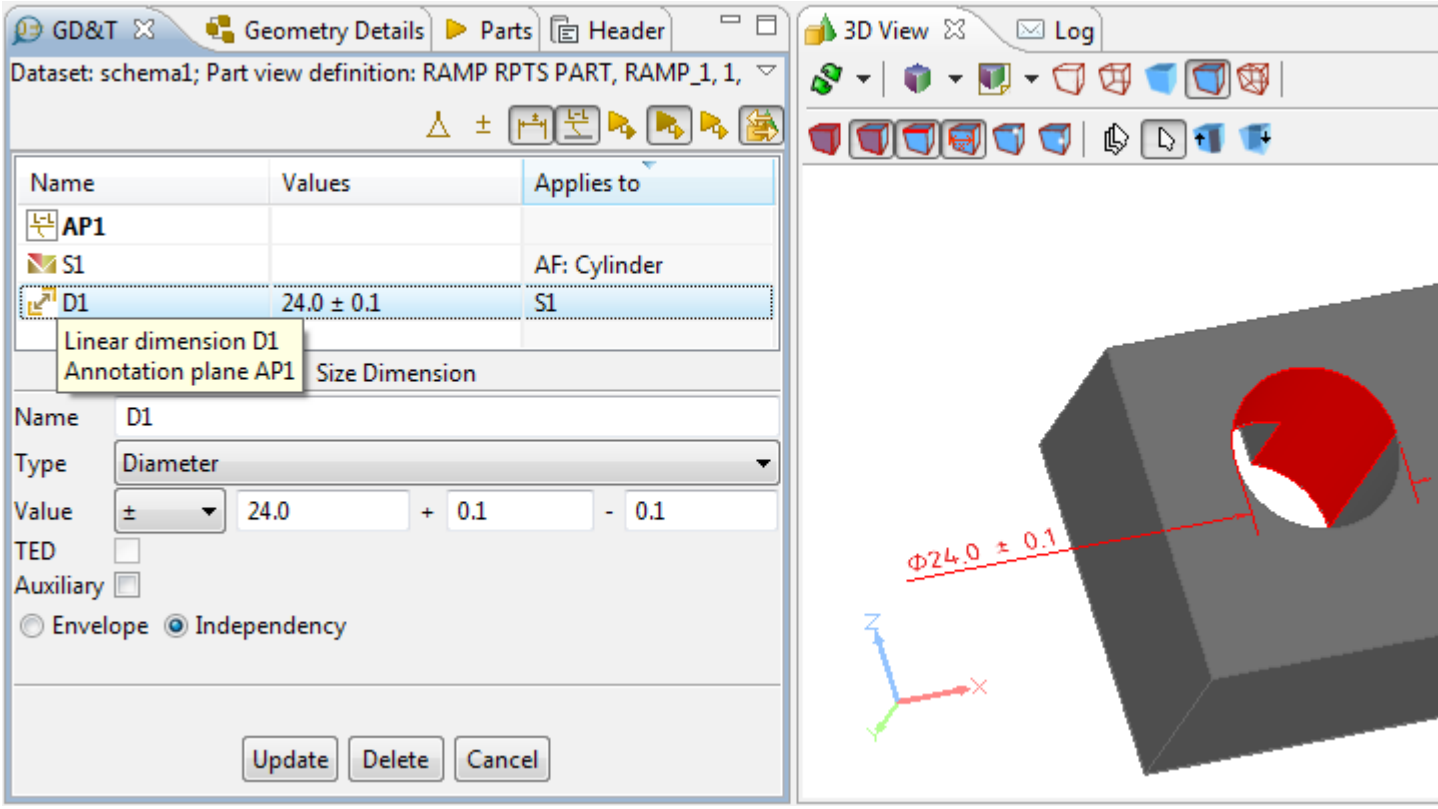

Figure 2. Created size dimension is highlighted in GD&T view table and presented in 3D View (highlighted together with the related cylinder surface)

Note. To view the list of available shape aspects and dimensions, the **Shape aspects** and **Dimensions** filter buttons must be switched On (above the table).

Angular/Distance/Size Dimension Wizard

The **Angular/Distance/Size Dimension Wizard** guides a user, how to create a particular dimension for the selected shape aspect:

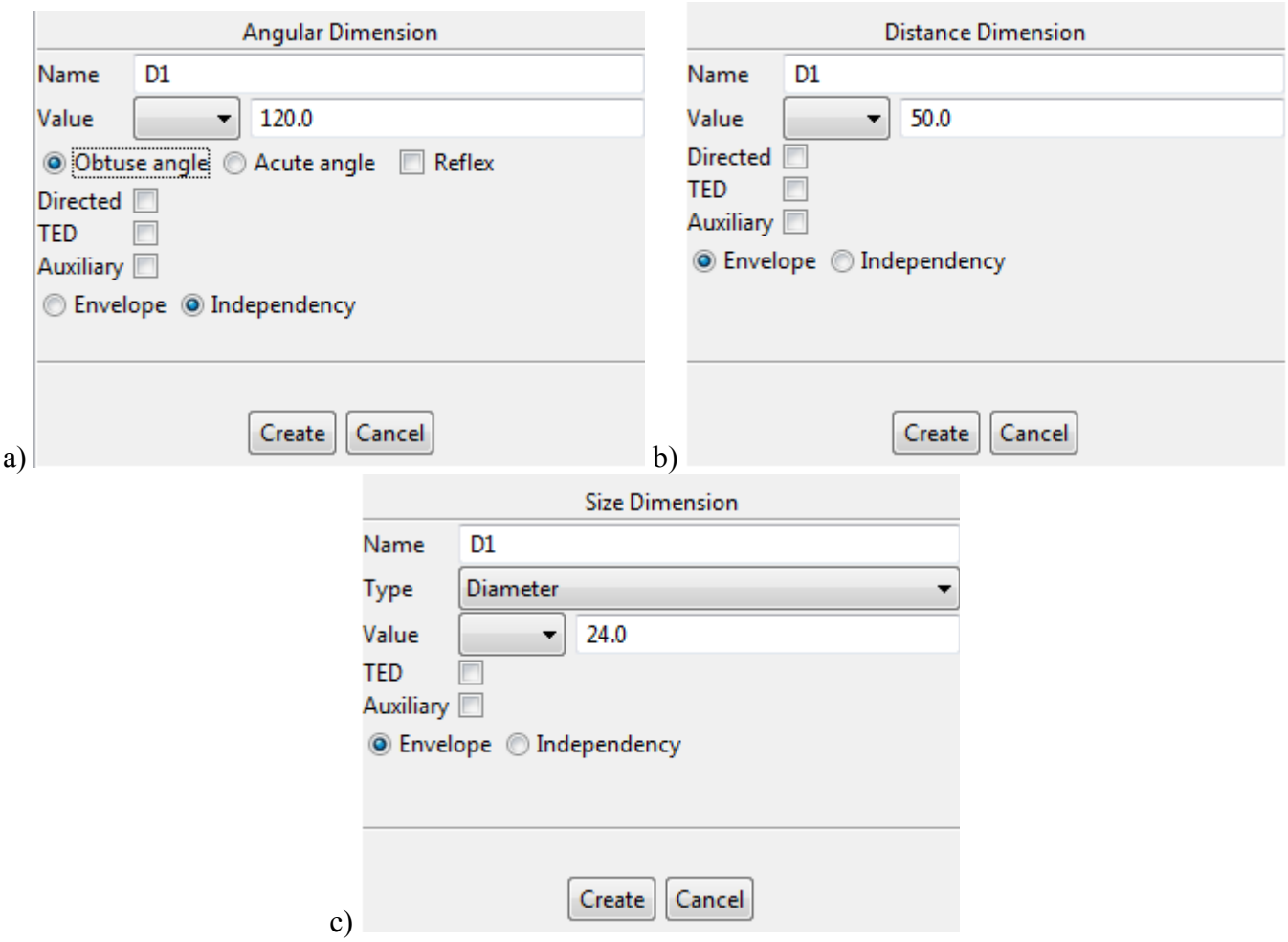

Figure 3. a) Angular Dimension wizard, b) Distance Dimension wizard, c) Size Dimension wizard

- **Name** text field presents a name of the dimension, which appears in the **Name** table column after the dimension is created. By default the name is entered automatically D<number>. The default value can be edited or a new text can be added to the **Name** field.
- **Type** list presents the types of size dimension, available for the selected shape aspect. For example, in case the *Circle* shape aspect is created, the **Type** list contains three predefined values: **diameter** (to measure diameter of the selected circle), **radius** (to measure radius of the selected circle) and **length** (to measure arc length of the selected circle). In case of the *Sphere*, the **Type** list contains predefined values: **Spherical diameter** and **Radius**. In other cases only one type value may appear in the list.
  Note. In case distance dimension or the angular dimension, the selection of dimension type is not available as the **Type** combo box does not appear at all (see Figure 3 a and b).
  Select an applicable value from the list to define the type of dimension. **Note.** In our example the **Diameter** type is selected (Size Dimension wizard dialogue).
- **Value** indicates the amount a measured distance can vary (dimension with tolerances). **Note.** Value should not be negative (there are some exceptions for angular dimension). The error message appears in the wizard window bottom section, informing about wrong value.
  Select the type of the value in the drop down list. Available predefined values are:
  - Select **Limit** to set limits of the dimension value (see Figure 4):

- Enter the lowest value into the **Low** text field. **Note.** In case of an angular dimension, the **Low** value must not be less than -90 degrees.
- Enter the highest value into the **High** text field. **Note.** In case of an angular dimension, the **High** value must not be more than 450 degrees.

  Note. In case the dimension in one direction only (only *Low* or *High* value is specified for the dimension), the created dimension value is followed by text string *min./max.* in the **GD&T** view table **3D View** (for example '24 min.' or '24.01 max.').

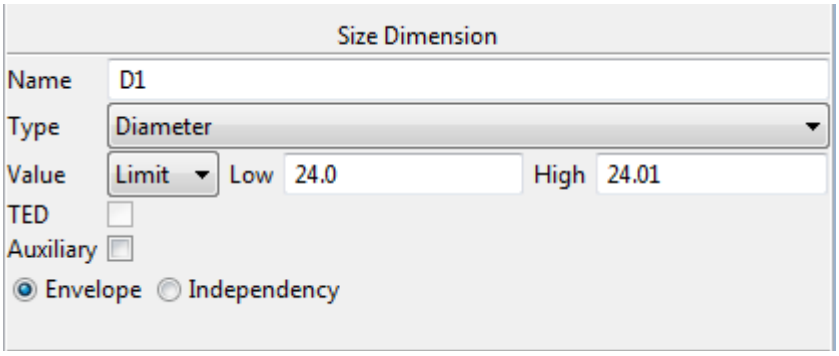

Figure 4. Size Dimension wizard page with selected Limit value in Value drop down list

- Select ± **(plus minus)** to set the exact value and then the bounds of the dimension (see Figure 5):
  - Enter the exact value into the **Value** text field.
  - Enter the plus value into the + text field.
  - Enter the minus value into the - text field.

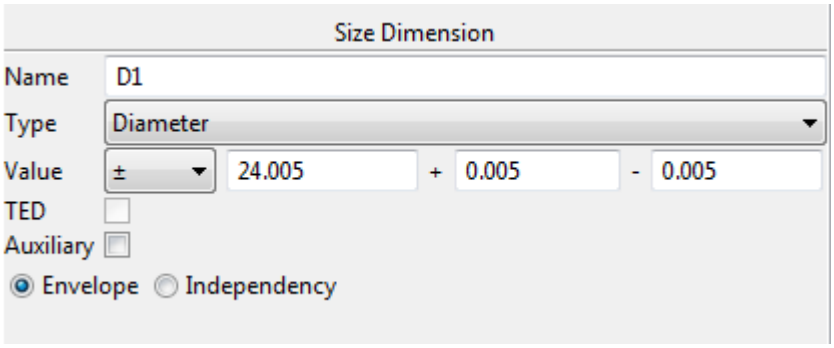

Figure 5. Size Dimension wizard with selected ± value in the Value drop down list

- Select **Class** to set the exact value and tolerance class (defined in ISO 286) for the dimension (see Figure 6):

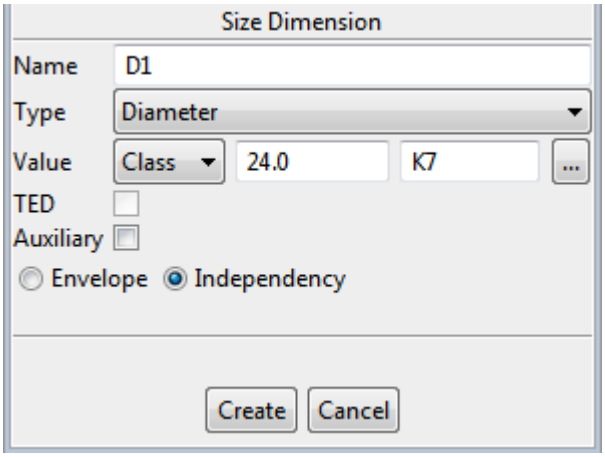

Figure 6. Size dimension wizard with selected Class value in the Value drop down list

- Enter the exact value into the **Value** text field.
- Enter the tolerance class code into the next text field (i.e "K7").

Or

- Press the ... icon button next to the text field. The **Tolerance Class** dialogue window appears on the screen (see Figure 7):
  - Select the fitting type for the tolerance class **Shaft** or **Hole** by switching *On* particular radio button:
    - **Shaft** - for outer features of size;
    - **Hole** - for inner features of size.
  - Select deviation value in the **Deviations** table.
    Note. *Deviations* are populated according to the selected fitting type. When user selects an appropriate deviation the *grades* are filled in and user can select it.
  - Select the grade value in the **Grades** table.
  - Press **OK** button to confirm selections and close the window.

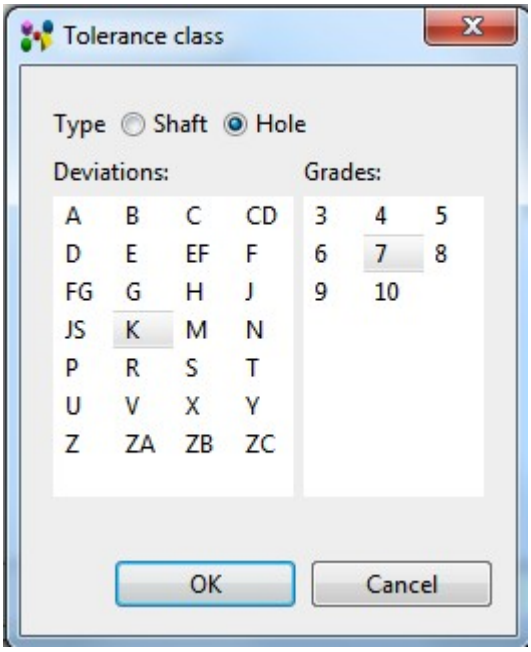

Figure 7. Tolerance Class dialogue window to select tolerance class for dimension

  - Select empty line - to set the exact value (see Figure 3) and create a dimension without tolerance . Enter the value of the dimension into the **Value** text field.
- In case the angular dimension, the radio group (**Obtuse angle**, **Acute angle** radio buttons and **Reflex** check box) to select, which angle should be measured (obtuse, acute or reflex) appears under the **Value**. **Note.** *Obtuse angle* plus *Acute angle* equals 180 degrees (see Figure 8a). *Reflex angle* is more than 180, but less than 360 degrees (see Figure 8b - different reflex angles of the same intersecting planes).
  - **Obtuse angle** - to measure a bigger angle between two intersecting planes/lines;
  - **Acute angle** - to measure a smaller angle between two intersecting planes/lines;

Note. It is not possible to select obtuse/acute for angles that are periods of 90 degrees (including 0).

  - **Obtuse angle + Reflex** - to measure a reflex angle of the obtuse angle between two intersecting planes/lines;
  - **Acute angle + Reflex** - to measure a reflex angle of the acute angle between two

intersecting planes/lines.

Note. It is not possible to select reflex for angles that are periods of 180 degrees (including 0).

Note. In case the conical face, the Angular Dimension wizard does not contain the above radio group.

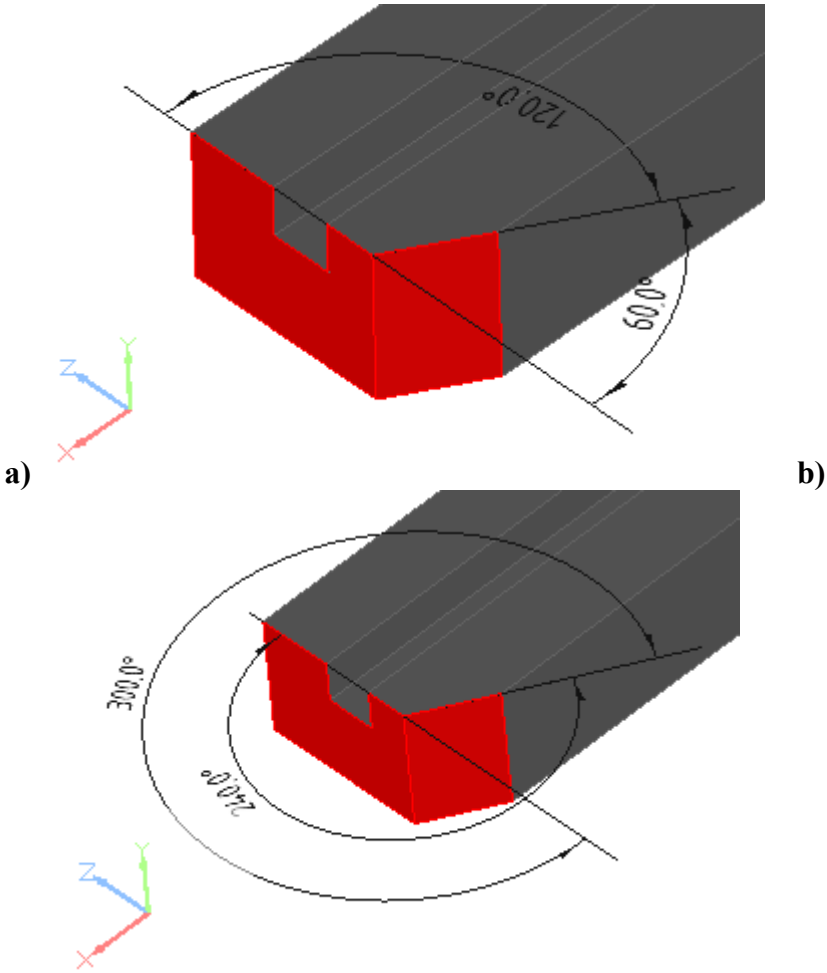

Figure 8. a) Acute ('60 °') and obtuse ('120 °') angle for same intersecting planes; **b)** obtuse + reflex ('240 °') and acute + reflex ('300 °') for same intersecting planes

- **Directed** option appears only in case two shape aspects are selected. Select this option to emphasize that the distance must be measured exactly from the first selected shape aspect to the second one.
- **TED** (theoretically exact dimensions) option. In ASME they are called basic. For the presentation it is drawn a box around the value. TED is theoretically exact and it can not have a tolerance. Select this option to define the dimension is theoretically exact dimensions.
- Select the **Auxiliary** (optional) option to define the dimension is auxiliary.
- Radio buttons group (select one of the radio buttons):
  - **Envelope** - for ASME the Envelope requirement is default. In ISO it is indicated by Ⓔ.
  - **Independency** - this is a default in ISO. In ASME it is indicated by Ⓘ.
- After all required data are entered, press the **Create** button.
  Note. If some required data are not specified, the error message is displayed in the wizard page and the **Create** button is disabled until the mandatory data are entered.

When creating another dimension, the previous choice is pre-set for the following parameters:

- size type (radius, diameter etc.);
- value type (+/-, limit etc.);

- directed flag;
- TED flag;
- auxiliary flag;
- envelope flag.

  Note. If some remembered value does not apply to a new dimension (e.g. a size type to a distance dimension), then it is simply ignored.

See 3D Viewer > Management of GD&T Elements -> Dimensions for dimension presentation in **3D View**.

Update Dimension

To update a dimension:

- Select a dimension entry in the **GD&T** view table. The **Angular/Size Dimension** wizard dialogue appears in the details area.
- Enter new or edit parameters for the selected dimension.
- Press the **Update** button to confirm your changes.

Note. If the selected dimension is not supported by IDA-STEP, it is only possible to change a **Name** of such dimension.

Delete Dimension

To remove a dimension:

- Select the dimension element in the **GD&T** view table.
- Press **Delete** button in the **GD&T** view details area.

Or

- Right click on the dimension entry in the **GD&T** view table.
- Select **Delete** menu item in the appeared context sensitive menu.

To top

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9.7.8. Geometric Tolerances

Geometric tolerances show acceptable variations (maximum and minimum limits) of size, form, profile, orientation, location, and runout of a feature.

In the **GD&T** view geometric tolerances are displayed in the table list, when the **Tolerances** filter button is switched *On* (see Figure 1):

- **Name** column contains a symbol that represents the geometric characteristic (type of tolerance) to which a tolerance is being applied (in our example there are flatness and position tolerance symbols). Fly over the **Name** cell to view the content of the tool tip:
  - Type of the geometric tolerance with a name,

    - Annotation plane, the selected geometric tolerance is currently associated to.
- **Values** column contains information of the tolerance, for example diameter symbol, tolerance value, material conditions of the tolerance, projection with value, datum reference, material conditions of datums and etc.
- **Applies** to column contains the shape aspect, to which the tolerance is applied.

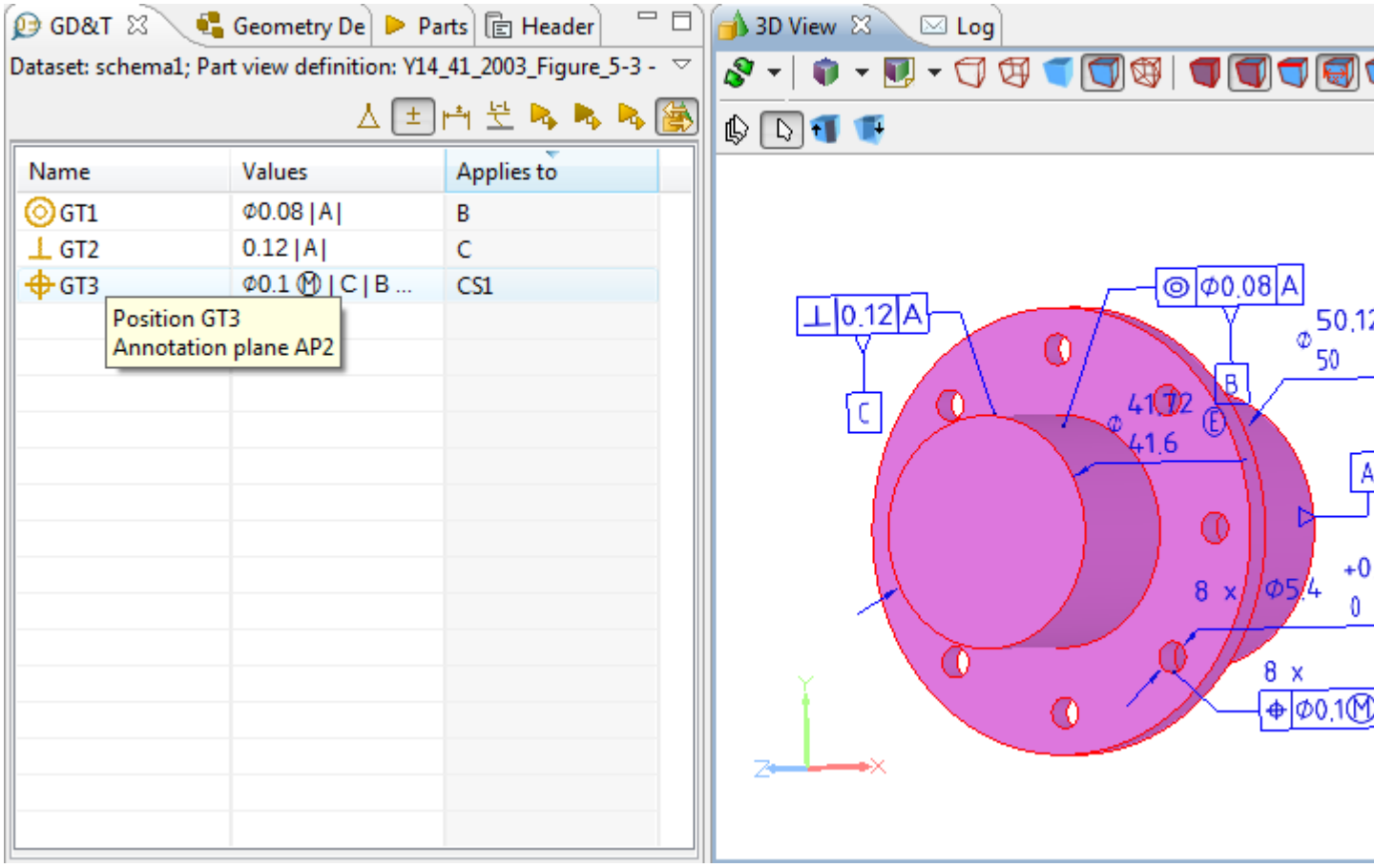

Figure1. GD&T view table with filtered list of tolerances and 3D view with tolerance annotation

Create new geometric tolerance

To create a new geometric tolerance:

- Select a shape aspect in the **GD&T** view table. **Note.** To view the list of shape aspects - turn on the **Shape aspects** button.
- Right click and select **New Geometric Tolerance** item in the appeared context sensitive menu.
- The **Create Geometric Tolerance** wizard appears under the table (see Figure 2). How to fill the wizard page with data, see Create Geometric Tolerance Wizard.

The created geometric tolerance entry appears in the **GD&T** view table (see an example in Figure 1).

Create Geometric Tolerance Wizard

The **Create Geometric Tolerance** wizard (see Figure 2) helps to create a geometric tolerance for selected shape aspect/surface component:

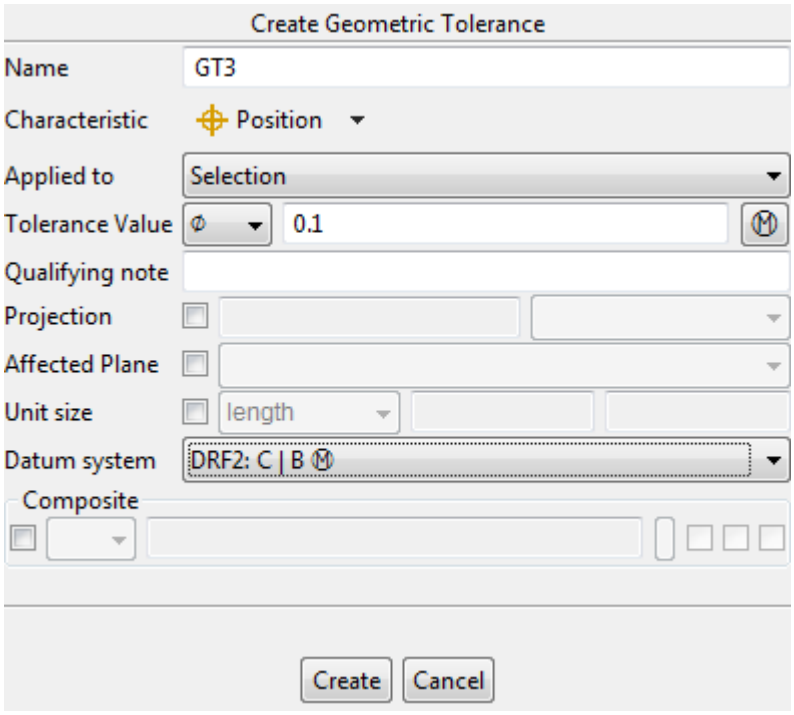

Figure 2. Create Geometric Tolerance wizard with specified parameters

- Enter a text, by which a geometric tolerance is known, into the **Name** text field. **Note.** By default this field is filled automatically with the text '*GT<number>*'. You can edit or enter a new name of the geometric tolerance to be created.
- In the **Characteristic** drop down list select the type of geometric tolerance to be created (see Figure 3). **Note.** In case an incorrect or not available type of tolerance is selected, the error message appears in the bottom section of the wizard.

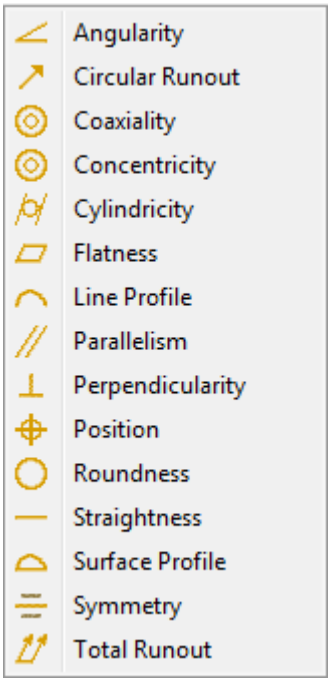

Figure 3. List of geometric tolerance characteristics

- In the **Applied to** drop down list select the element to which the tolerance to be applied. Note. IDA-STEP suggests an element according the selected shape aspect (for example, if the selected shape aspect is of type *plane*, the **Applied to** field will be filled with value *Selection*). Available predefined values are:
  - **Selection** - currently selected shape aspect;
  - **Individual Median Feature(s)** - if multiple features are selected, use median features from each selected geometry individually;

- **Single Median Feature** - the median feature to construct is between two selected geometrical features (e.g. two parallel planes, two concentric cylinders or in general two equidistant surfaces, or two equidistant curves or a wedge);
- **All around** - all surfaces connected end-to-end 360 degrees around the part;
- **All over** - surfaces all over the part.

- The **Tolerance Value** line is split up into three parts:
  - Combo box - select a type of the tolerance zone:
    - none - the tolerance value used for linear dimensions;
    - ø - the tolerance zone is circular or cylindrical;
    - Sø- the tolerance zone is spherical.
  - Text filed - enter the tolerance value. This parameter is mandatory. If no value is entered, the error message appears on the bottom section of the wizard (i.e., 'Tolerance value is not specified').
  - Button next to the text field - select modifier (boundaries of a perfect form). Press the button. The **Modifiers** dialogue window appears on the screen (see Figure 4). The dialogue window contains two sections **Modifiers supported by ISO** and **Modifiers not supported by ISO**. You can collapse or expand those sections by using double arrows next to the section label or just by clicking on the label. To select an applicable modifiers, set a check mark in front of a needed modifier. **Note.** In some cases, i.e. *Modifiers supported by ISO*, *Unequally disposed* modifier, additionally the value should be specified in the text field. The *Unequally disposed* modifier can be applied to the profile tolerance (both - surface and line) only.
    Press **OK** button to confirm selections and close the window. Selected modifiers appear next to the tolerance value field.

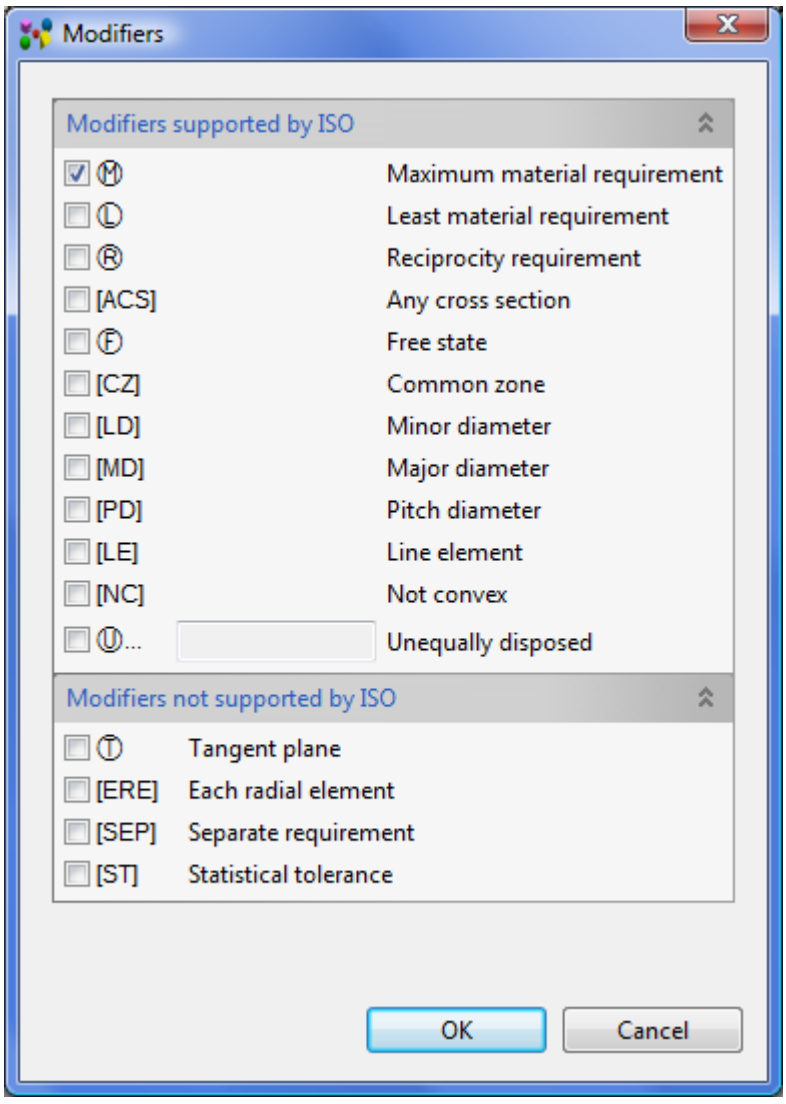

Figure 4. Modifiers dialogue window with selected Maximum material requirement modifier

- The **Qualifying note** text field is the place to enter additional text for a geometric tolerance. Enter an applicable text into the text field. This text appears in the geometric tolerance annotation below the datum reference frame (in **3D View**, see Figure 5). In case the composite geometric tolerance is created, the entered text appears on the right side of the particular tolerance.

Figure 5. Geometric tolerance annotation with qualification note ('*8 holes*')

- The **Projection** option is to define a projection tolerance zone. The projection option is enabled for positional, angularity, parallelism, perpendicularity, concentricity and symmetry tolerances.
    - Select the **Projection** check mark.
    - Enter the projection value into the text field.
    - Select the projection end feature in the combo box drop down list.
- The **Affected Plane** option - select this option to define the plane to apply the tolerance value. Press the arrow button next to the text field and select a particular plane in the drop down list.
- The **Unit size** option - select this option to specify the size of a segment. Select an area of the segment in the combo box drop down list. Available values:
    - **length** - in case of the length area, enter a number, which represents a length of the unit.
    - **circular** - in case the circular area, enter diameter value into the number field. A round unit basis area is defined by a diameter symbol followed by the diameter value in the **GD&T** view table and in the **3D View**.
    - **square** - in case the square area, enter a length of the side of the square into the number field. A square unit basis area is defined by a square symbol followed by the length of the square side value (see an example in Figure 6).
    - **rectangular** - in case the rectangular area, enter length and width values into the number fields next to the combo box.

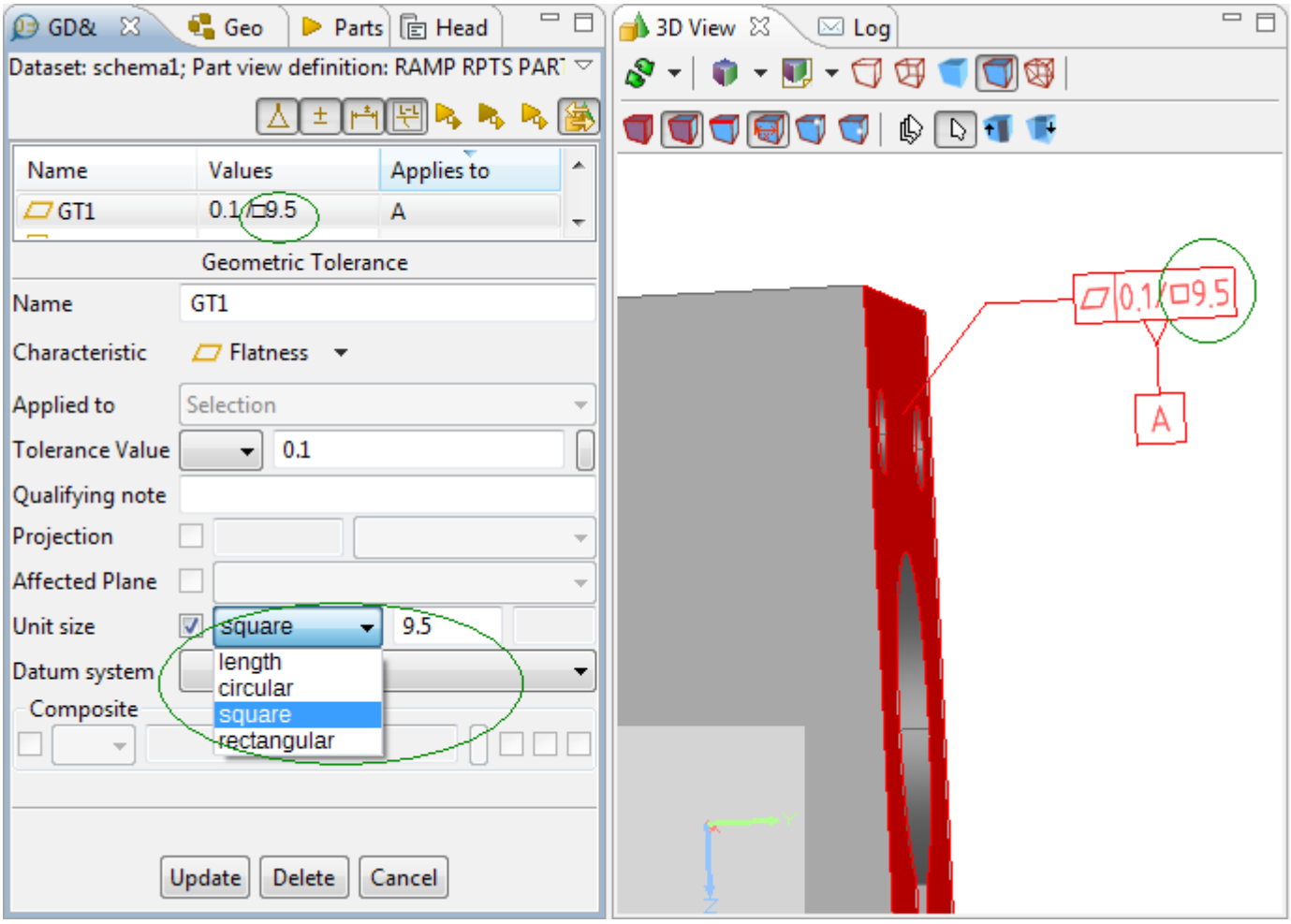

Figure 6. GD&T view (left): created geometric tolerance with defined square unit area; 3D View (right): presentation

- The **Datum system** layout - select the datum system for a geometric tolerance in the combo box drop down list. Note. The datum system must exist in data or be previously created (how to create datum system refer to clause Create datum system).
- The **Composite** layout - select this option to create a combination of several tolerance frames. This option is available for line and surface profile, position tolerances. Define tolerance parameters for each line of tolerance frames (tolerance zone, value and material condition symbol).
- Press the **Create** button to confirm selection and create a geometric tolerance.

See also 3D View -> Management of GD&T Elements -> Geometric Tolerances (in case **3D View** is available).

To top

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9.7.9. Text Annotation

A *Text Annotation* in IDA- STEP is a 3D annotation with a free text (one or several lines).
A user has a possibility to create, modify, delete a text annotation in the **GD&T** view. The created text annotation is presented in the **3D View** (see Figure 1).

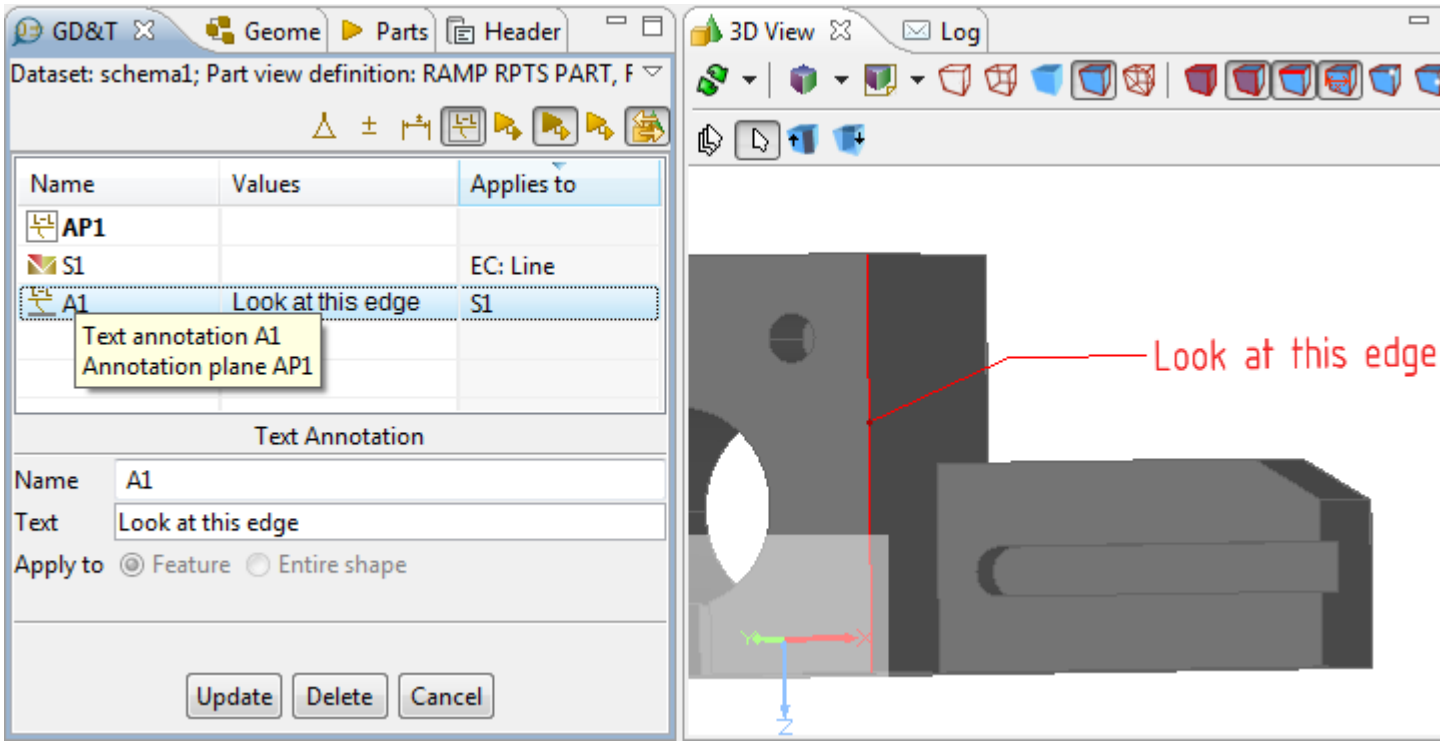

Figure 1. Text annotation in GD&T view and in 3D View

Create text annotation

Follow the routine to create a text annotation in the **GD&T** view table:

- Select a shape aspect in the table. In our example a *Line* is selected.
- Right click and select the **New Text Annotation** item in the appeared context sensitive menu.
- The **Create Text Annotation** wizard dialogue appears in the details area of the **GD&T** view. **Note.** The same result can be achieved by pressing the **Create Text Annotation** button in the main tool bar or selecting in the main manu bar **GDT -> Create Text Annotation**. How to manipulate with the wizard refer to clause Create Text Annotation Wizard.

The created text annotation entry appears in the **GD&T** view table (see Figure 1):

- the identification of a text annotation is presented in the **Name** column,
- the shape aspect the text annotation is applied to (i.e. *S1*) is displayed in the **Values** column,
- the geometry element the created shape aspect is applied to (i.e. *Line*) is defined in the **Applies** to column.

Fly over the text annotation icon to view a tool tip, which contains additional information:

- The text *Text Annotation* with the text annotation identification value, and
- The annotation plane the created text annotation is currently associated to.

Note. A new annotation plane (i.e. *AP1*) is created together with a new text annotation (i.e. *A1*), in case no annotation plane were previously created.

Create Text Annotation Wizard

The **Create Datum Feature** wizard helps to create a datum feature for selected element (see Figure 2):

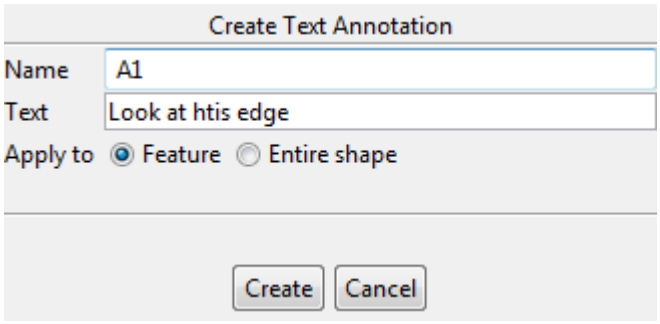

Figure 2. Create Text Annotation wizard page

- The default identification label is automatically added in the **Name** text field (in our example 'A1'). To change the value - enter a new text, identifying the text annotation.
- Enter a text (one or several lines) for the text annotation into the **Text** field.
- Choose if the text annotation should apply to a selected geometric feature or to the entire shape by selecting a particular radio button in the **Apply to** radio group:
    - Select the **Feature** radio button to apply a text annotation to the selected geometric feature, or
    - Select the **Entire shape** radio button to apply a text annotation to the whole entire shape.
- Press the **Create** button to create a new text annotation.

Update text annotation

To update a text annotation parameters:

- Select a text annotation entry in the **GD&T** view table. The **Text Annotation** wizard dialogue appears in the details area.
- Change parameters for the selected annotation in the wizard. How to do this, refer to clause Create Text Annotation Wizard.
- Press the **Update** button to confirm your changes.

Delete text annotation

To remove a text annotation:

- Select an applicable text annotation entry in the **GD&T** view table.
- Press **Delete** button in the **GD&T** view details area.

Or

- Right click on a text annotation entry in the **GD&T** view table.
- Select **Delete** menu item in the appeared context sensitive menu.

To top

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9.8.1. Geometry Details View

The **Geometry Details** view presents a hierarchy of construction geometry models, shape

representations, related to a particular part version, and style/annotation models with associated objects, such as annotation planes and occurrences (see Figure 1).
The content of the **Geometry Details** view depends on the selection in the **Parts** view (selected part version). If no part version is selected in the **Parts** view, the **Geometry Details** view shows an empty page with a message "There is no selected part".

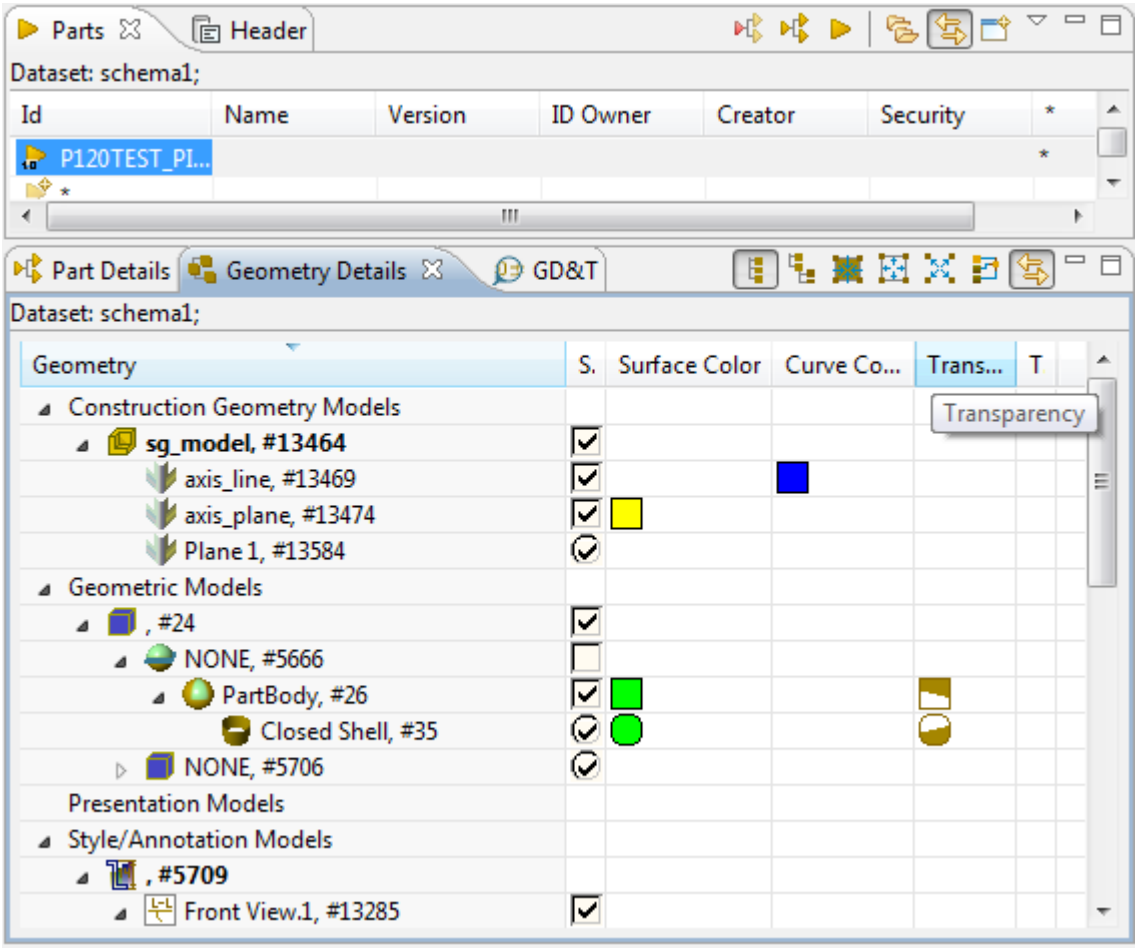


Figure 1. Geometry Details view

The **Geometry Details** view space is occupied by a tree table:

- **Construction Geometry models -** hierarchy of supplementary geometry models and their details plus styling. The node types in a standard view:
  - Constructive geometry representation,
  - Placement,
  - Curve/Edge,
  - Surface/Face,
  - Vertex,
  - Point.
- **Geometric Models** - hierarchy of geometry models of the selected part/version/view with details and styling. The node types in a standard view:
  - Shape,
  - Shape relationship,
  - Solid,
  - Shell,
  - Face,
  - Edge,
  - Vertex.
- **Style/Annotation Models** - hierarchy of style/annotation models with their annotation planes with associated annotation occurrences objects (see Styling Models for details). The node types in

a standard view:
    - Style Model,
    - Annotation Model,
    - Style + Annotation Model,
    - Not styled or annotated model,
    - Annotation planes,
    - Annotation occurrences.
- **Presentation Models** - presentation areas and views.

Data representation for all non-static entries in the first column of the **Geometry Details** view is organized in the following way:

- an icon, in front of each node is specific to the type and role of the object,
- name of representation or representation item (if the name is an empty string, it is displayed just the separator),
- separator "," and
- hash-number, e.g. "#1234".

Note. Look to the fly by text to learn more details about each non static node.

The default column perspective contains the following columns (full name of the column is presenetd in tool tips):

- **Show/Hide** - the check box to specify whether a particular object is visible or not in the **3D view** (see Editing Visibility for more details);
- **Surface Color** - a color for a surface (see Editing Color for more details). The 'square color' indicates explicitly set color, the 'round color' - inherited color.
- **Curve Color** - a color for an edge (see Editing Color for more details). The 'square color' indicates explicitly set color, the 'round color' - inherited color.
  Note. The **Geometry Details** view does not display "default" styling information. All cells that use the actual system defaults are empty.
- **Transparency** - transparency for a surface (see Editing Transparency for more details);
- **Transformation** - the transformation data of the shape representation object (see Editing Transformation Data for more details). If the selected element contains transformation data, the **T** letter symbol appears in the column cell. Detailed transformation data are presented in the fly by text.

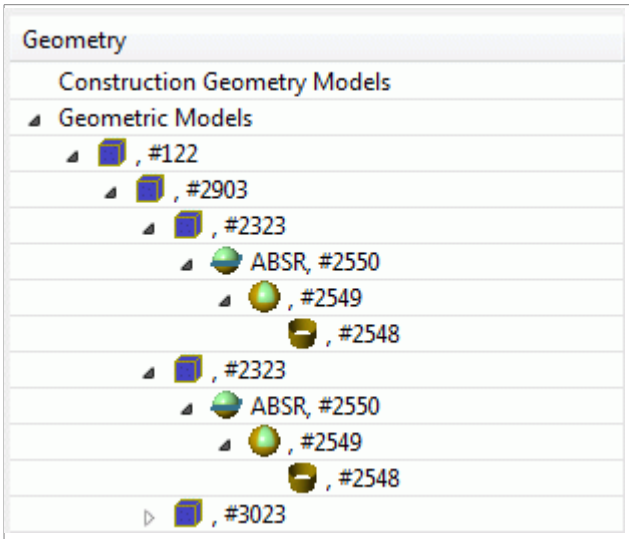
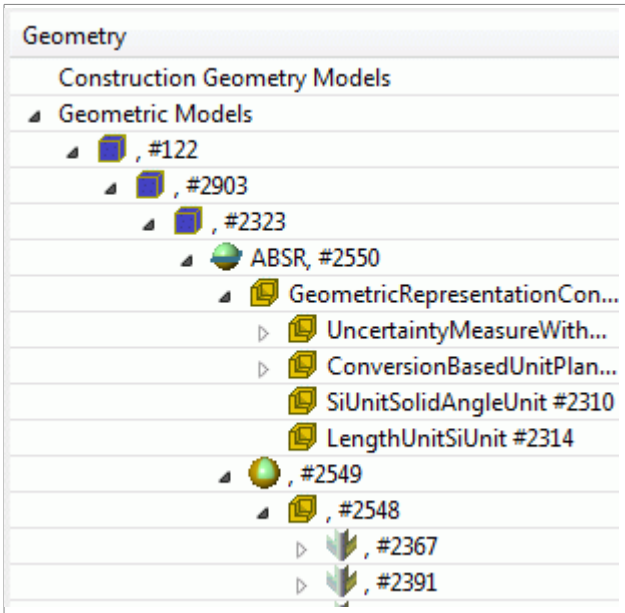
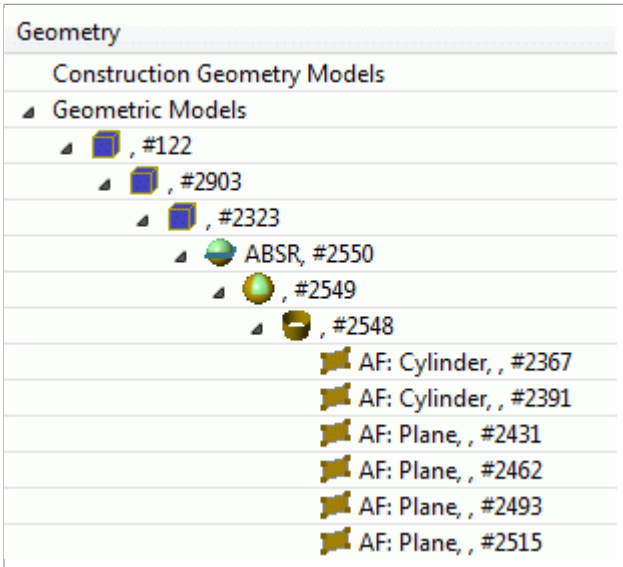
Note. The **Geometric Details** view does not display any default styling information, which can be found in **Preferences -> 3D View -> Display Styles** page.

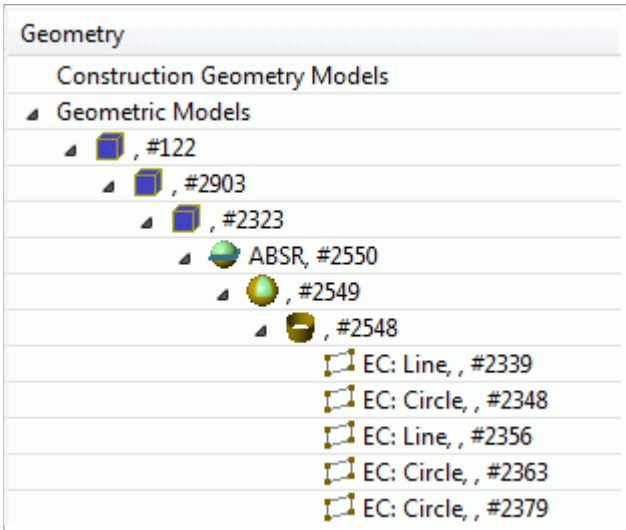
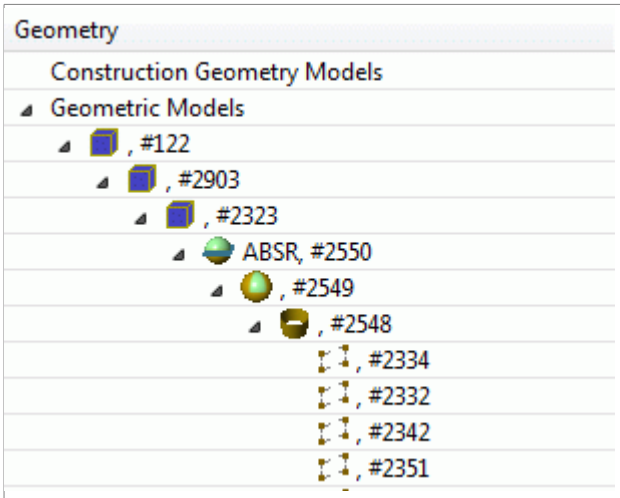
To top

Tool Bar

The **Geometry Details** view contains it's own tool bar, where a preferable view of the geometry details hierarchy can be switched On/Off.

Button Icon	Description

	Show standard view	
	Show along STEP data structures	
	Show Face objects	

	Show Edge objects	
	Show Vertice objects	
	Show geometry with shared context	
	Link with Views	When this option is turned *On*, the **Geometry Details** view is linked with the **3D View** and **Part Details** view. Any selection in one of the views is reflected in the other views.

To top

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9.8.2. Selection

The selection of some geometry element in the **Geometry Details** view initiates the highlighting of corresponding geometry objects in the **3D View** and related shape aspects in the **Parts Details** view (see Figure 1).

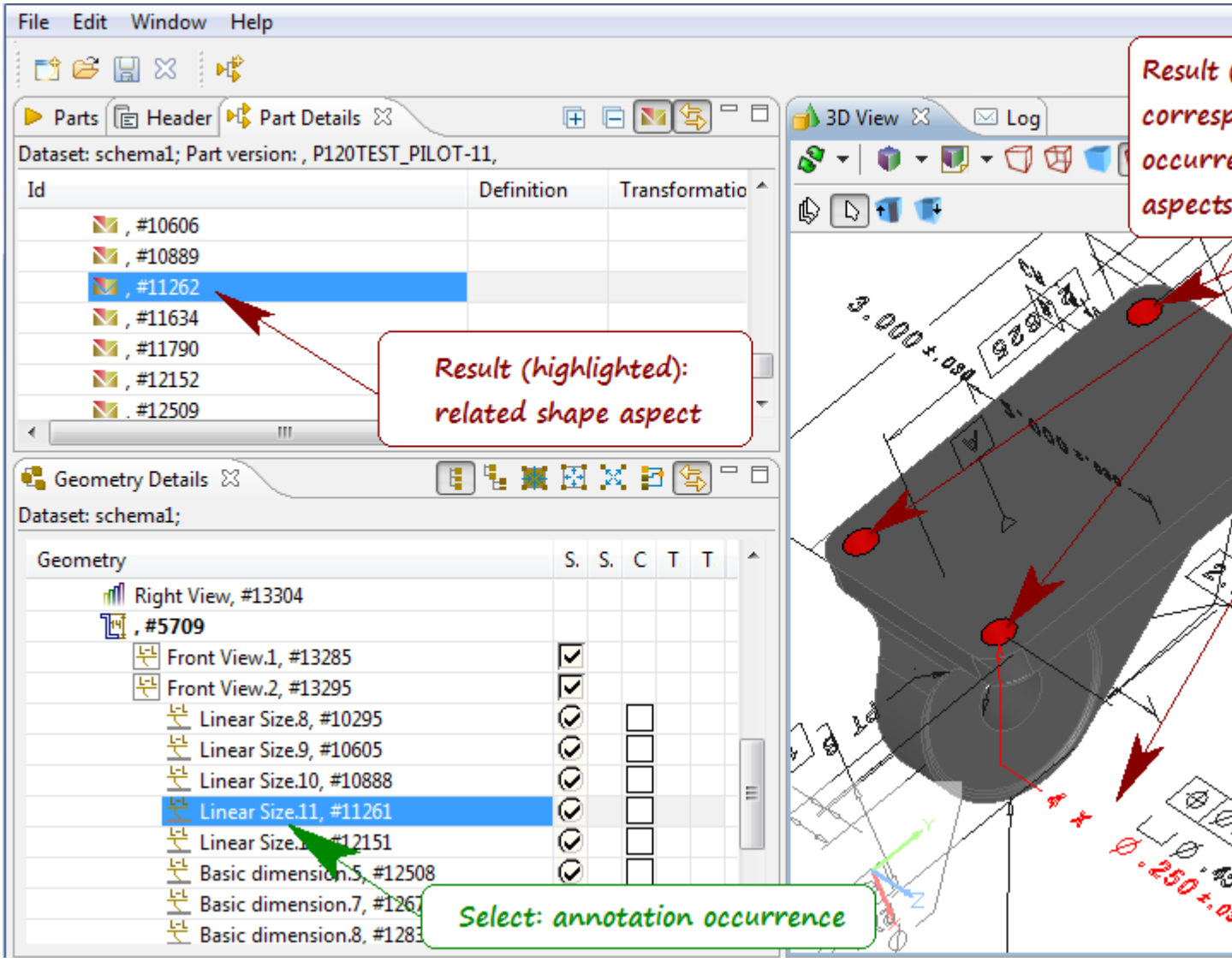

Figure 1. Selection in Geometry Details view initiates highlighting of related objects in 3D View and Part Details view

Note. You can use buttons in the tool bar of the **Geometry Details** view to switch an applicable view of the geometry details hierarchy. This enable you to select various geometry elements, which are not displayed in the standard view.

To top

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9.8.3. Construction Geometry Models

The construction geometry models with elements (lines, planes) are listed under the static node **Construction Geometry Models** in the **Geometry Details** view (see Figure 1):

- **Root node** - construction geometry models;
- **Child node** - simple construction geometry element (line, plane, point).

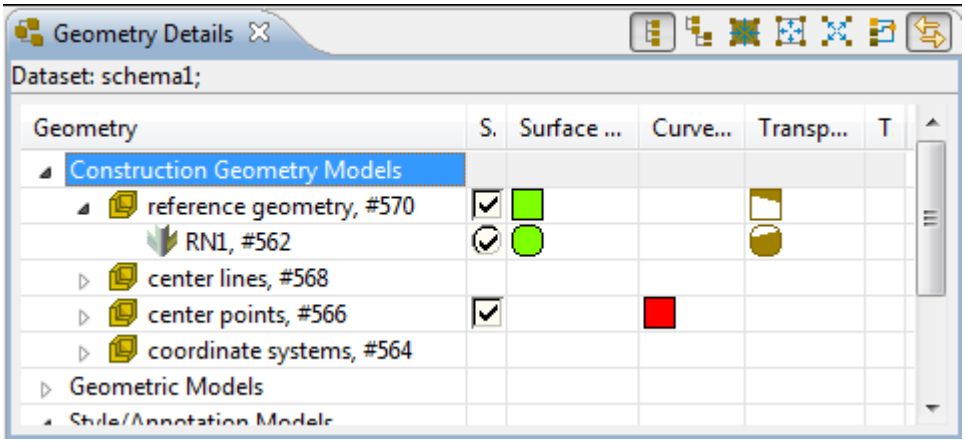

Figure 1. Geometry Details View, Construction Geometry Models hierarchy

Create new model

Follow the routine to create a new construction geometry model:

- Right click on the **Construction Geometry Models** static node.
- Select **Create model** menu item in the appeared context sensitive menu.

A new node appears under the **Construction Geometry Models static** node, named 'New model' (see Figure 2).

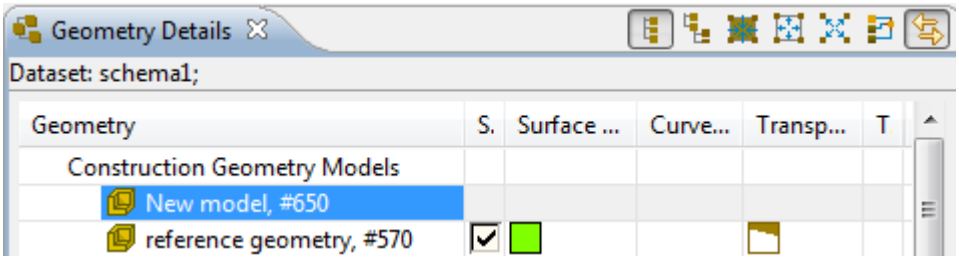

Figure 2. Created 'New model' node under Construction Geometry Models static node

How to create lines and planes for a construction geometry model, see Create line, Create plane.

Rename construction geometry model

The initial name of the selected construction geometry model can be changed by:

- Select the construction geometry node (for example 'New model').
- Double click or pres F2 key to enter the cell.
- Enter an applicable text, which represents the model.

The entered name appears instead of the previous one under the **Construction Geometry Models** static node and is updated in the **3D View** icon drop down list.

Create annotation plane

It is possible to to create an annotation plane from a plane that is available in **Construction Geometry Models**:

- Right click on a plane under the **Construction Geometry Models** branch.
- Select **Create annotation plane** menu item in the appeared context sensitive menu.

In case no style/annotation model is active, a new annotation model is created under the **Style/Annotation Models** branch, otherwise the active sty;e/annotation model is supplemented with a new annotation plane.

Activate construction geometry model

If there exist only one construction geometry model, it is the default model. In other case, an applicable model should be activated:

- Right click on a construction geometry model and select **Activate model** menu item in the appeared context sensitive menu.

  Or

- Press an arrow next to the button in the **3D View** tool bar and select an applicable construction geometry model in the drop down list.

The active construction geometry model is displayed in bold style under the **Construction Geometry Models** branch.

Style elements

There is a possibility to style construction geometry models and their elements:

- construction geometry model - change visibility, surface and curve colour, transparency;
- line, vertex point - change visibility and curve colour;
- plane - change visibility, surface colour and transparency.

For more information see Editing Visibility, Editing Colors, Editing Transparency.

To top

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9.8.4. Style/Annotation Models

Models, listed under the **Style/Annotation Models** static node in the tree table of the **Geometry Details** view (see Figure 1) are of two kinds: styling (e.g. assigned colours, visibility, transparency) and defined annotations (typically draughting annotations).

- **Root node** - style/annotation models of the current shape representation. The icon in front of the node name identifies a content of a styled model:
  - Style Model - style only;
  - Annotation Model - annotation only;
  - Style + Annotation Model - style and annotation;
  - No Style/Annotation Model

Note. The fly by text in the **Style/Annotation Models** tree also identifies a content of a styled model (i.e. "DraughtingModel (Style):<name>", "DraughtingModel (Annotation):<name>" and etc.).

- **Child nodes** - linked style/annotation models, annotation planes with associated annotation

occurrences.

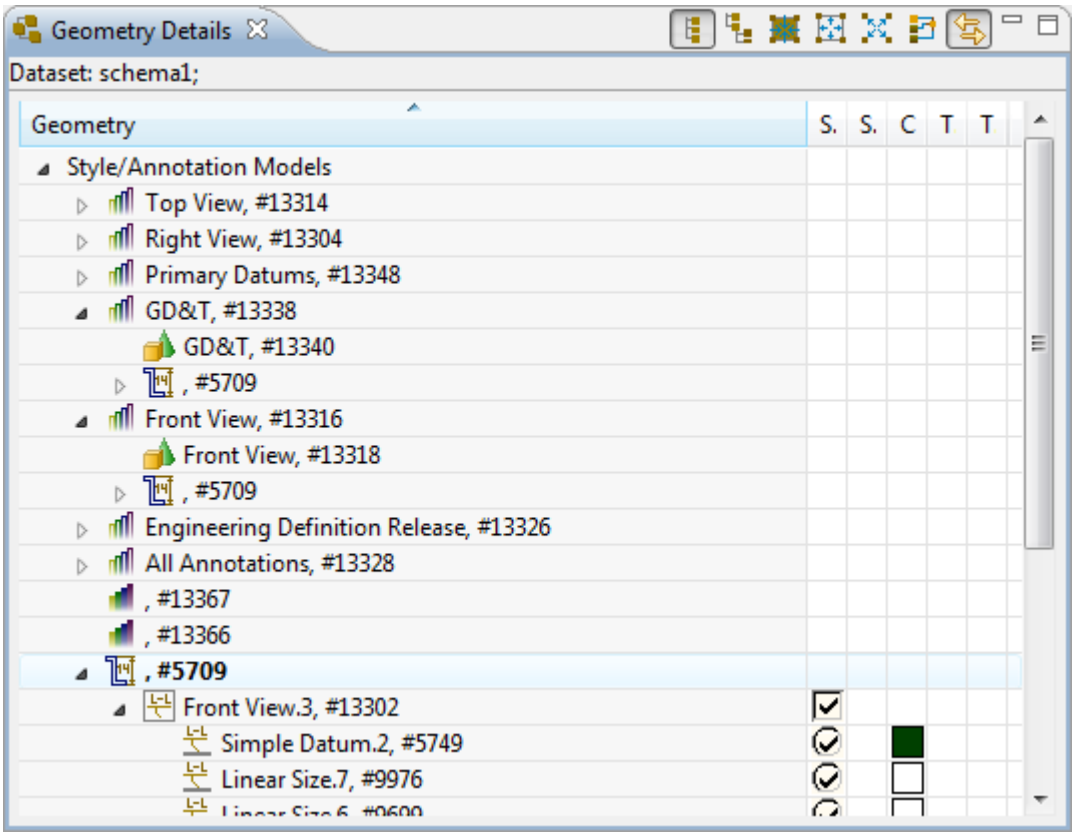

Figure 1. Hierarchy of styling models

Find below the description, how to manage style/annotation models.

Change Active Style/Annotation Model

While loading STEP data, some style/annotation model is automatically selected to be activated upon displaying some 3D shape. In case the automatically selected style/annotation model is not suitable for a user this model can be changed by activating another style/annotation model.
There are two ways, how to change the currently active style/annotation model:

1. Right click on the root node under the static **Style/Annotation Models** node in the tree table of the **Geometry Details** view and select **Activate** menu item in the appeared context sensitive menu.
2. Select a style/annotation model in the icon combo box, which is located in the tool bar of the **3D View**. A particular node is highlighted under the **Style/Annotation Models** node in the **Geometry Details** view tree table.

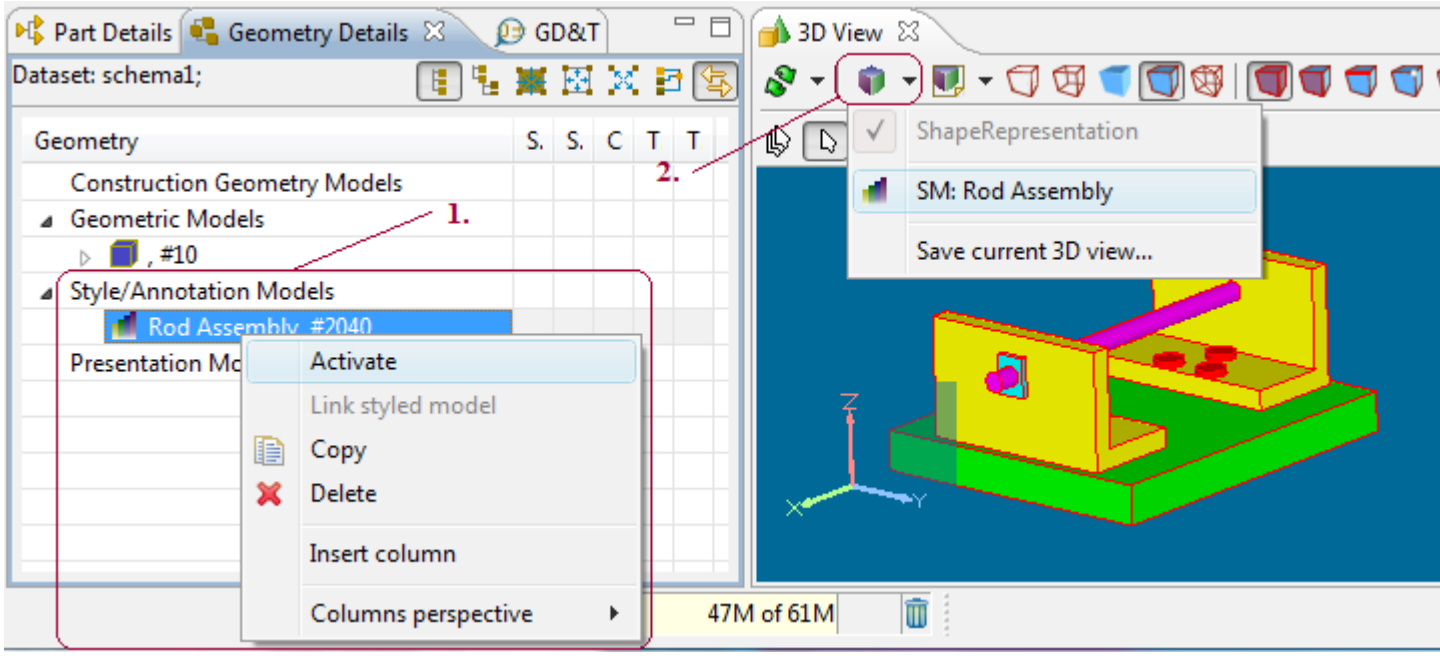

Figure 2. Activate Style/Annotation Model ('Rod Assembly' - style model)

Create Style/Annotation Model

The editing of representation properties is performed in the context of the active style/annotation model. If there is no active model selected, a new style/annotation model is created as a root node under the **Style/Annotation Models** node. A new value of the representation property should be associated with the newly created style/annotation model.
Note. The geometry with no style/annotation is available after STEP file is initially opened and no style/annotation model is selected for display or after the icon button is presses in **3D View** tool bar.

Create new

This function creates a model with default presentation properties:

- Right click on the **Style/Annotation Models** node in the tree table of the **Geometry Details** view.
- Select the **Create new** menu item in the appeared context sensitive menu (see Figure 3).

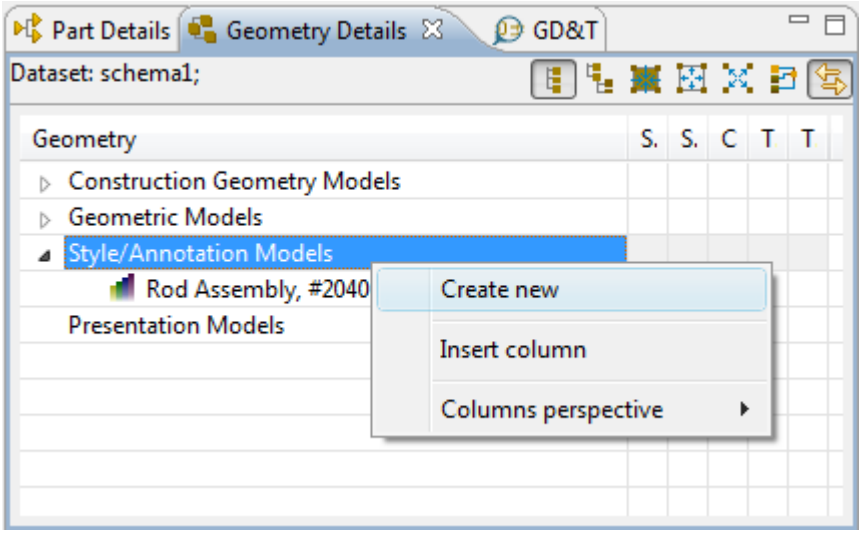

Figure 3. Geometry Details view, Create new menu item

A new node with the icon appears as a root node under the **Style/Annotation Models** branch (see Figure 4).

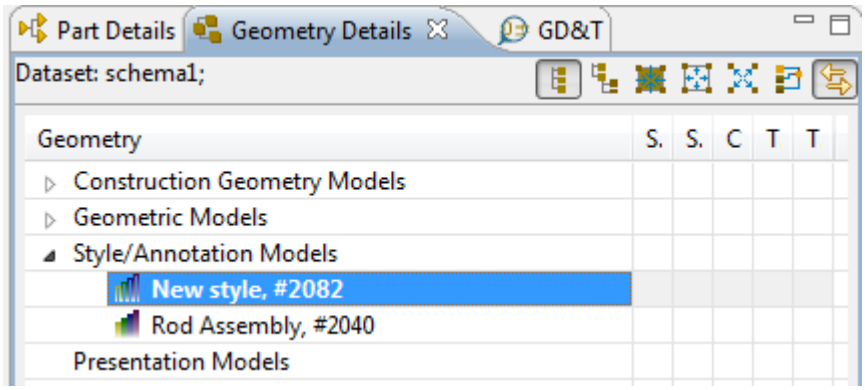

Figure 4. Created 'New style' node under the Style/Annotation Models branch

Note. The newly created node is highlighted (made active for the current shape representation). The same model name is made active in the icon combo box in the **3D View**.

Create Style Model

Create a new Style Model under the **Style/Annotation Models** branch by editing initial style for the shape representation without any active style/annotation model:

- Unselect the active styling model (if any is selected) by pressing on icon button in the **3D View**.
- Change any property (visibility, surface/curve colour, transparency) of an applicable shape representation object.

A new style model node (i.e. '**New style 1**') is created in a tree table under the **Style/Annotation Models** node as a root node. **Note.** Names of new style/annotation models are created with the number suffix to ensure uniqueness in the same representation context. This style model name is marked as active in the icon combo box in the **3D View**. And also a new style of the shape representation is applied to the **3D View** object (see Figure 5). The tool tip of the icon button contains the name of the currently created style model.

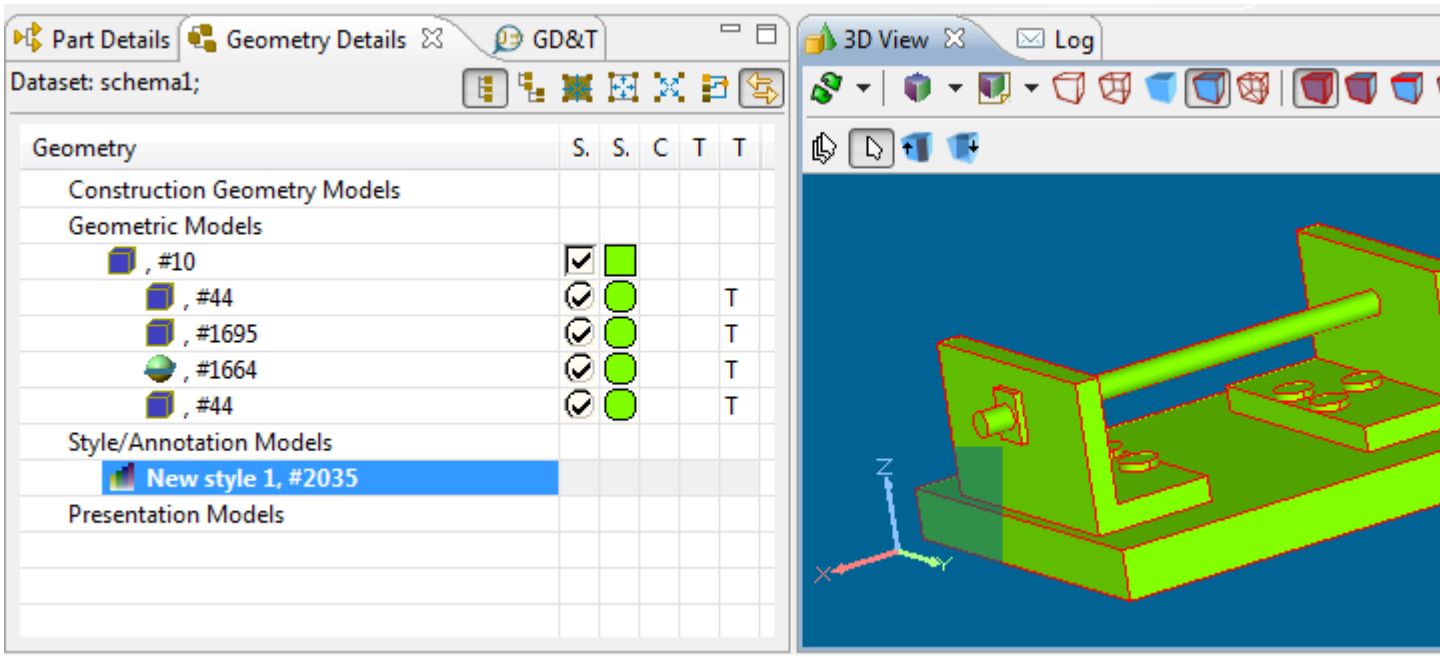

Figure 5. New styling model highlighted in tree table under Styling Models node (left), selected in drop down list and applied to 3D view object (right)

Note. The icon identifies, that this model contains styling only (visibility, surface/curve colour or transparency).

Create Annotation Model

The Annotation Model node is identified by the icon .
Follow the routine to create a new annotation model:

- Make active an applicable style/annotation model which is marked with icon under the **Style/Annotation Models** branch in the **Geometry Details** view.
- Add some annotations in the **3D Viewer** (for example a new annotation plane).

The initial icon of the activated model is changed to and the node of newly created annotation plane appears as a child node of the annotation model branch.

Create Style + Annotation Model

The model, which contains styling and annotations is identified by the icon (i.e. +).
The Style and Annotation Model is created if:

- Annotation Model is supplemented with some styling (visibility, colour or transparency), and
- Style Model is supplemented with some annotation planes/occurrences.

To top

Rename Style/Annotation Model

There is a possibility to give a new name for a styling model. This will update style/annotation model name in the tree table of the **Geometry Details** view and icon combo box drop down list of the **3D View**.
Follow the routine to rename a style/annotation model:

- Select a style/annotation model node in the tree table.
- Double click on the node or press **F2** key. The editing mode is switched on.
- Enter a new word or group of words that represent a new name of a style/annotation model and press **ENTER** key.

The style/annotation model new name appears in the tree table and it is updated in the icon combo box drop down list (**3D View**).

To top

Editing Style/Annotation Model

The editing of a style/annotation model is available if this model is active. The following changes can be made in the **Geometry Details** view:

- Show/hide geometric models, annotation planes and occurrences (see Editing Visibility for details);
- Change surface colour for geometric models;

- Change curve colour for geometric models, annotation planes and occurrences;
  Note. See Editing Color for more details.
- Change transparency for geometric models (see Editing Transparency for more details).

Note. In case the style/annotation model is changed, the icon is applied automatically (i.e, the style model is supplied with annotations and vice versa, etc.).

To top

Link Style/Annotation Model

There is a possibility to link some style/annotation model with another style/annotation model, especially if the model does not have any style/annotation itself.
Follow the routine to link a root styling model with an applicable styling model:

- Select a styling model to be linked with the root styling model.
- Right click and select **Copy** menu item in the appeared context sensitive menu.
- Select a root styling model to be linked with a copied to the clipboard model
- Right click and select **Link styled model** menu item in the appeared context sensitive menu.

To top

Delete Style Model

You can delete an incorrect or a redundant style/annotation model with all linked models from the **Geometry Details** view. This will remove the style/annotation model from the tree and the icon drop down list in the **3D View**. **Note.** If the active style/annotation model is deleted, this will change a selection in icon drop down list.
Follow the routine to delete a style model:

- Select an applicable style/annotation model root node in the tree table under the **Styling Models** branch in the **Geometry Details** view.
- Right click and select **Delete** menu item in the appeared context sensitive menu.

Note. You can delete only the root style models. The child styling models can be unlinked only.

To top

Unlink Style/Annotation Model

Follow the routine to unlink the child style/annotation model:

- Select an applicable child node under the **Style/Annotation Models** branch.
- Right click and select **Unlink this model** menu item in the appeared menu item.

The selected style/annotation model disappears from the tree table of the **Geometry Details** view.

To top

Clear Annotation Styles

The style for annotation elements can be applied individually or inherited from the parent element. For example, in case the annotation occurrence is not styled individually, it inherits a style from the parent annotation plane. If the style of the parent annotation plane is removed, the same happens with the style of the child annotation occurrence. If the annotation occurrence is styled individually (the coloured square in the **Curve colour** column), the remove function (*Clear styles*) of the annotation plane style does not influent the style of this annotation occurrence. The *Clear styles recursively* function on the parent element removes all styles, even the individually styled annotation occurrences of child elements. Follow the routine to remove styling of the parent element, where some child elements have their own set styles:

- Right click on a parent annotation element in the tree table. The context sensitive menu appears on the screen.
- Select one of:
  - **Clear styles** - remove styling for the currently selected element (see Figure 6). After the parent styling is removed, individual styling is used for the child element, which are individually styled (see Figure 7).
  - **Clear styles recursively** - remove styling for the currently selected element and all child objects recursively (even individually set styles).

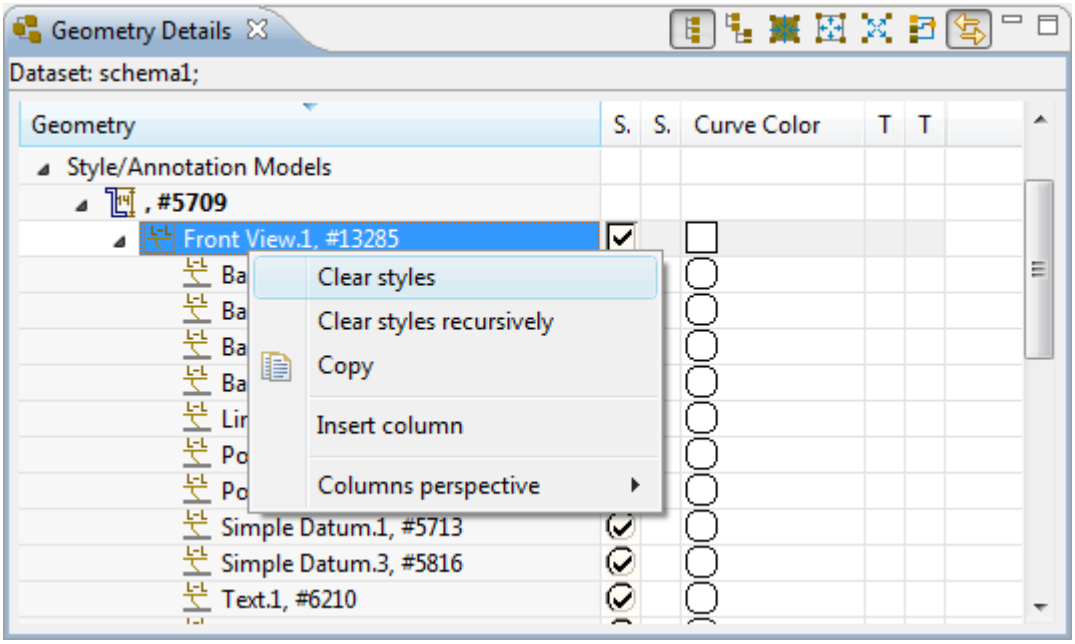

Figure 6. Selected annotation occurrence with explicitly defined style (i.e. white colour rectangle), Context sensitive menu

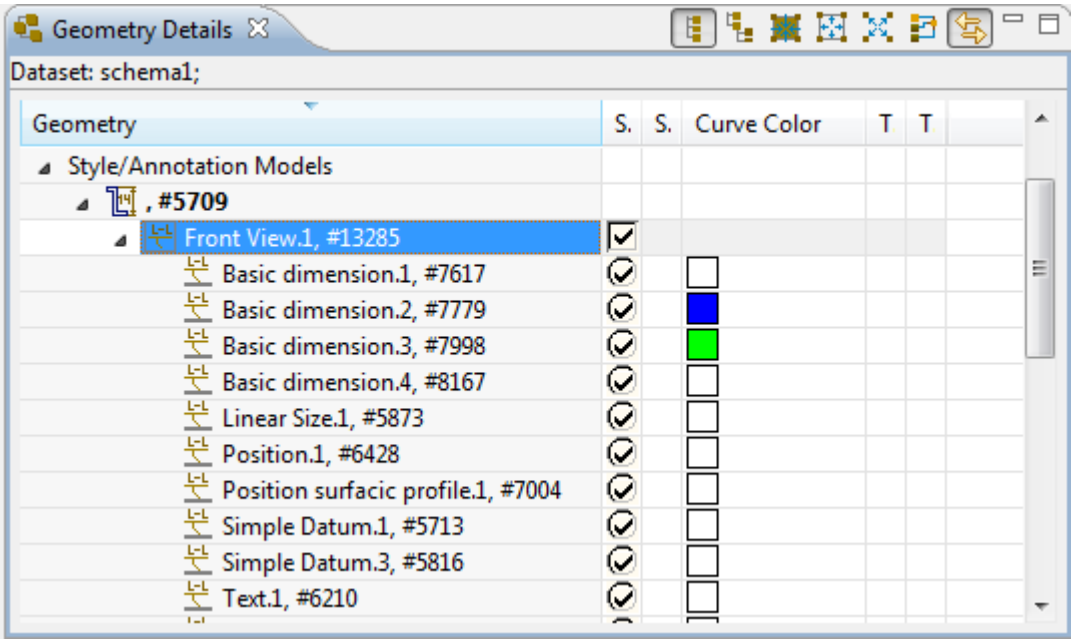

Figure 7. Individually styled annotation occurrences (coloured rectangles)

To top

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9.8.5. Saved Views

The *Saved view* is a stored and retrievable specific orientation and magnification factor of a model. In IDA-STEP the saved view is stored as a child node of the particular styling model under the Styled Model branch in the **Geometry Details** view (see Figure 1).

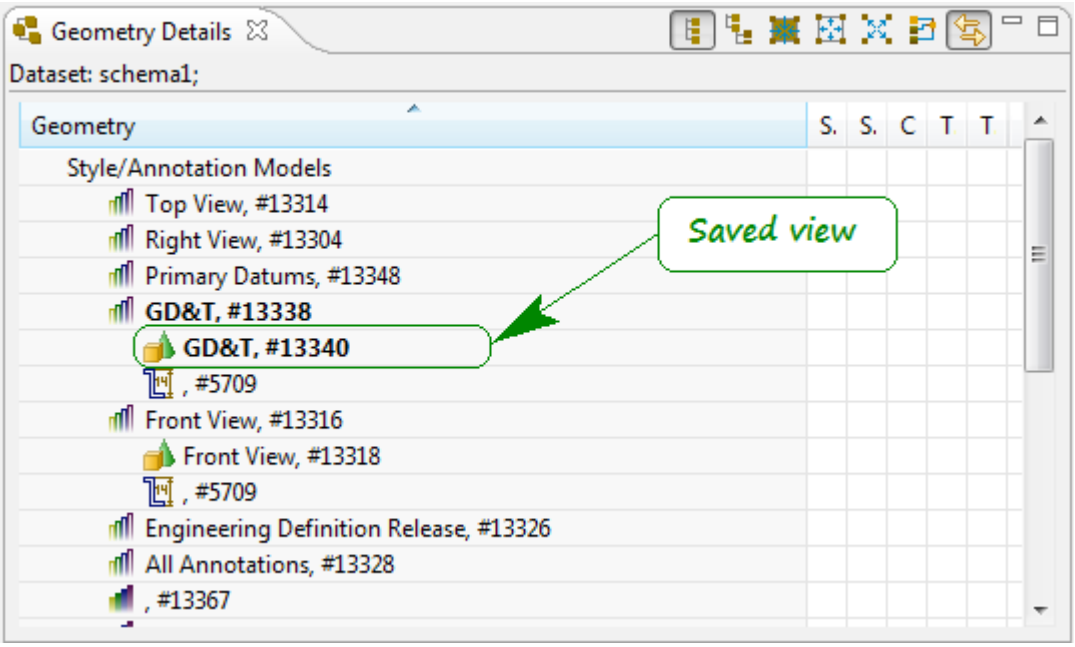

Figure 1. Saved view (activated) under styled model branch

Activate Saved View

To retrieve the selected saved view:

- Right click on an applicable saved view node.
- Select **Activate this view** menu item in the appeared context sensitive menu.

The selected saved view is activated together with the parent styled model. **Note.** The active saved view (and styled model) node fonts are displayed bold (see Figure 1).
The particular saved view can be activated by selecting a particular item in the **Styled model** drop down list in the **3D View**.

Rename Saved View

An incorrect name of an applicable saved view can be renamed:

- Select the saved view node.
- Double click or press **F2** key to turn editing mode *On*.
- Edit the current text or enter a new one and press **Enter** key.

The new name of the saved view appears in the Styled Models tree.

Delete Saved View

To remove redundant or incorrect saved view:

- Right click on the saved view node.
- Select **Delete** menu item in the appeared context sensitive menu.

The selected saved view node disappears from the Styled Models tree.

To top

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9.8.6. Editing Visibility

An object, selected in the **Parts** view, can be visible or not in the **3D View**. The visibility is controlled by check-box in the tree table **Show/Hide** column of the **Geometry Details** view.
There can be the following states of the **Show/Hide** column cell (see Figure 1):

1. **Empty cell** - there is no explicitly set style data.
2. **Square box with a check mark inside** - the row object is explicitly styled / visible;
3. **Square box without a check mark** - the row object is explicitly styled / invisible;
4. **Square box with rounded corners, with a check mark inside** - the row object has inherited styling data / visible;
5. **Square box with rounded corners, without a check mark** - the row object has inherited styling data /invisible.

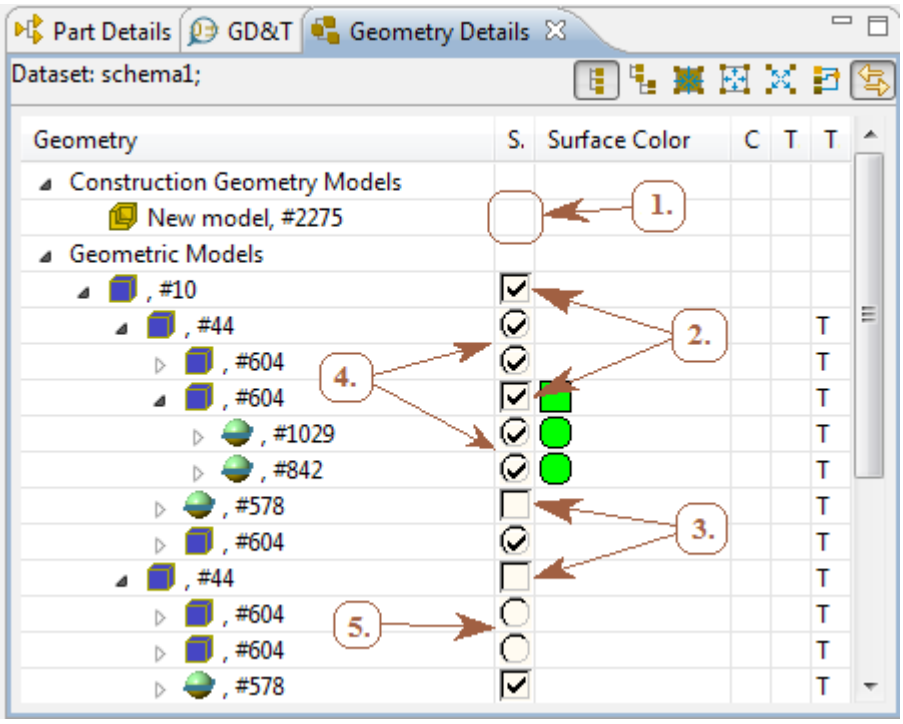

Figure 1. States of Show/Hide column cells

After un-checking the check-box, the selected object is made invisible in the **3D View**. **Note.** All child geometry elements, like surfaces of cube are made invisible together with a selected object. There is a possibility to show/hide individual surfaces together with edges (see Figure 2).

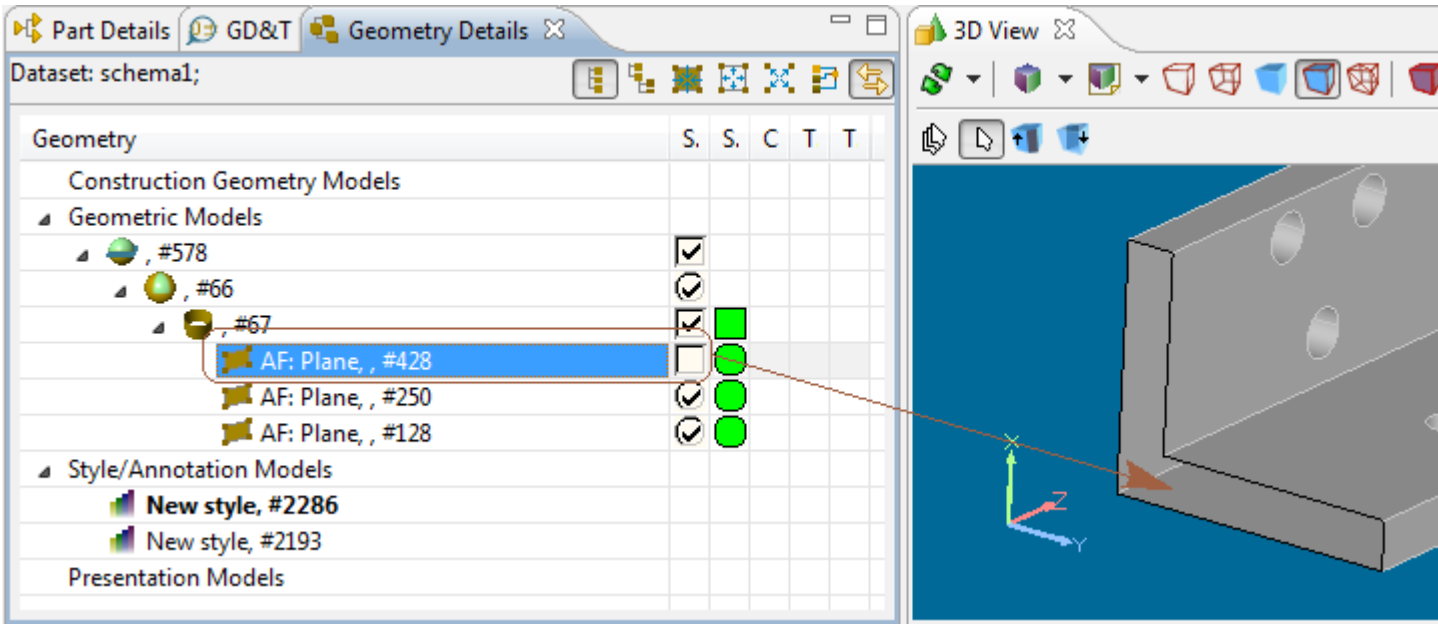

Figure 2. Un-checked surface (in Geometry Details view) is hidden in 3D View

The following table shows the visibility state of the particular object, ‘visible’ or ‘invisible’ in the **3D View**.

Child object	Parent object	Show?
False	False	False
False	True	False

True	False	True
True	True	True
Null	Null	True
Null	False	False
Null	True	True
False	Null	False
True	Null	True

False - individually set to be invisible, True - individually set to be visible, Null - not individually styled

For example, if a child object is individually set as visible, it is represented in the 3D view, in spite of the parent object is set invisible (see Figure 3).

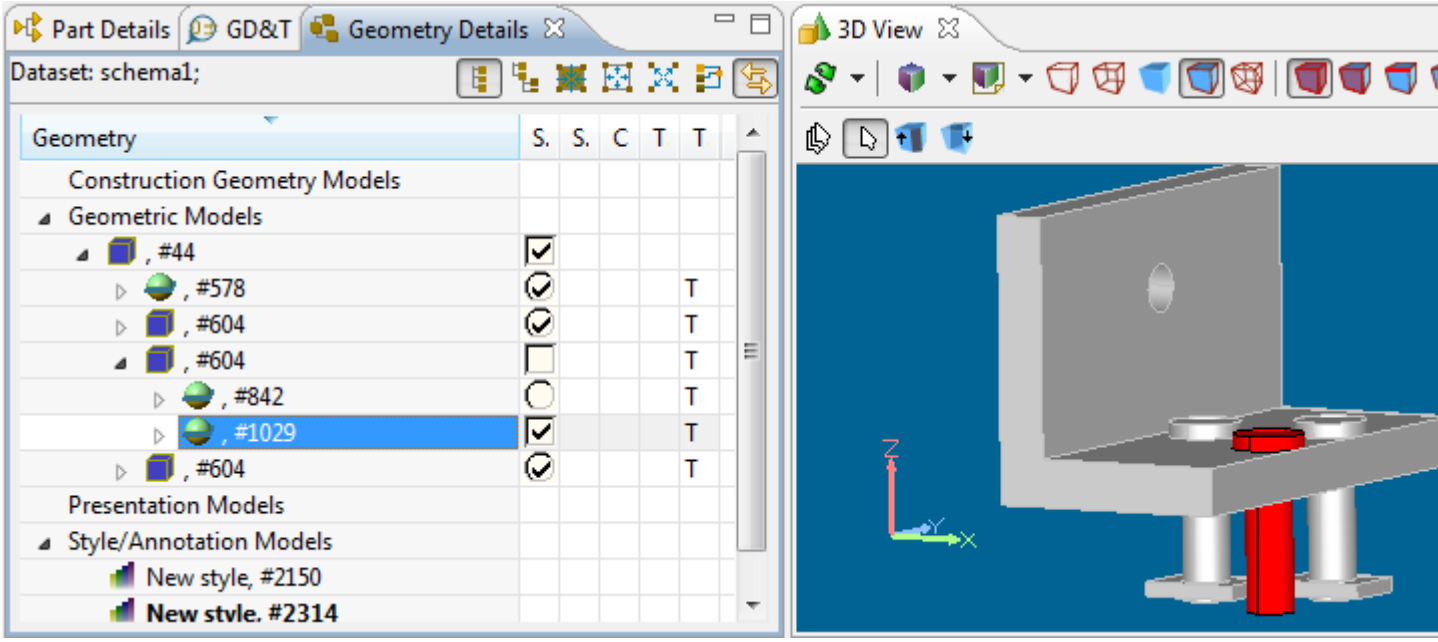

Figure 3. Child object (highlighted in both views) individually set as visible, parent object (#1029) is set as invisible

Another example, if child object is not individually styled, then parent styling is used, when deciding if show/hide the child object.

Clear Show/Hide value

The empty cell in the **Show/Hide** column is displayed right after a STEP file is initially opened and no styling model is selected. There is a possibility to make cells of the **Show/Hide** column empty

- Click on the empty squared check box on the root node under the **Construction Geometry Models/Geometric Models** branch.

Or

- Right click on the **Show/Hide** column cell of the root assembly node and select **Clear** menu item in the appeared context sensitive menu (see Figure 4).

Note. No other styling should be applied to the child row object. This *Clear* function is not available for annotation planes/occurrences under the **Style/Annotation Models** branch.

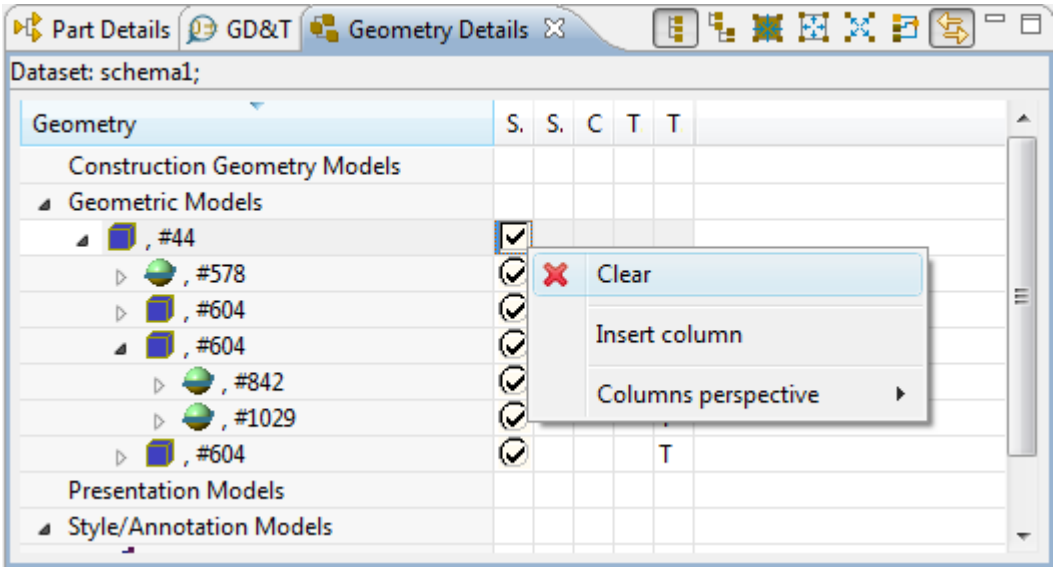

Figure 4. Clear menu item

The **Clear** operation can be used to make the state of a child row object from "explicitly set invisible" to "inherited invisible/visible" (the inherited state depends on the state of the parent row object).

Show/Hide Annotations and Annotation planes

The show/hide for annotations under the **Styled Models** branch works the similar way as written above:

- **Show/Hide** column for annotations should not be empty (of the activated styling model).
- The state of cells should be:
  - **Square box with a check mark inside** - the row object is explicitly styled / visible;
  - **Square box without a check mark** - the row object is explicitly styled / invisible.
  - **Square box with rounded corners, with a check mark inside** - the row object has inherited styling data / visible;

If a particular annotation is used in several styling models, the visibility status of this annotation should be reused (i.e. visible/invisible in all styling models).
Note. An annotation plane is not visible in the **3D View** untill it is highlighted/activated.

To top

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9.8.7. Editing Color

The **Geometry Details** view tree table contains two columns for editing colour:

- **Surface color** - to change a surface fill colour;
- **Curve color** - to change a surface edge or a wireframe colour.

Each object may have an inherited or an individually set colour. If the colour of a child object is not individually styled, the child object inherits the colour of a parent object. If the object has an individually styled colour, it does not inherit the colour of the parent element.
The following content of the **Surface color** and **Curve color** columns is available (see Figure 1):

- *Empty cell* - there is no infomration about a surface colour for the selected row object.
- *Squared icon with colour inside* - the style data for the row object contains information about a colour (surface or curve). A colour of the icon matches the colour in data.
- *Rounded square icon with colour inside* - the style data for the row object dos not contain information about a colour (surface or curve). However, an upper row object has some colour set and that colour is inherited by th elower object (the current row object). A colour of the icon matches the inherited colour of the surface/edge.

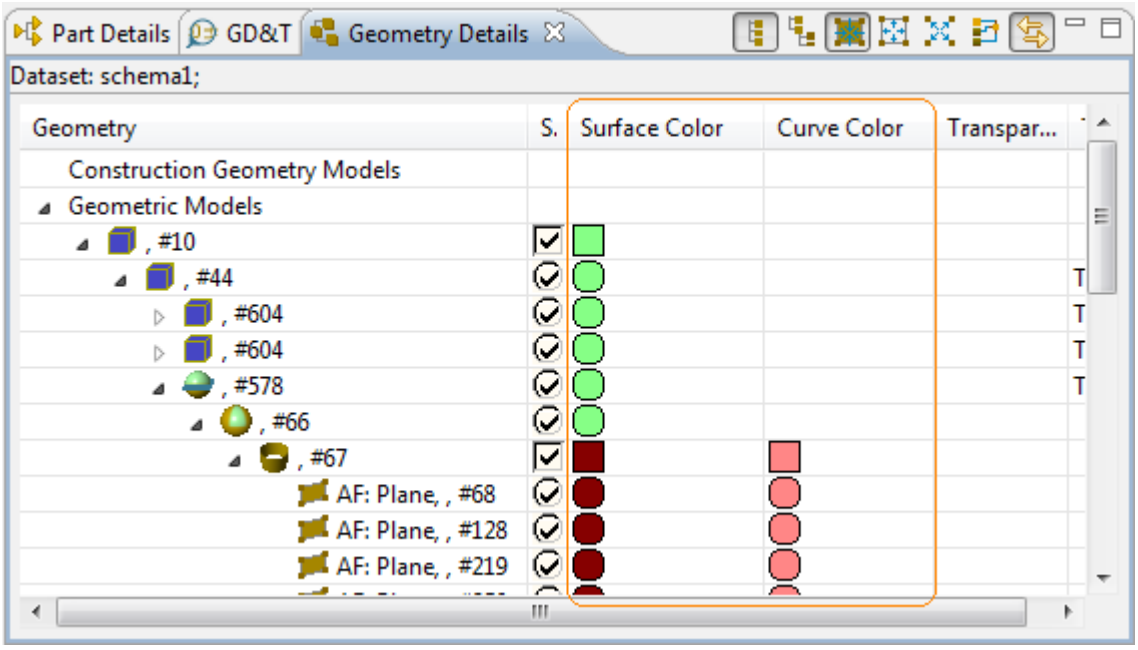

Figure 1. Surface color and Curve Color columns with available content

Change colour

To change a colour for an applicable row object:

- Press on the particular column cell. The **Color** window appears on the screen (see Figure 2).

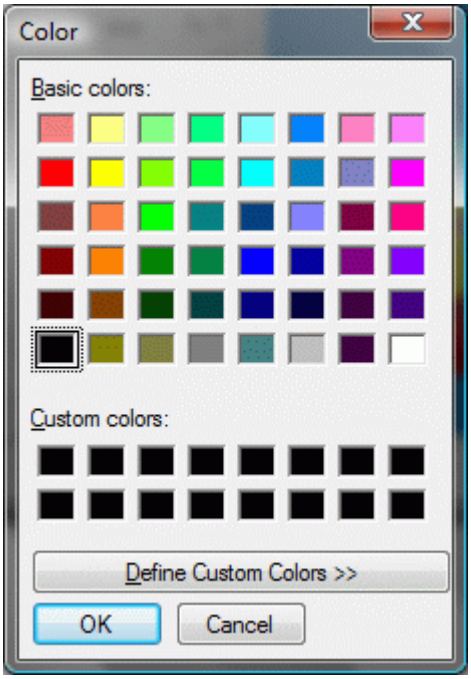

Figure 2. Color window

- Select an applicable colour by pressing on a colour square and press **OK** button. The selected colour is applied for the current row object.

To top

Remove colour

To remove a surface or edge colour:

- Right click on the colour cell. **Note.** Only the explicitly set colour (squared colour icon) can be removed.
- Select the **Delete** menu item in the appeared context sensitive menu.

The colour value is removed from the cell.

To top

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9.8.8. Editing Transparency

The transparency of a particular object can be managed by selecting transparency level value from the list in the **Transparency** column of the **Geometry Details** view tree table (see Figure 1). Available values of transparency level are:

- Opaque,
- 25 %,
- 50 %,
- 75 %,
- 100 %.

Note. The *Reset* function sets default transparency level.

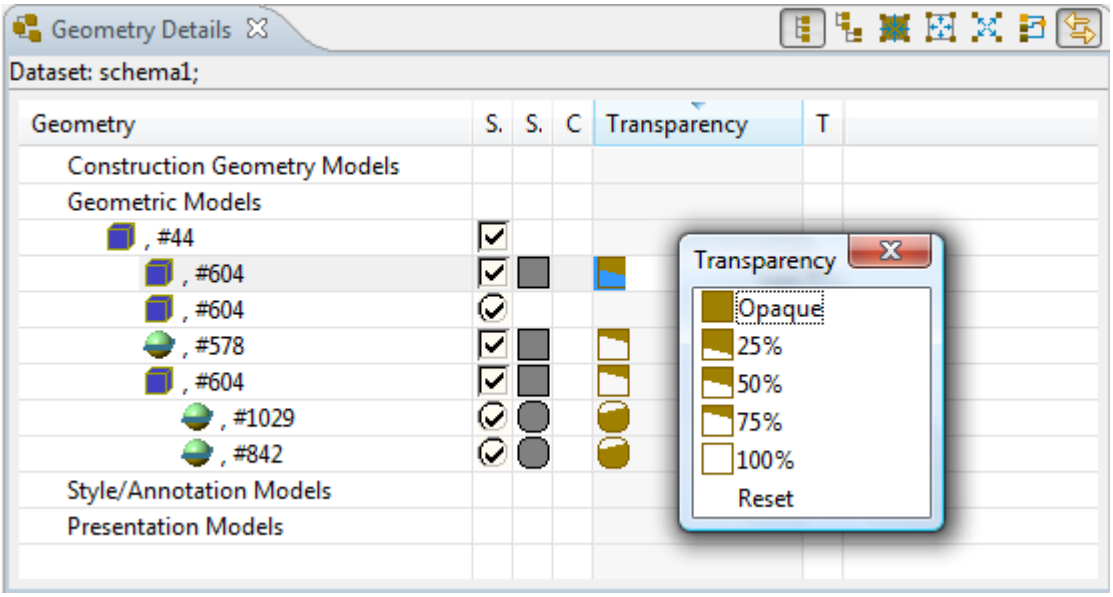

Figure 1. Geometry Details view, Transparency column

An empty cell in the **Transparency** column indicates the respective row object is not transparent (i.e. it is opaque).

Set (update) transparency value

The single click on the cell in the **Transparency** column opens the **Transparency** window (see Figure 1). The user have to select one of predefined values in the list. The selected transparency level will be applied and represented in the **3D View**.
If the edited row object already has a set **Surface Color** value, it is reused for the transparency mesh. Otherwise, it is used the inherited surface colour. If there is no information about the color, then the "dark gray" colour is used for the transparency mesh.

Let's examine how to change transparency level for a row object:

- Click on the cell in the **Transparency** column.
- Select an applicable predefined value in the **Transparency** window list.

The selected object with entered transparency level should be displayed in the **3D View**. The same transparency is applied for all child geometry elements, like surfaces of cube, together with a selected object (see Figure 2). **Note.** There is a possibility to change a transparency level for individual surfaces.

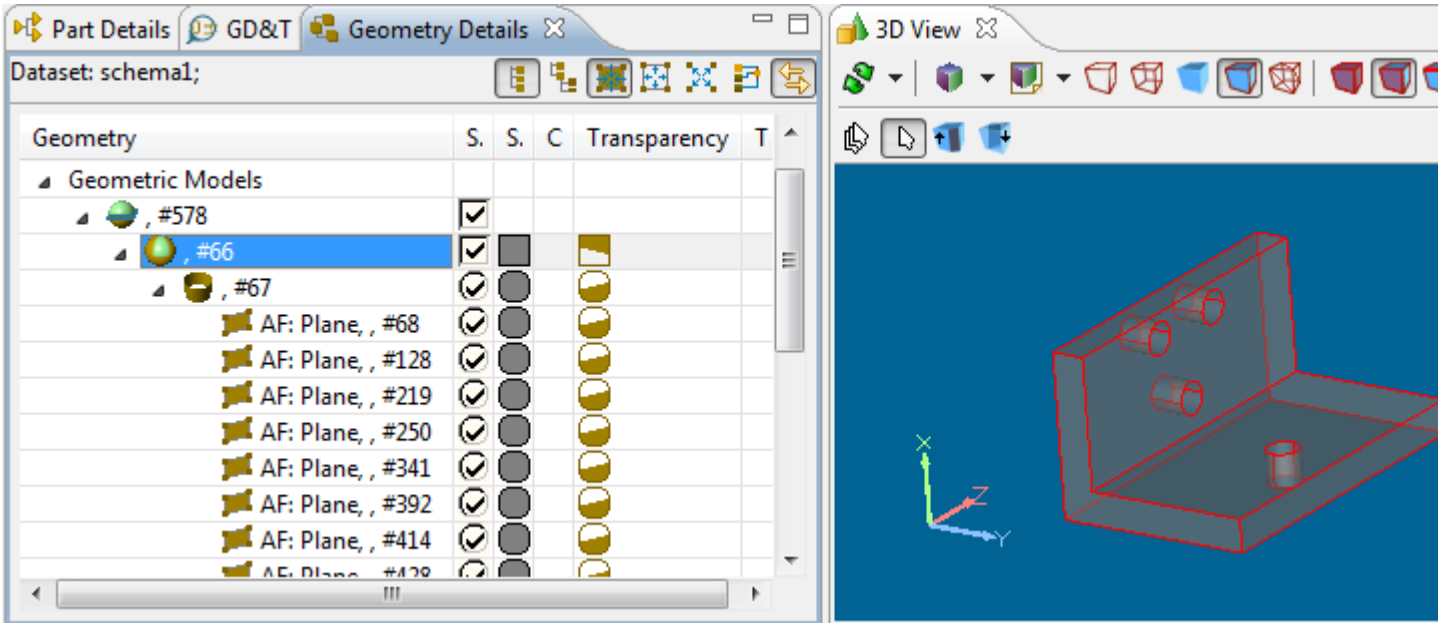

Figure 2. Objects with selected transparency level (in Geometry Details view) are represented in 3D View

Each object may have an inherited or an individually set transparency level. If the transparency of a child object is not individually styled, the child object inherit the transparency of a parent object. If the object has an individually styled transparency, it do not inherit the transparency of the parent element. The explicitly set transparency level value is presented with the squared icon, and the inherited one - with a rounded icon (see Figure 3).

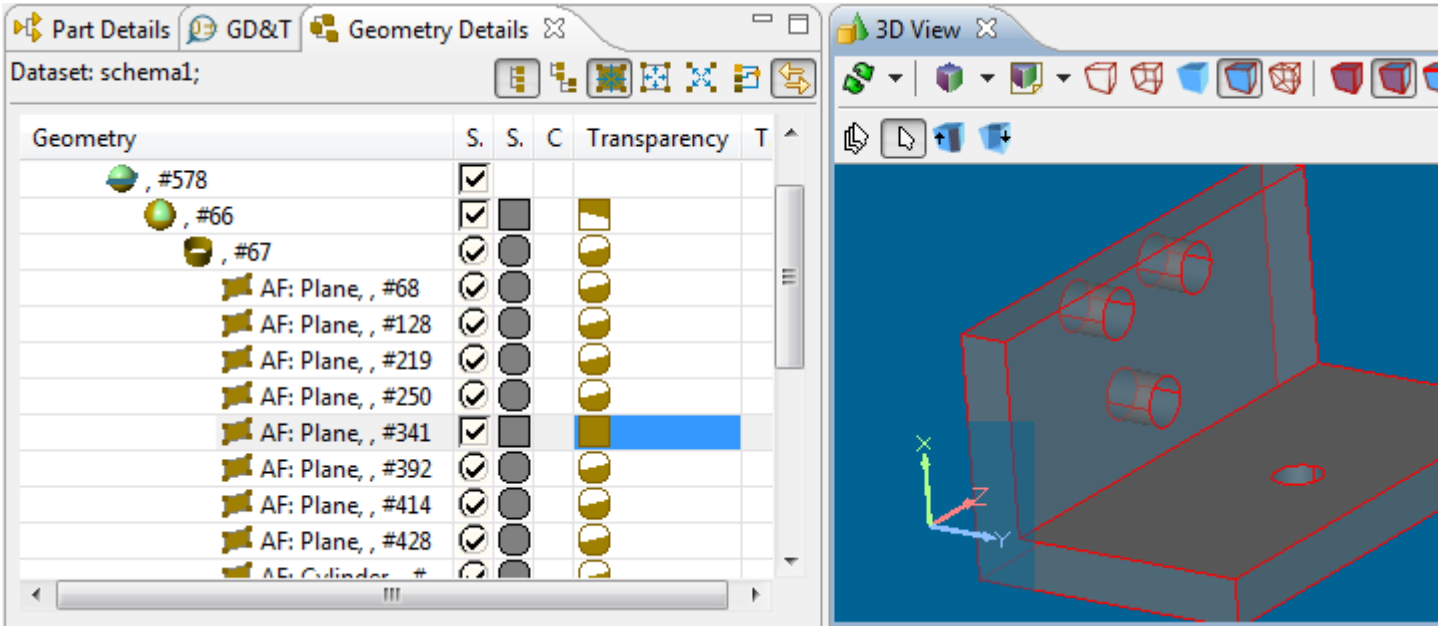

Figure 3. Whole assembly is made transparent (child objects have inherited transparency), except one child object (set individually to be opaque)

To top

Delete transparency value

There is a possibility to remove a redundant or an incorrect explicitly set transparency level value. **Note.** The inherited transparency level value (rounded icon) can not be removed.
To remove transparency level value:

- Right click on a transparency cell (squared icon only).

- Select **Delete** menu item in the appeared context sensitive menu.

The value from the cell is removed.
Note. If there was no surface colour specified in the **Surface Color** cell, the colour for the transparency mesh is removed.

To top

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9.8.9. Editing Transformation Data

The transformation data for a shape representation are presented in the **Transformation** column in the **Geometry Details** view. In case the transformation data exist, the **T** letter is presented in the **Transformation** column cell. Detailed information is presented in the tool tips (see Figure 1).

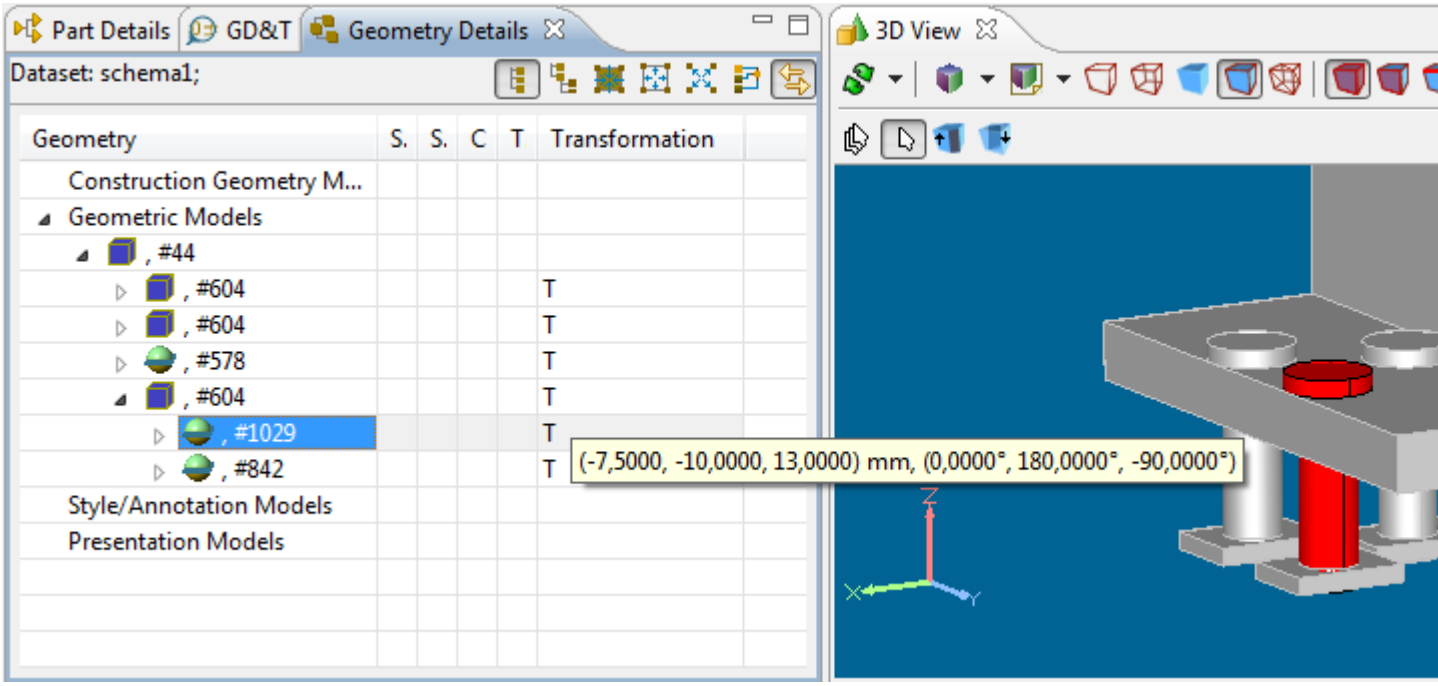

Figure 1. Geometry Details view with Transformation column and tool tip (left side) and highlighted shape representation object (right side)

Follow the routine to change the transformation data for the selected shape representation object:

- Click on the cell in the **Transformations** column next to the shape representation object, which transformation data should be changed.
  The selected shape representation object is highlighted in the **Geometry Details** view tree table (see on the left of Figure 1) and also in the **3D View** (see on the right of Figure 1).
  The **3D Transformation editor** appears on the screen to change transformation data (see Figure 2).

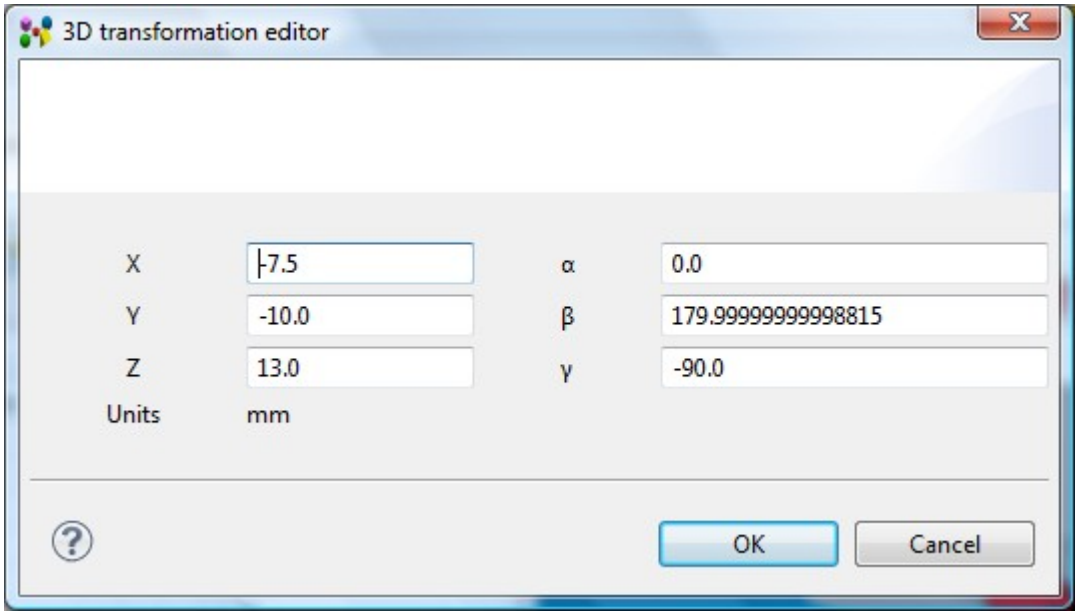

Figure 2. 3D transformation editor

- Enter a number value into the **X** field to change the translation along X axis;
- Enter a number value into the **Y** field to change the translation along Y axis;
- Enter a number value into the **Z** field to change the translation along Z axis;
- Enter a number value into the **α** field to change the rotation around X axis;
- Enter a number value into the **β** field to change the rotation around Y axis;
- Enter a number value into the **γ** field to change the rotation around Z axis.

- Press **OK** button to confirm your changes.

- The defined transformation data appears in the **Transformations** column tool tip and the view in the **3D View** is updated (see Figure 3).
  Note. Units in the column are converted to the units, defined in the **Quantity and Units** preferences (**Window -> Preferences -> Numbers -> Quantities and Units**).

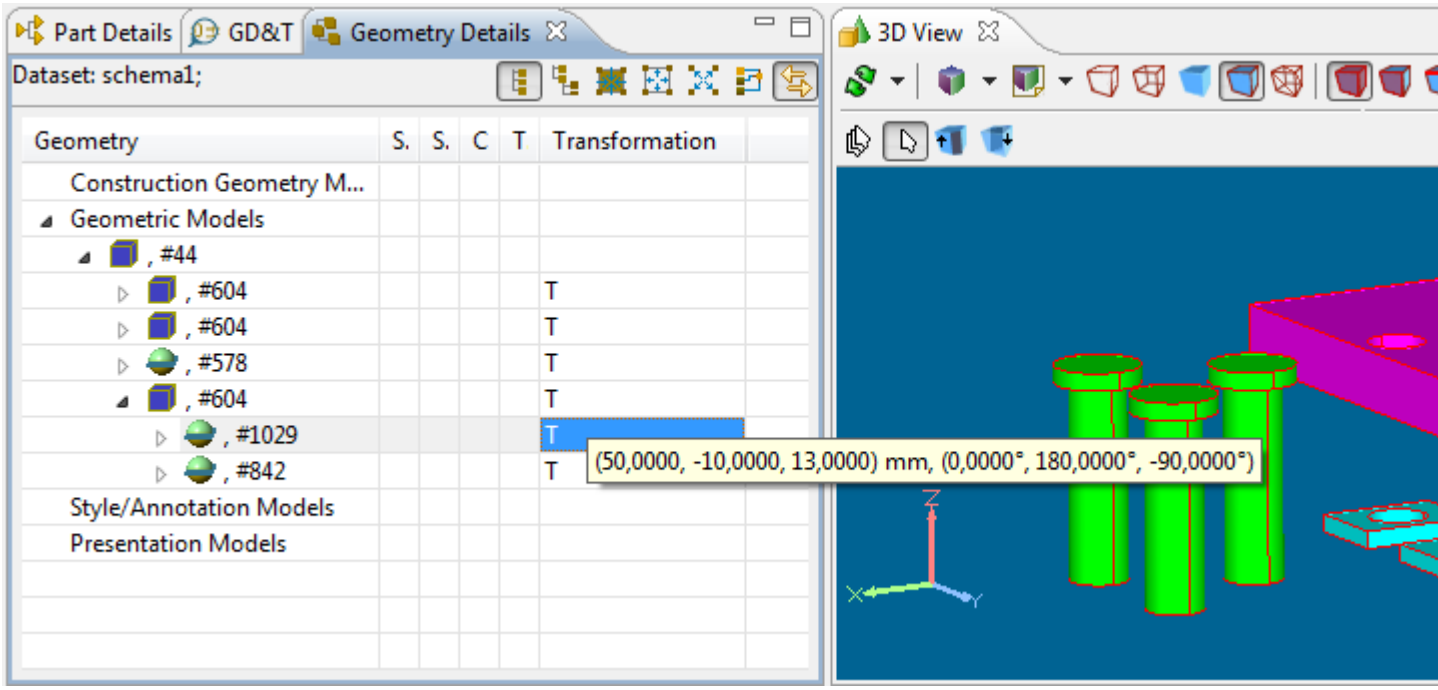

Figure 3. Transformation are changed (tool tip), new transformations applied in 3D View (right side)

To top

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9.9.1. Header View

A file header is the top section of an electronic file which contains a history of all changes made to it (with categories such as Date, Author, Description of Change). The header is updated each time the file is modified.
The **Header** view presents the following information of the STEP file (see Figure 1):

- **Description** - a description of the current file.
- **Change date** - indicates the time when this file was created/modified.
- **Author** - the name and mailing address of the person creating this exchange structure.
- **Organization** - the organization to whom the person belongs to.
- **Preprocessor version** - the name of the system and its version which produces this STEP file.
- **Originating system** - the name of the system and it's version which originally created the information contained in this STEP file.
- **Authorization** - the name and mailing address of the person who authorized this file.
- **Default language** - specifies the assigned a default language for either all or for a specific data section. This is needed for those Express schemas which do not provide the capability to specify in which language string attributes of entities such as name, and description are given.
- **Context identifiers** - specifies additional context information for all or single data sections. This can be used e.g. for STEP-APs to indicate which conformance class is covered by a particular data section.

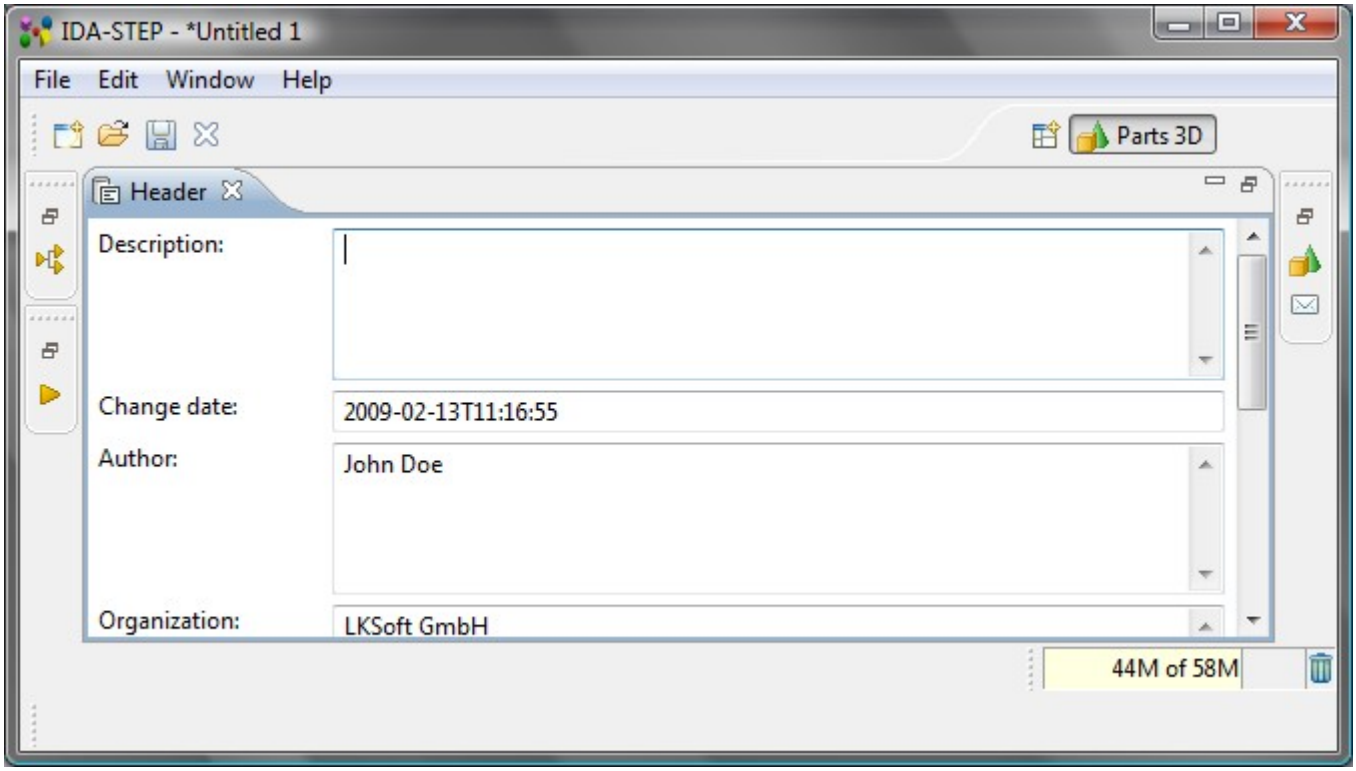

Figure 1. Header view

To top

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9.10.1. Log View

The **Log** view presents messages of IDA-STEP application which are relevant to the user (see Figure 1). Each message is displayed on a separate row, in date and time order (most recent first). The messages may be selected and copied in the **Log** view and then pasted to the preferable text editor.

Figure 1. Log view

Open the **Log** view by selecting in the main menu **Window -> Show View -> Other -> General -> Log**.

To top

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9.11.1. Maintenance Activities View

An *Activity* is the identification of the occurrence of an action that has taken place, is taking place, or is expected to take place in the future.
The **Maintenance Activitie**s view presents a flat list of maintenance activities (See Figure 1). The first column presents the row element - maintenance activity identifier. Other columns contain data of the particular maintenance activity. The default column perspective contain the following columns:

- **Id** - specifies an identifier of the *maintenance activity*.
- **Name** - specifies the word or group of words by which the *maintenance activity* is referred to.
- **Description** - specifies additional information about the *maintenance activity*.
- **Chosen Method** - the task method that specifies the procedure selected to carry out the *maintenance activity*.
- **Subject** - specifies the product as individual, which *Element* is under work.
- **Element** - specifies the piece of product data that is under work.
- **Start Date** - specifies the date when the *maintenance activity* is started.
- **End Date** - specifies the date when the *maintenance activity* is finished.
- **Concerned Organization** - specifies the organization object that is affected by the result of the *maintenance activity*.
- **Supplying Organization** - specifies the person in organization or an organization that carry out the work.
- **Requestor** - specifies the person in organization or an organization that requested the *maintenance activity*.
- **Related Activity** - specifies the related *maintenance activity*.

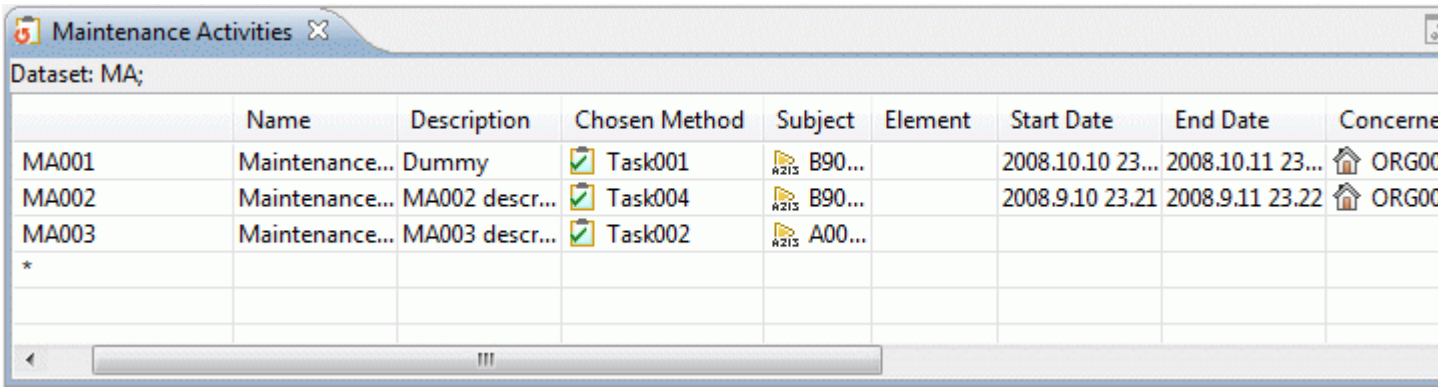

Figure 1. Maintenance Activities view (default columns)

To top

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9.11.2. Creating Maintenance Activity

Follow the routine to create a new maintenance activity element in the **Maintenance Activities** view table list (see Figure 1):

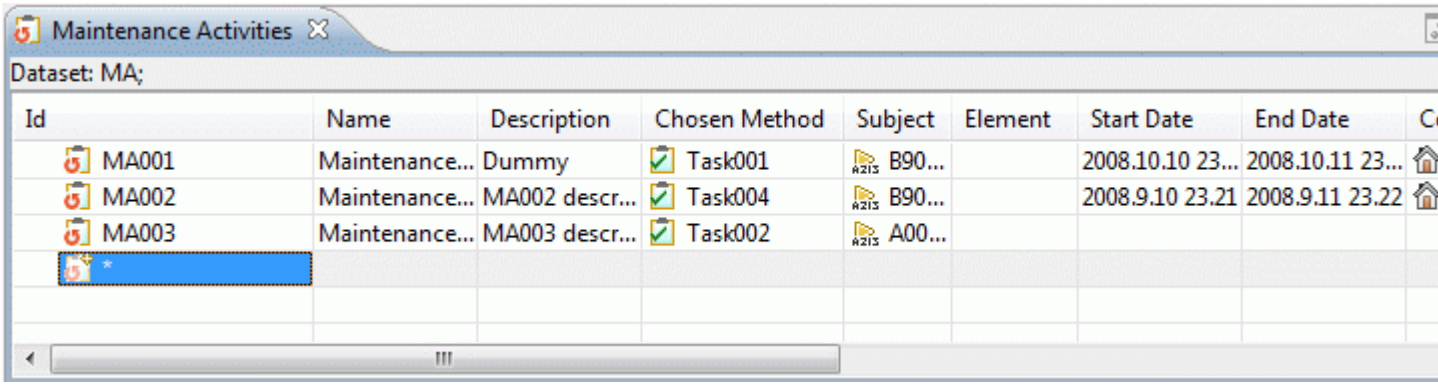

Figure 1. Maintenance Activities view with highlighted auto row

- Select the auto row in the table (see Figure 1 highlighted cell in the **Id** column).
- Enter an identifier of the new maintenance activity element.
- Press **Enter** key. A new maintenance activity element appears in the **Maintenance Activities** view table (see Figure 2).

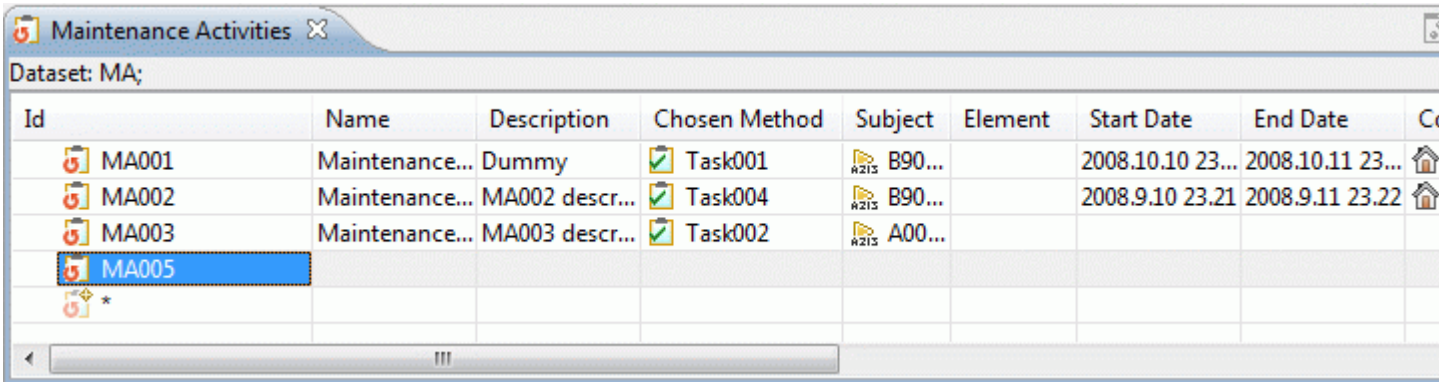

Figure 2. Created maintenance activity (highlighted)

To top

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9.11.3. Editing Maintenance Activity Data

Activate a **Maintenance Activities** view. Select an applicable maintenance activity element in the table list. You can edit the following:

- Enter a new or edit the current maintenance activity identifier in the **Id** column.
- Enter the text, which defines the maintenance activity name into the **Name** column.
- Enter additional information about the maintenance activity in the **Description** column.
- The **Chosen Method** column value can be changed only by copying and pasting a particular task method from the **Tasks** view (if available).
- Select an applicable product as individual element from the drop down list in the **Subject** column or press button to open **Select entity dialog** window to view the whole list of available product as individual elements. The value can be changed also by copying and pasting a particular product as individual element from the **Products** view (if available).
- Select an applicable assembly component, which parent assembly is a design of the *Subject*, from the drop down list in the **Element** column or press button to open the Select entity dialog window to get the whole list of available *elements*. The value can be changed also by copying and pasting a particular assembly component, which parent assembly is a design of the *Subject* from the **Part Details** view (if available).
- Enter a new date and time by typing value or select it from the pop up table in the **Start Date** column.
- Enter a new date and time by typing value or select it from the pop up table in the **End Date** column.
- The **Concerned Organization** column value can be changed only by copying and pasting a particular organization from the **Persons and Organizations** view (if available).
- The **Supplying Organization** column value can be changed only by copying and pasting a particular person in organization or an organization from the **Persons and Organizations** view (if available).
- The **Requestor** column value can be changed only by copying and pasting a particular person in organization or an organization from the **Persons and Organizations** view (if available).
- Enter a new or edit the current value in the **Value** column edit box. The units may be changed by using **Unit wizard**.

To top

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9.11.4. Removing Maintenance Activity

Activate the **Maintenance Activities** view. Follow the routine to remove a particular maintenance activity element form the table:

- Select a maintenance activity element in the table list.
- Right click on the **Id** column.
- Select the **Delete** menu item in the appeared context sensitive menu.

The selected maintenance activity element is removed from the **Maintenance Activities** view table list.

To top

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9.12.1. Part Details View

The **Part Details** view content is dependent on the selection in the **Parts** view. The **Part Details** view presents a hierarchy of part definitions of a part version, selected in the **Parts** view (see Figure 1).
The **Part Details** view title presents the following information:

- **Datasets:** the currently opened dataset;
- **Part version:** the part version, selected in the **Parts** view, details.

Default **Part Details** view columns:

- **Id** - part definition/part definition component identifier. The check box in the front of the first column controls visibility and transparency of an assembly component.
- **Definition** - the definition, which the component is derived from.
- **Transformations** - the transformation data of the part definition component.

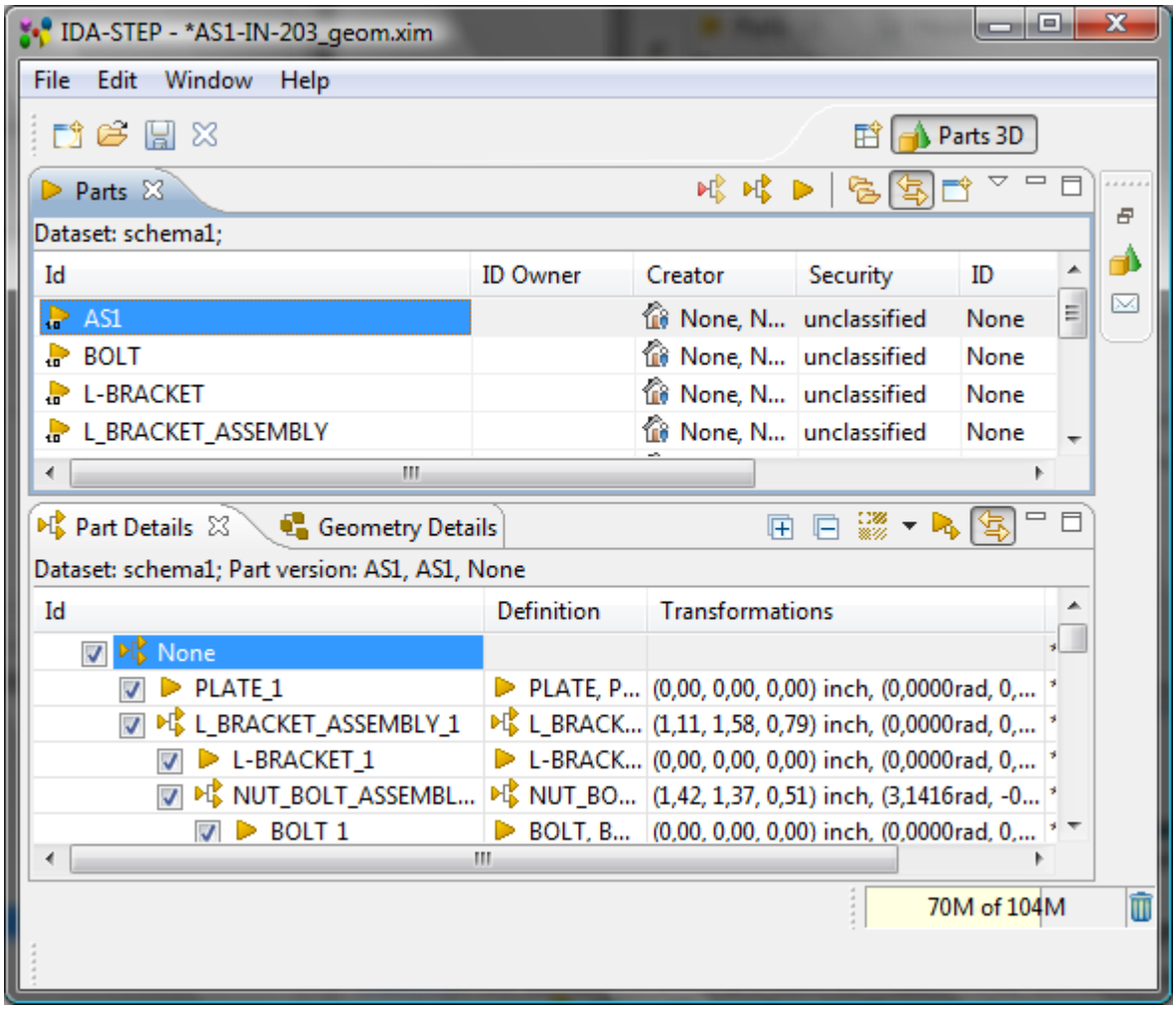

Figure 1. Selected part version in the Parts view and it's definitions in the Part Details view

If the opened XIM or STEP file contains PCA design data, the **Part Details** view may display electronical components with terminals (see Figure 2). By default, **Part Details** view shows only the flat list of electronical components. To view related electronical component terminals, press **Show Shape Aspects** button in the view tool bar.

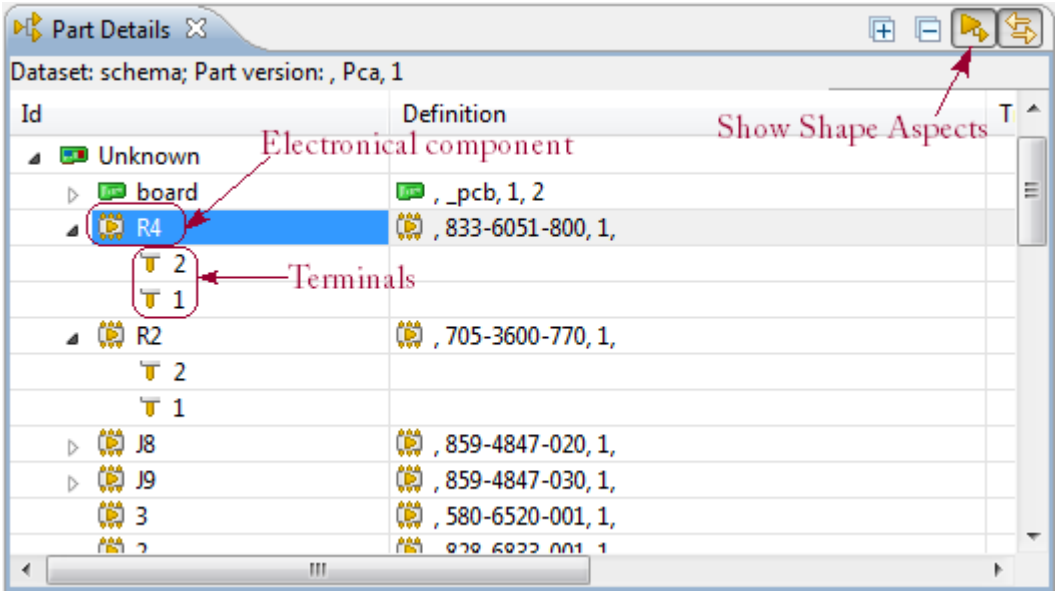

Figure 2. Part Details view with electronical component terminals

The **Part Details** view is is capable to display following additional element types, if the special **Show Shape Aspects** button is pressed in the **Part Details** view tool bar (see Figure 3):

- shape aspect (datums, datum features, composite shape aspects and derived shape aspects);
- dimensions (linear distance, diameter size etc.);
- geometric tolerances (perpendicularity, position etc.).

Note. This feature is experimental and may be unstable. It can be changed/removed in upcoming releases.

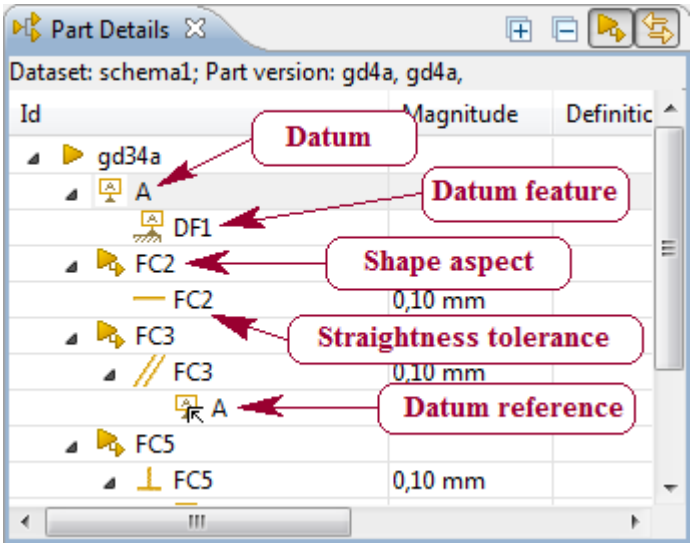

Figure 3. Part Details view with shape aspects

A list of possible node types:

- shape aspect,
- angularity tolerance,
- circular runout tolerance,
- cylindricity tolerance,

- concentricity tolerance,
- datum feature object,
- datum object,
- datum reference,
- dimensional size object,
- flatness tolerance,
- geometric tolerance object,
- line profile tolerance object,
- linear dimension object,
- parallelism tolerance object,
- perpendicular object,
- position tolerance,
- roundness tolerance object,
- symmetry tolerance object,
- straightness tolerance object,
- stratum profile tolerance object,
- total runout tolerance.

Tool tips

The **Part Details** view has a tool tip over each node. Component nodes (and transformations) have a single line tool tip that shows just the text, which appears in the **ID** column. All other nodes (presumably related to shape aspects) have two-line tool tips:

- the first line shows a type, a name and a hash code;
- the second line shows if any geometry is associated with the node (i.e., if clicking on the node would result in any selection/highlight event in **Geometry Details** view or **3D view**). The second line reads "Has geometry:" and the possible values are "YES" or "NO".

Tool bar

The table below presents the available tool bar buttons in the **Part Details** view.

Button icon	Description	
	Expand All	Expand all tree nodes.
	Collapse All	Collapse all tree nodes.
	Current action	Determine the kind of actions, either to make invisible or to set transparency to some level. The predefined values are: invisible, Transparent by 25 %, Transparent by 50 %, and Transparent by 75 %. The default value is invisible.
	Show Shape Aspects	see Figure 2
	Link with	When this option is turned *On*, the selection in the **Part Details** view is represented in the **3D View** and vice versa.

	Other Views.	

To top

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9.12.2. Selection

The selection of some part or assembly in the **Parts Details** view initiates the highlighting of corresponding geometry objects in the **3D View** and related geometry elements in the **Geometry Details** view (see Figure 1).

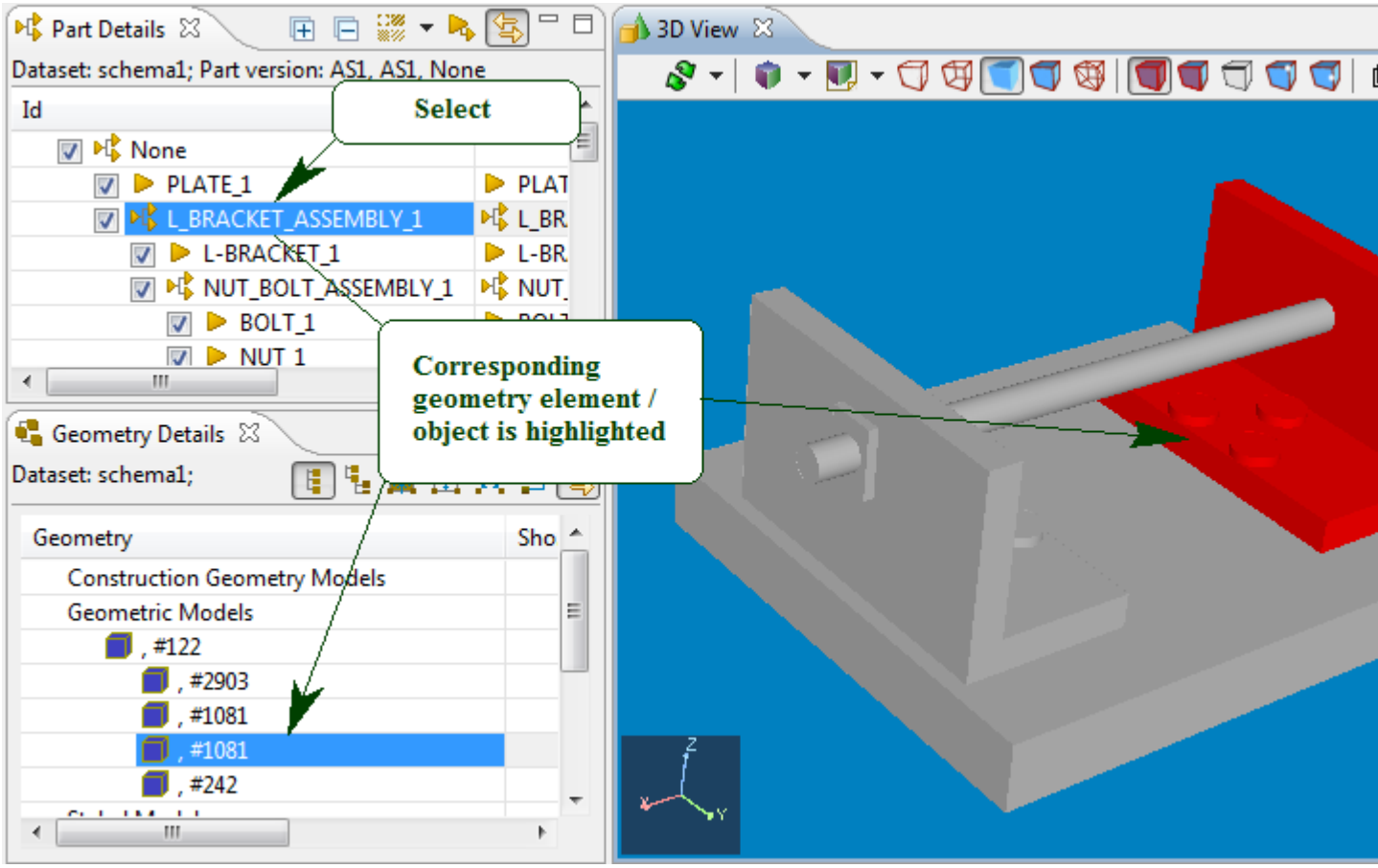

Figure 1. Selected part in Part Details view triggers highlighting of corresponding geometry elements in other views

If a part is modeled by using feature based paradigm, an applicable shape aspects can be selected in the **Parts Details** view and then cross-highlighted in other views (see Figure 2).

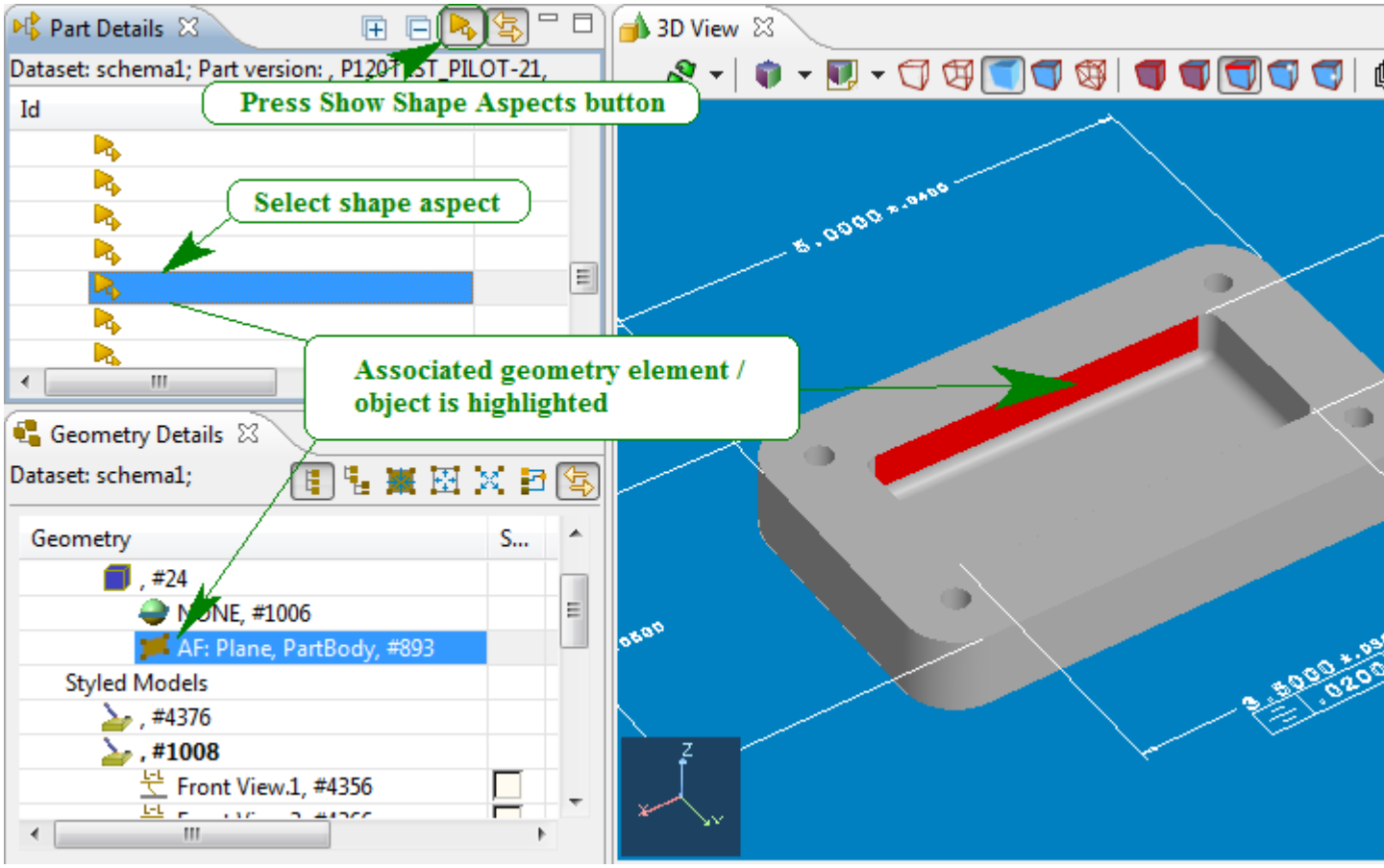

Figure 2. Selected shape aspect in Part details view initiates highlighting of associated geometry element/object in other views

To top

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9.12.3. Visibility and Transparency

With the **Viewer Pro 3D feature** you can control the visibility and transparency of assembly components. For this a check box shows up in each line of the assembly tree together with a combo box in the tool bar. If the check box is set, the corresponding assembly component is visible. If it is unchecked, the corresponding assembly component get either invisible or transparent dependent on the state of the **Visibility** combo box (see Figure 1).
Note. This capability is only available if the **Show Shape Aspect** button in the tool bar of the **Part Details** view is not pressed.

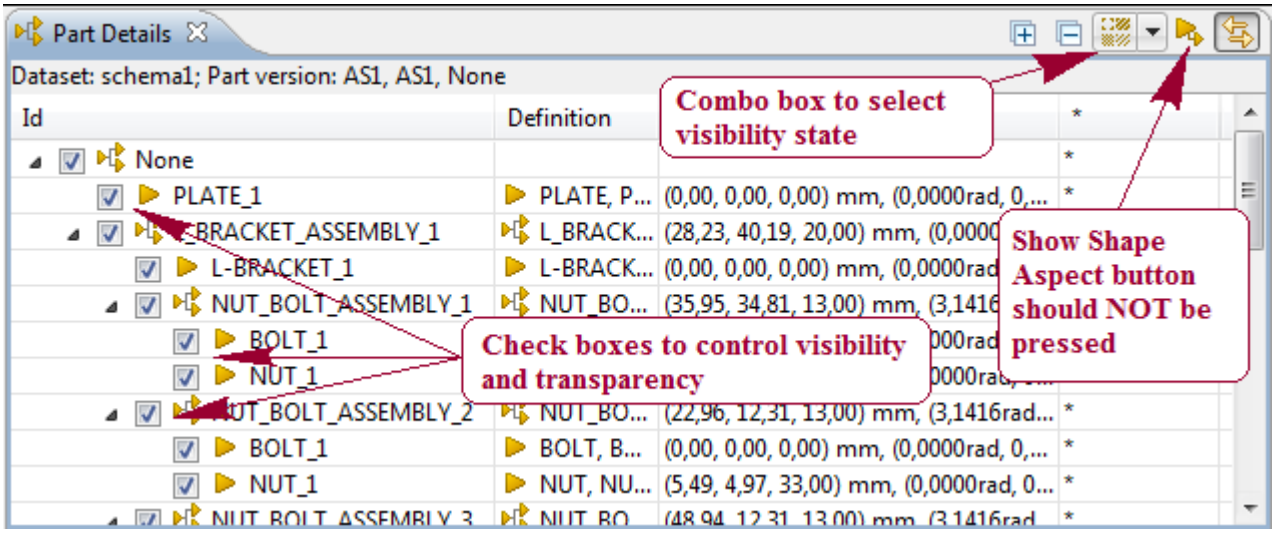

Figure 1. Check box to control visibility and transparency and button to select action type

The **Visibility** combo box can have one of these states:

- Invisible (this value is the default value),
- Transparency by 25%,
- Transparency by 50%,
- Transparency by 75%.

Changing the value of the **Visibility** combo box has by itself no effect. Only when un-setting one of the check boxes the actual state of the combo box is copied for the affected assembly component. So it is possible to make some components invisible and have other components on different levels of visibility. In principle visibility and transparency of assembly components can also be controlled in the **Geometry Details** view, but controlling them directly in the assembly tree is typically much easier.

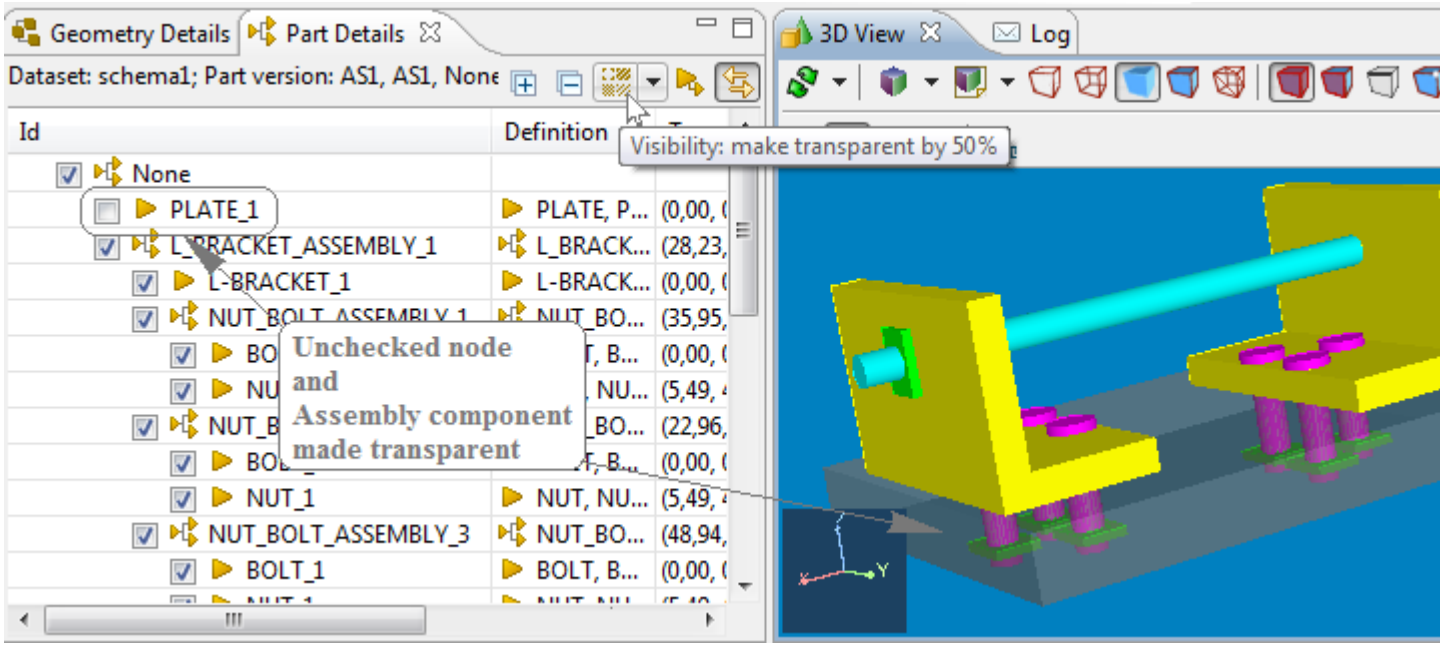

Figure 2. Example on how to make an assembly component transparent

Figure 2 shows an example on how to make an assembly component transparent. First set the visibility combo box to 50% transparency and then unset the check box of an assembly component.

When making an intermediate assembly component that is itself an assembly invisible or transparent, the whole subtree inherits this information. But you can nevertheless change the state of it's leaves afterwards.

Note. The visibility/transparency is directly coupled with **Geometric Details** view, independent on whether the user has opened it or not.

To top

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9.12.4. Editing Part Details

Activate the **Part Details** view to edit data. You can make the following actions for the selected object in the tree table:

- **Product definition** - the root node in the **Part details** view:
  - Enter a new or edit the current identifier in the **Id** column (see Figure 1).

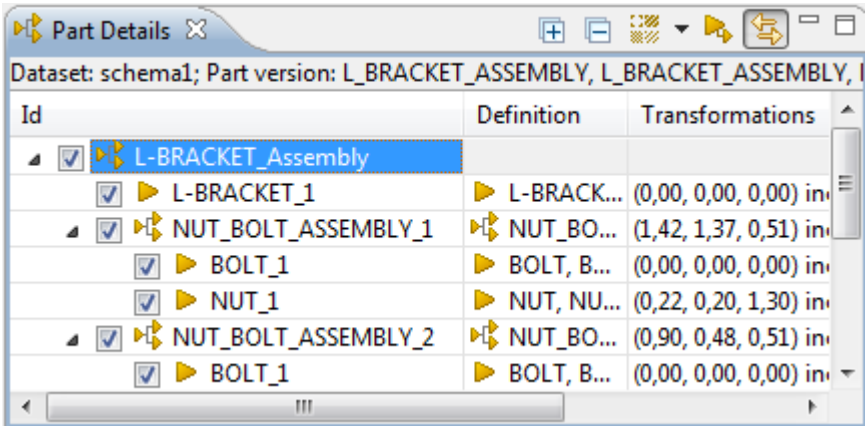

Figure 1. Part Details view, selected root node in Id column

- **Product definition component** - the component beneath the root node (i.e. second, third and deeper level node):
  - Enter a new or edit the current identifier in the **Id** column.
  - Select a new definition for the component in the **Definition** column:
    - Select an applicable item from the drop down list.

    Or

    - Press the button next to the text field. The **Select entity** dialog window appears on the screen (see Figure 2):

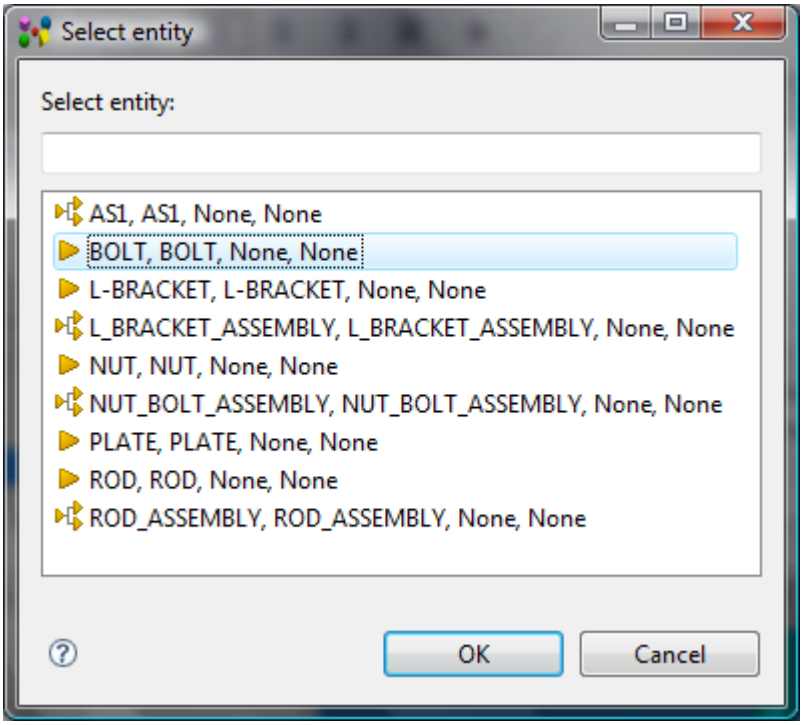

Figure 2. Select entity dialog window

- Select an applicable entity in the list. You can use the filter on the top section of the window to filter out particular entities.
- Press **OK** button. The selected entity data appears in the **Definitions** column. The particular 3D object appears in the **3D View**.

- Edit component transformation data in the **Transformations** column:
  - Click on the cell in the **Transformations** column. The **3D Transformation editor** appears on the screen to change transformation of a 3D object if needed (see Figure 3):

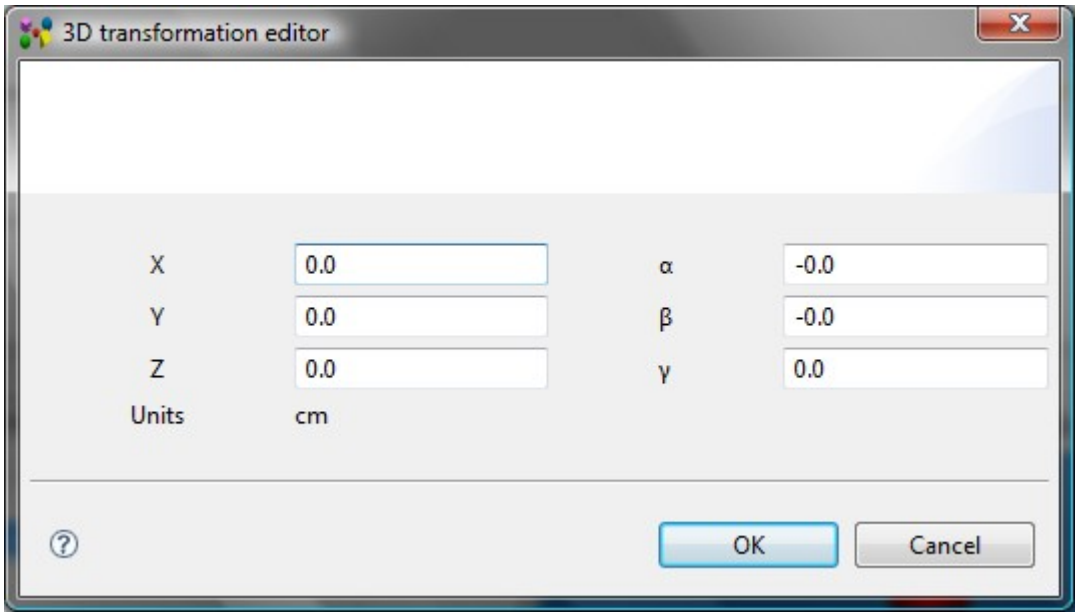

Figure 3. 3D transformation editor

- Enter a number value into the **X** field to change the translation along X axis;
- Enter a number value into the **Y** field to change the translation along Y axis;
- Enter a number value into the **Z** field to change the translation along Z axis;
- Enter a number value into the **α** field to change the rotation around X axis;
- Enter a number value into the **β** field to change the rotation around Y axis;
- Enter a number value into the **γ** field to change the rotation around Z axis.

- Press **OK** button to confirm your changes. The transformation data appears in the **Transformations** column.
  Note. Units in the column are converted to the units, defined in the **Quantity and Units** preferences (**Window -> Preferences -> Numbers -> Quantities and Units**)

To top

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9.12.5. Create Assembly from Parts

IDA-STEP allows user to select multiple parts from the file and then create assembly from them by creating component from each part with zero transformation in the new assembly.

Follow the routine to create an assembly from selected parts:

- Press **Create assembly from parts...** button in IDA-STEP main tool bar.
- The Create assembly dialogue window appears on the screen (see Figure 1):
  - Enter an assembly identifier into the **Assembly ID** text field. By default the text *newID* is entered into this field.
  - Select an applicable parts to be included into the assembly by pressing CTRL + selecting parts. By default all parts are selected.
  - Press **OK** button to create an assembly.

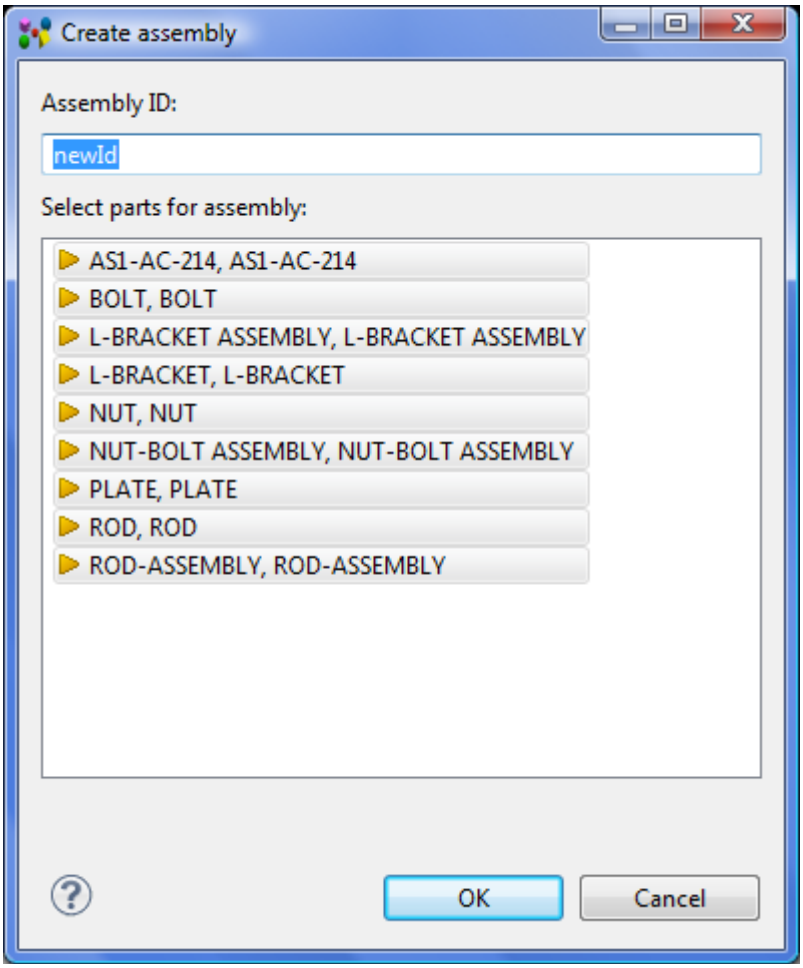

Figure 1. Create assembly page

To top

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9.12.6. Adding Component

The **Part Details** view content is dependent on the selection in the **Parts** view. If the selected part version in **Parts** view has a part definition, it is presented as a root node in the **Part Details** view. In other case, you have to create a part definition as a root node to be able to add new components. Follow the routine to add a root node in the **Part Details** view:

- Select an autorow in the **Part Details** view (see Figure 1).

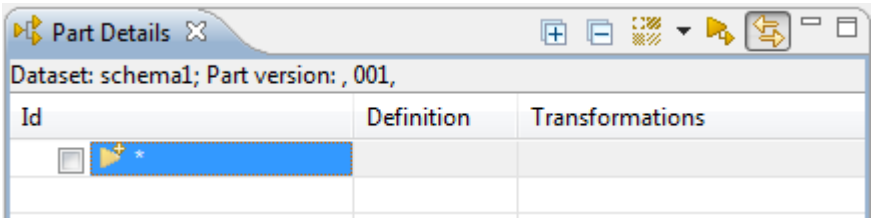

Figure 1. Part Details view, autorow

- Enter an applicable identifier for a part definition into **Id** column and press **Enter** key. The root node is created (see Figure 2).

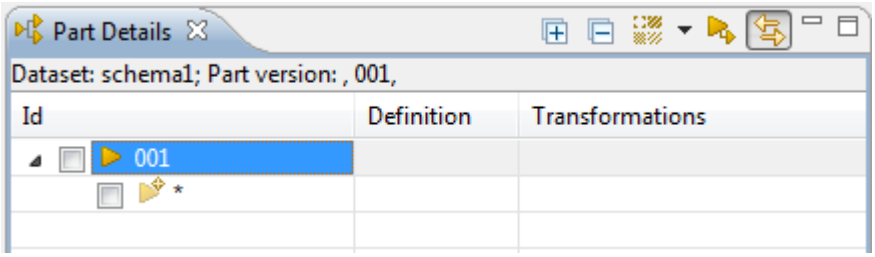

Figure 2. Created root node

Now you can add components to the part definition.

- Right click on the root node. Select **Add component** menu item in the appeared context sensitive menu (see Figure 3) or select the autorow node under the component you are going to add a new component to.

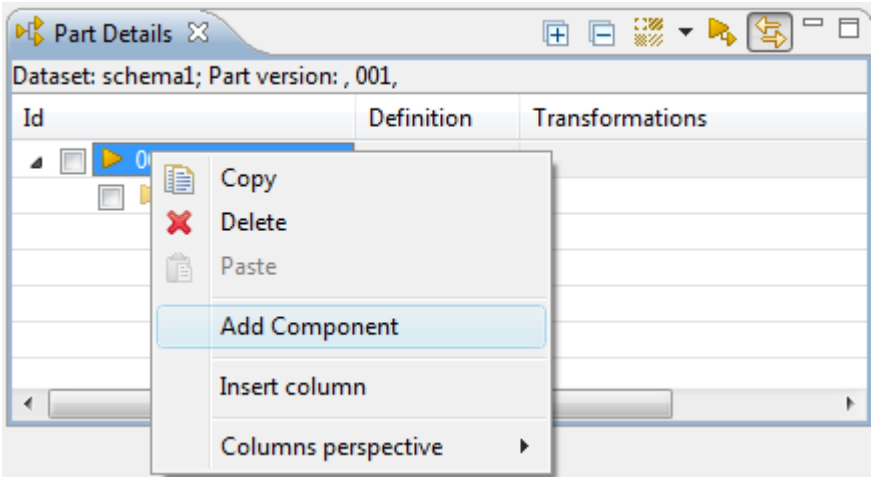

Figure 3. Context sensitive menu with selected Add component menu item

- Enter an identifier of a new component into **Id** column (see Figure 4) and press **Enter** key.

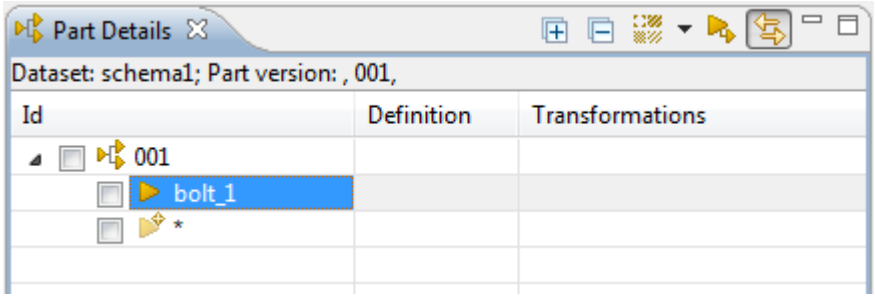

Figure 4. Created new component

- Select a definition for the component in the **Definition** column:
  - Select an applicable item from the drop down list.

Or

  - Press the button next to the text field. The **Select entity** dialog window appears on the screen (see Figure 5):

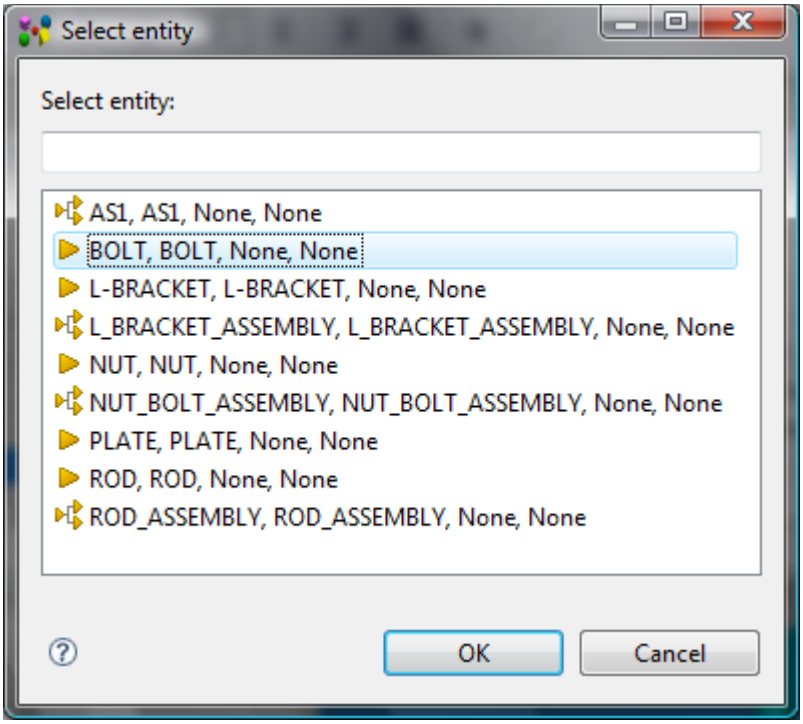

Figure 5. Select entity dialog window

- Select an applicable entity in the list. You can use the filter on the top section of the window to filter out particular entities.
- Press **OK** button. The selected entity data appears in the **Definitions** column (see on the left side of Figure 6). The particular 3D object appears in the **3D View** (see on the right side of Figure 6).

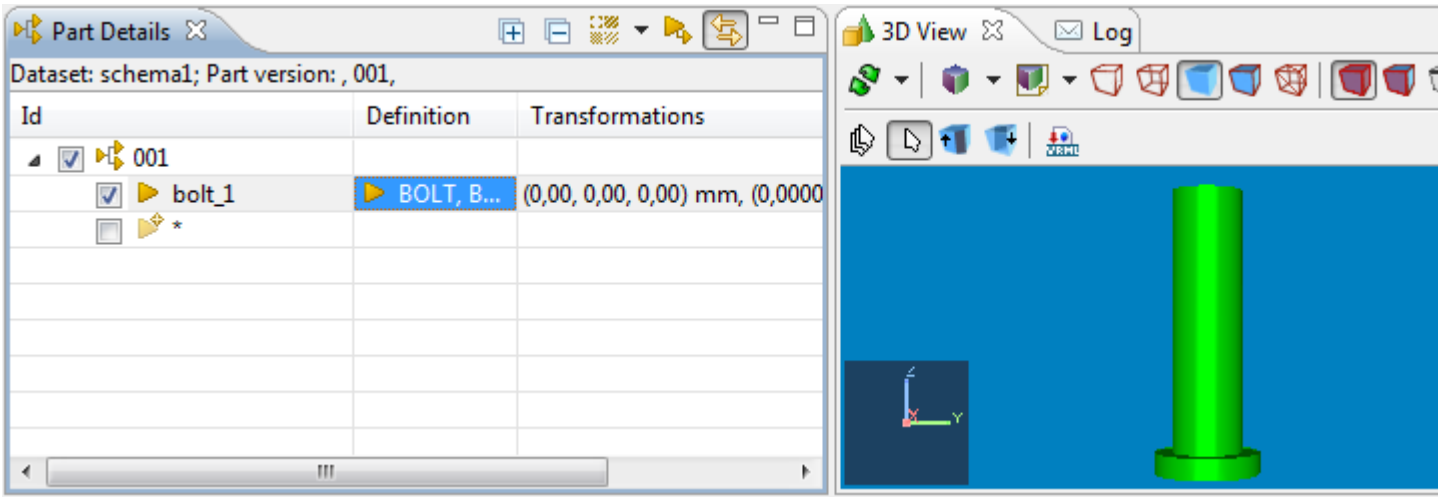

Figure 6. Selected definition in the definition column

- Edit component transformation data in the **Transformations** column:
  - Double click on the cell in the **Transformations** column. The **3D Transformation editor** appears on the screen to change transformation of a 3D object if needed (see Figure 7):

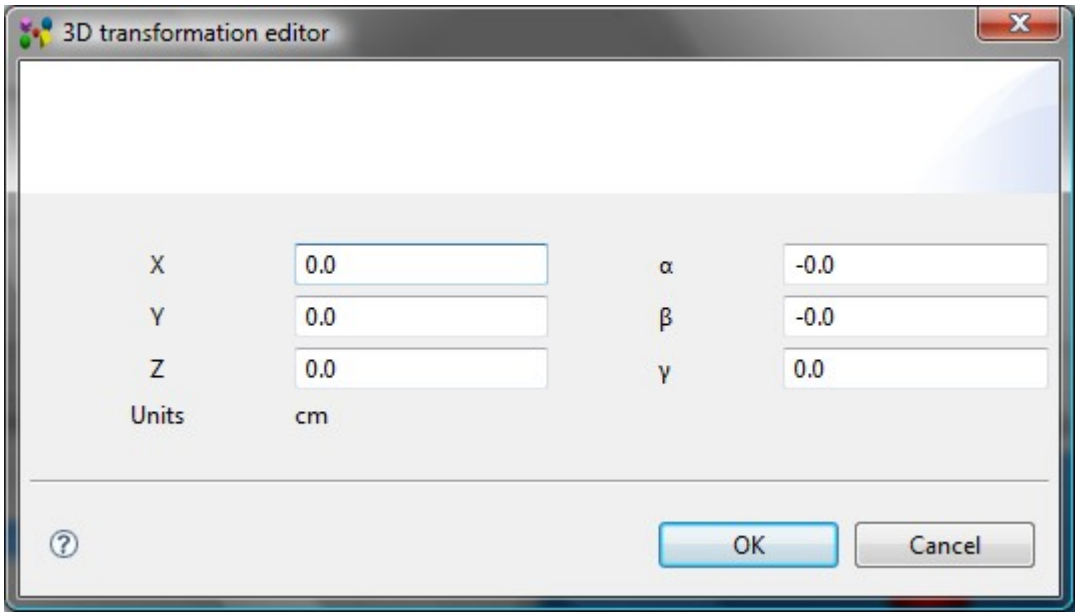

Figure 7. 3D transformation editor

    - **X** - translation along X axis;
    - **Y**- translation along Y axis;
    - **Z** - translation along Z axis;
    - **Units** - measure units (measure units, assigned in STEP file);
    - **α** - rotation around X axis;
    - **β** - rotation around Y axis;
    - **γ** - rotation around Z axis.
      Note. Rotation units are degrees (º).
  - Press **OK** button to confirm your changes. The transformation data appears in the **Transformations** column.
    Note. Units in the column are converted to the units, defined in the **Quantity and Units** preferences (**Window -> Preferences -> Numbers -> Quantities and Units**).

The **Add component** function can be used not only for adding components for the root part definition. You can add additional components for an applicable part definition component (see Figure 8).

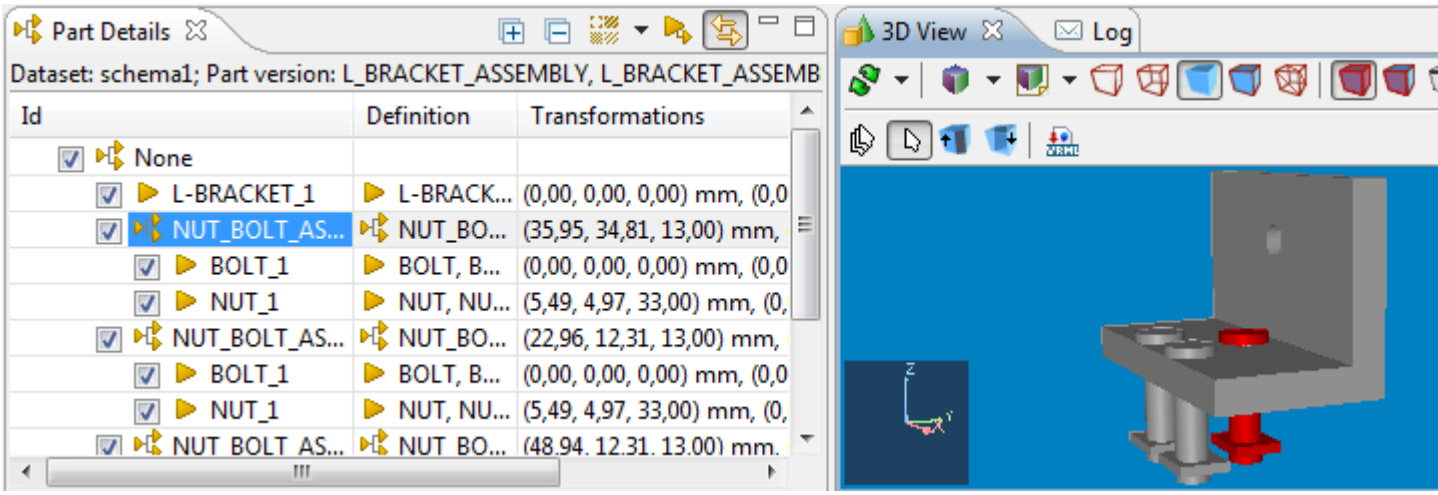

Figure 8. Assembly component (i.e. 'NUT_BOLT..'), selected in Part Details view and highlighted in 3D View

To top

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9.12.7. Add 3D Part to Library

IDA-STEP presents possibility to add 3D parts to the 3D Parts Library (see Manage 3D Part Libraries). To populate 3D parts library:

- Select a 3D part/assembly entry (top node) in the **Part Details** view table.
- Right click and select the **Add 3D Part to Library** item in the appeared context sensitive menu. The **Import 3D Part** wizard page appears on the screen (see Figure 1) with the 3D Part library hierarchy (libraries with categories, situated beneath):

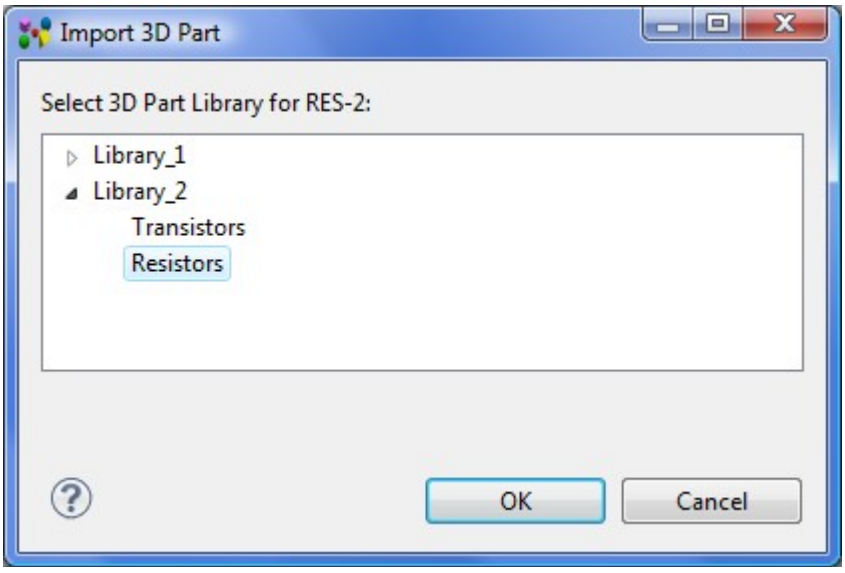

Figure 1. Import 3D Part wizard page with library hierarchy

- Expand an applicable tree branch, which presents a library (top node). In case the part should not be classified, select library name, otherwise select a category name.
- Press the **OK** button to confirm the selection and close the wizard page. The selected part is added to the library. To view results use the main menu **File -> Manage libraries -> 3D Part Libraries...**.

See also 3D Part Libraries.

To top

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9.12.8. Copying and Pasting

There are the following Copy / Pasting possibilities in the **Part Details** view:

- Copy/Paste part definition
- Copy/Past part definition component.

Note. IDA-STEP presents possibility to copy part definition or component in the **Part Details** view in one STEP file and paste into another STEP file.

Copy/Paste part definition

Follow the routine to copy/paste the root node in the tree table of the **Part Details** view:

- Select the root node.
- Right click and select **Copy** menu item in the appeared context sensitive menu or press **CTRL + C** keys.
- Select the autorow as a root node in the tree table. If the autorow is not in the root branch, right click on the autorow and select **Move to root** menu item in the appeared context sensitive menu.
- Right click and select **Paste** menu item in the context sensitive menu or press **CTRL + V keys**. The copy of the selected part definition appears in the tree table (see Figure 1).

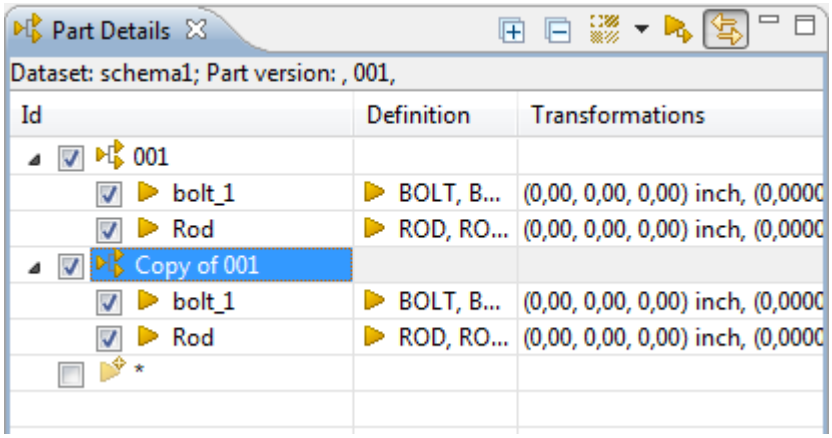

Figure 1. Copy of part definition in Part Details view

To top

Copy/Paste part definition component

Follow the routine to copy/paste a part definition component in the **Part Details** view.

- Select an applicable component.
- Right click and select **Copy** menu item in the appeared context sensitive menu or press **CTRL + C** keys.

- Right click on the destination object (part definition or part definition component) in the tree table.
- Select **Paste** menu item in the context sensitive menu or press **CTRL + V keys**. The copy of the selected component appears in the tree table (see Figure 2).

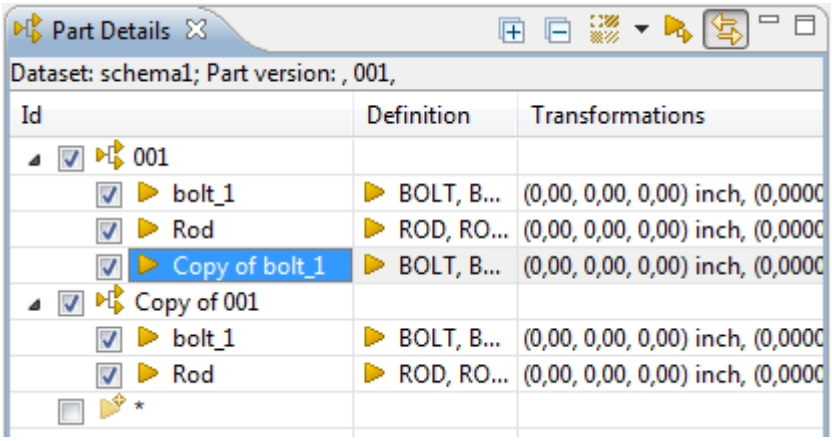

Figure 2. Copy of part definition component

To top

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9.12.9. Removing

To remove a redundant or incorrect product definition or product definition component from the tree table in the **Part Details** view:

- Select an object you want to remove.
- Right click and select **Delete** menu item in the context sensitive menu or press **Delete** key. The selected object is removed from the tree table.

To top

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9.13.1. Parts View

The **Parts** view presents the table list of part versions, available in the currently opened dataset (see Figure 1). The active dataset name is presented in the **Parts** view title.
The default columns, presented in the **Parts** view:

- **Id** - an identifier of a part version;
- **Name** - a word or words by which a part version is designated and distinguished from others;
- **Version** - a part version designation;
- **ID Owner** -an organization which has legal ownership of a part version and has the right to use it and has full control over it;
- **Creator** - an organization, which is a creator of the part version;
- **Security** - the level of confidentiality that is required in order to protect part version data against unauthorized usage;
- **<class attributes>** - the columns appears if the list of part versions belong to the classification, where some class attributes are created.

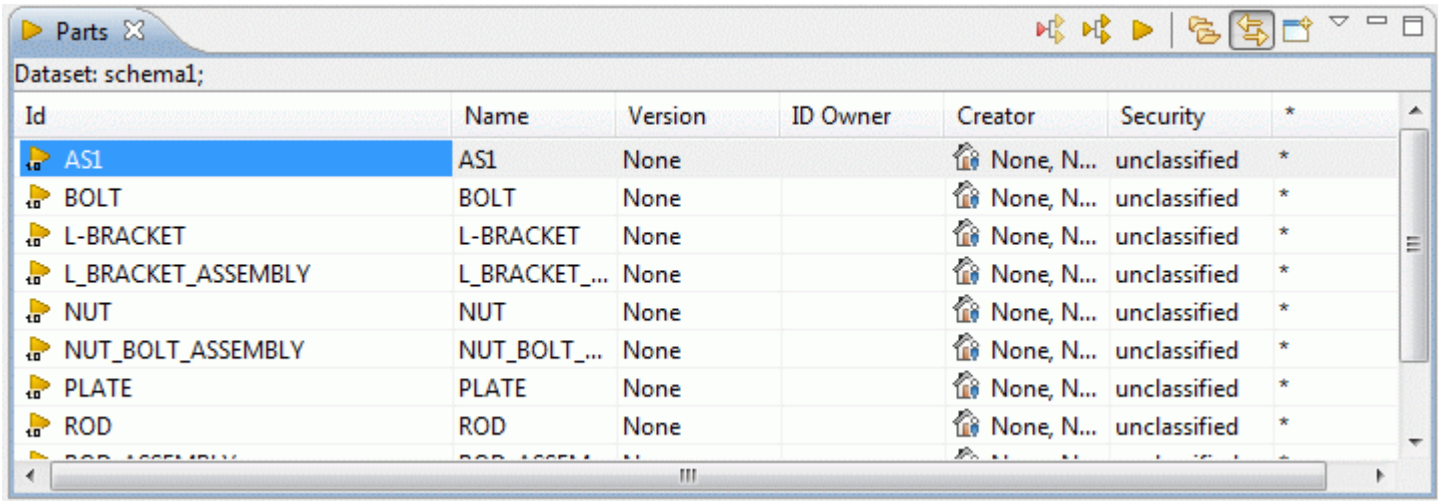


Figure 1. Parts view

By default, the **Parts** view presents only the table list of part versions, but if you press **Show classifications** button, a hierarchy of classifications appears on the left side of the window (see Figure 2).

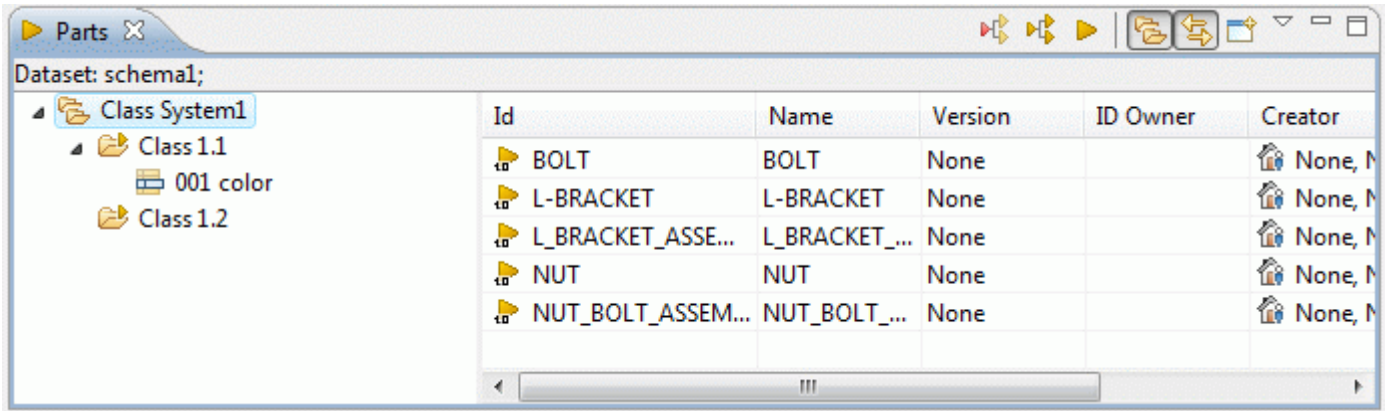
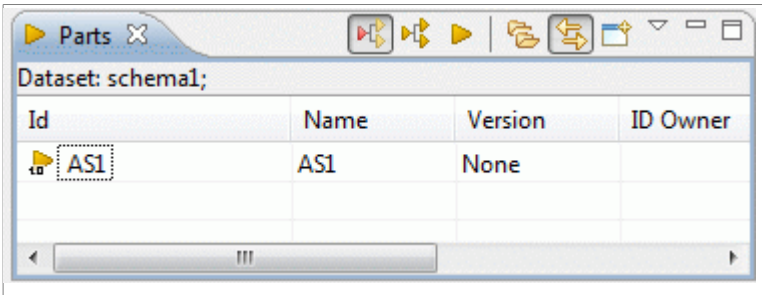
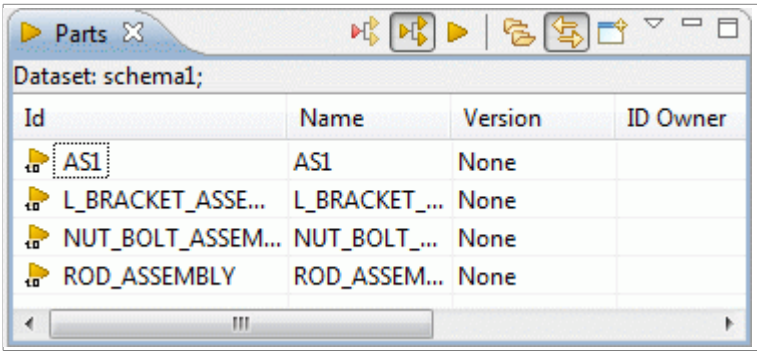
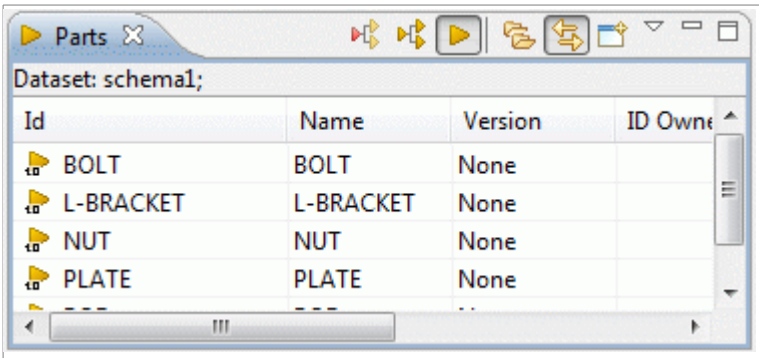
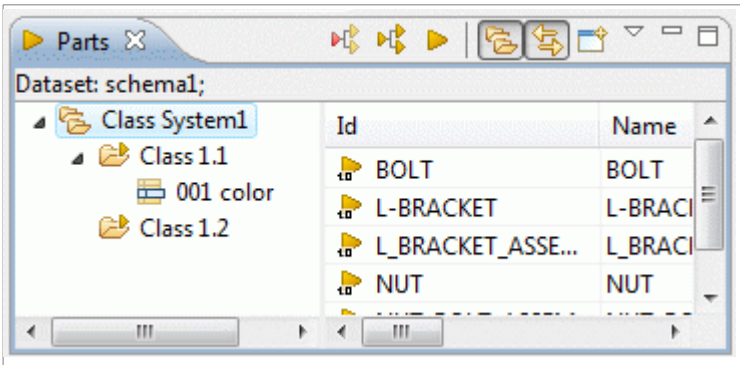


Figure 2. Parts view with hierarchy of classifications

Tool bar

Button Icon	Description
	Display only root parts 
	Display only assembly parts 

	Display only piece parts	
	Show classifications	
	Link with Other Views	When this option is turned *On*, the **Part Details** view presents a hierarchy of part definitions of the part version selected in the **Parts** view.
	Open a new window of the active view	

To top

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9.13.2. Part Classifications

A *classification* in is a systematic arrangement of parts in groups or categories according to criteria. Each classification may have sub-classification and classification attributes, which appear as an additional column in a table list of parts. Classifications are grouped into a particular classification system.

The **Parts** view presents a hierarchy of part classification on the left side of the window and a table list of available part versions on the right side of the window, after the **Show Classifications** button is switched *On* (see Figure 1).

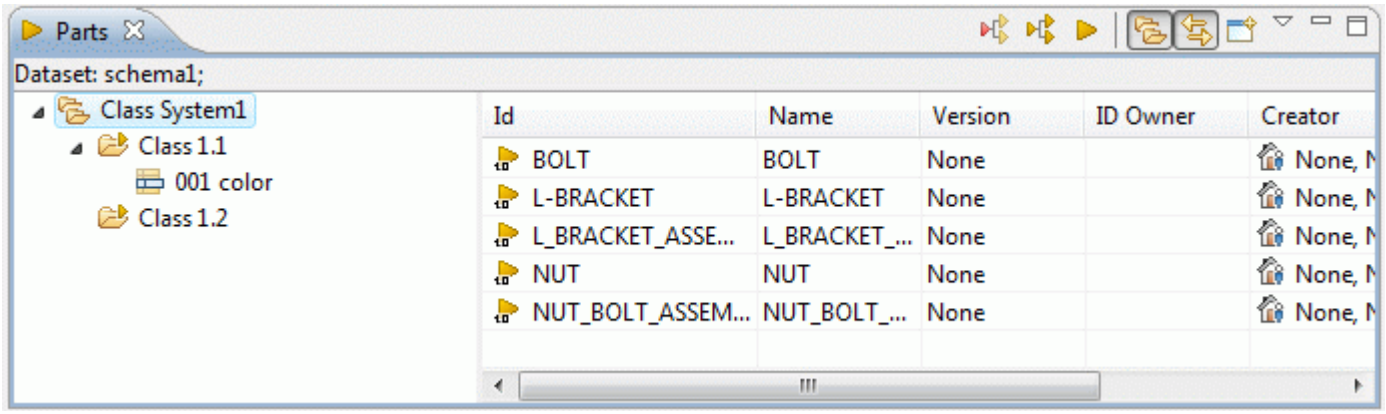

Figure 1. Part Classification window

To top

Classification Hierarchy

A *classification hierarchy* presents a classification relationship between class as a member, the specification of all the members of the class (see Figure 1). The classification hierarchy is visible in the **Parts** view only if the special button, i.e. **Show Classifications** is switched *On*.
Click on an applicable classification in the tree to view associated sub-classifications and classified part versions in the table list:

- Click on a parent classification to view all part versions, associated to this classification and sub-classifications, situated beneath (see Figure 2).

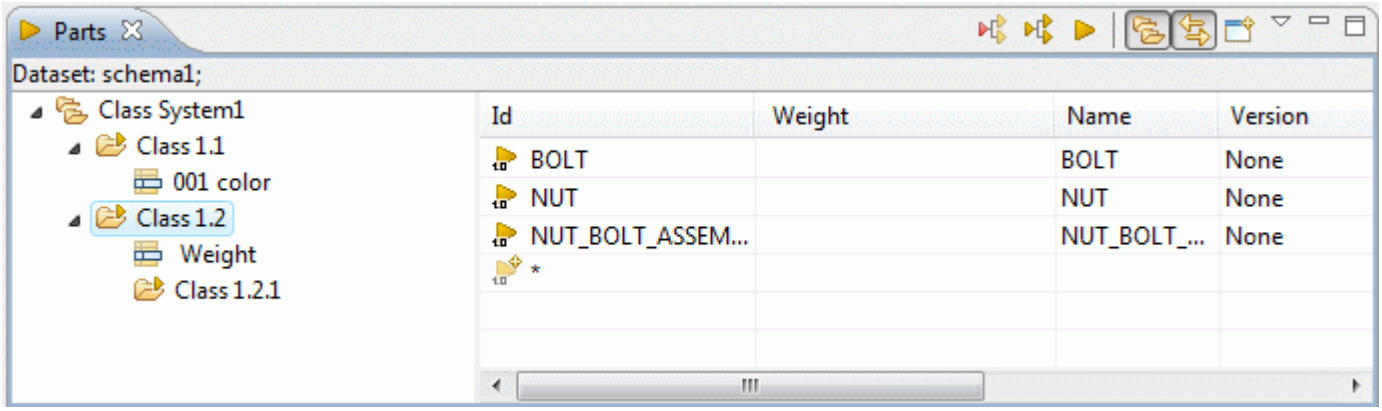

Figure 2. Parent classification 'tools' presents parts 'stick' and 'tip' which belong to 'special' classification

- Click on a "leaf" classification - the table list presents only the list of part versions that are associated to this "leaf" classification.
- You can view the list of all part versions (classified and not classified) after you unselect all nodes in the hierarchy by mouse click on the empty tree area.

To top

Class Attributes

A *classification attribute* specifies an assignment to product data.
The class attributes may be situated under an applicable classification branch in the classification hierarchy:

- Click on an applicable classification to view associated class attributes under the branch. Also you can see all associated class attributes in the table list as separate columns (see Figure 3).

Note 1. The table list shows columns of those class attributes, which are associated to the selected classification. The class attributes of sub classification do not appear in the columns (see Figure 2).

Note 2. A sub-classification inherits the class attributes from the parent classification.

- Click on a class attribute to view a page with the class attribute details (see Figure 4).

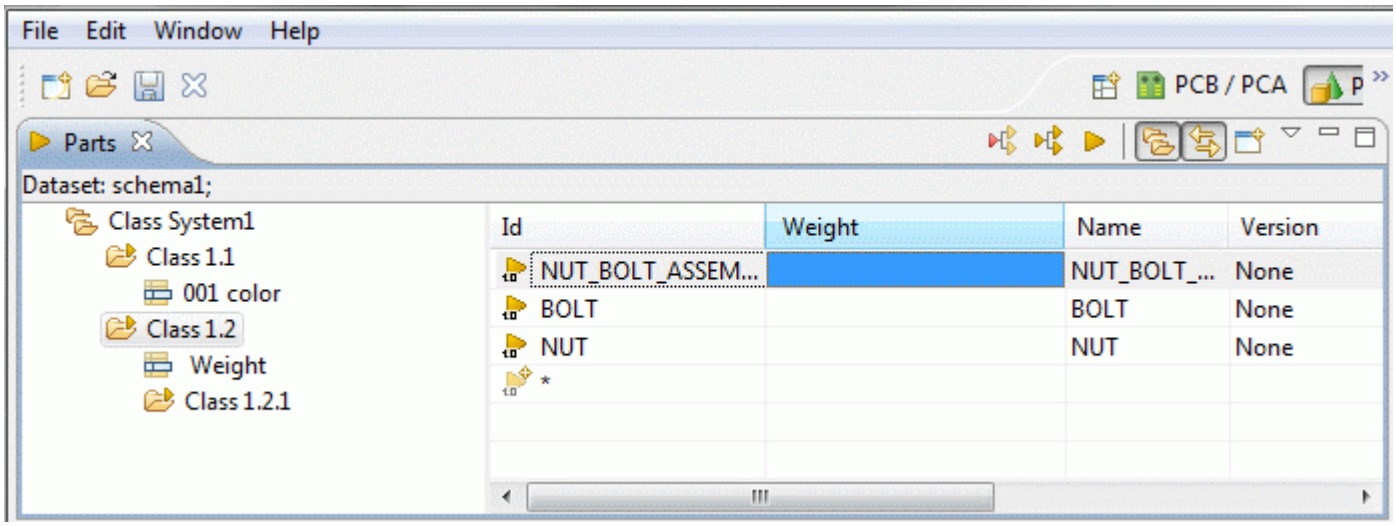

Figure 3. Class attribute 'weight' as a column 'weight' in the table list

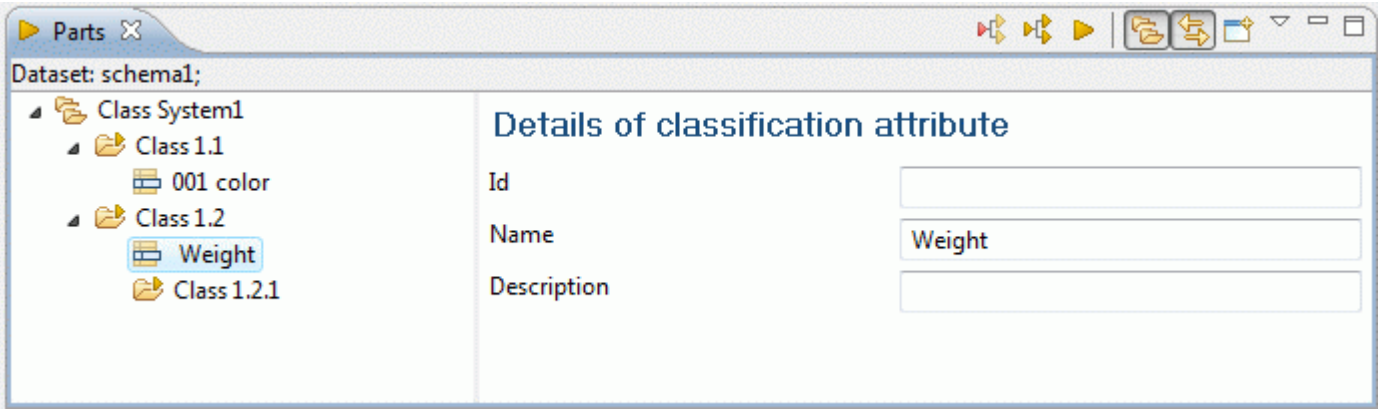

Figure 4. Selected class attribute in a hierarchy (left) and Details of class attribute (right)

To top

List of Parts

The table list on the right side of the **Part Classification** window presents the list of available parts with their attributes. Each part may have several versions, that are identified in the **Version** column. The part version is a subject to a part. A new part version will be assigned automatically to the classification the previous version was assigned to. The part is removed from the dataset after all version of this part are removed. If you move one part version to a particular classification - all part versions will be moved together.

If you click on a particular part in a table list, the classification (all classifications), for which the part is assigned to, is highlighted in the tree (see Figure 5). If you click on a particular classification node, you can view the list of parts, associated to this classification.

The list of all parts, which are classified and not, appears if no nodes are selected in the tree. If any node is selected, you can unselect it by mouse click on the empty tree area.

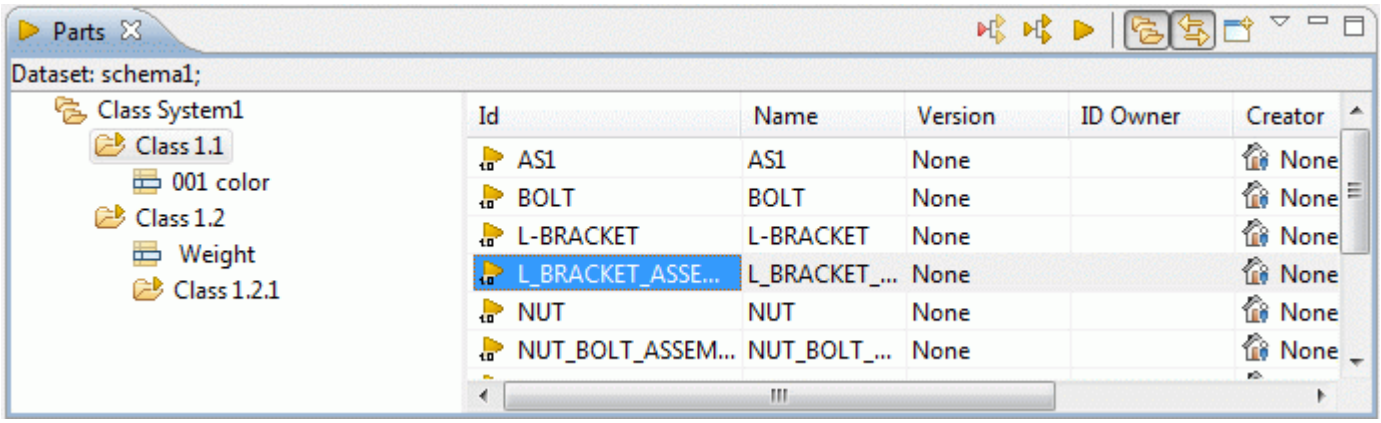

Figure 5. Part 'paint' belongs to 'materials' classification

To top

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9.13.3. Creating Part Version

You have possibility to create:

- unclassified part version, which does not belong to any classification

Or

- classified part version, which is associated to a particular classification while creation process.

Creating Unclassified Part Version

Follow the routine to create a part version:

- Unselect selected nodes in a part classification hierarchy by mouse click on the empty tree area. The list of available part versions appears in the table.
- Find the autorow with an asterisk at the end of the list (see Figure 1).
- Enter data into columns: **Id**, **Name**, **Version** and classification attribute if any.

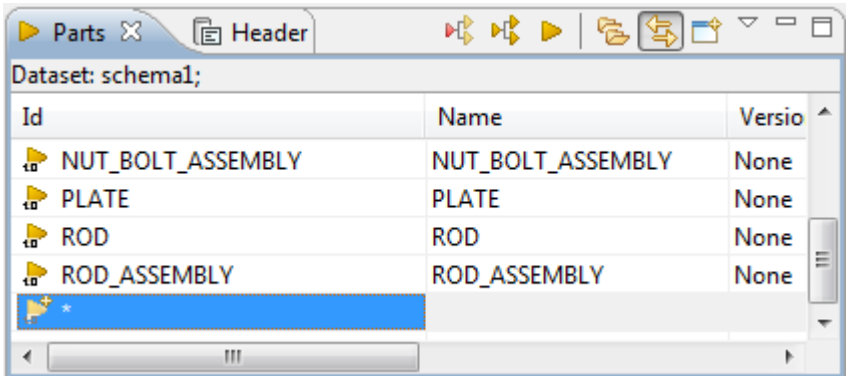

Figure 1. Row with an asterisk

To top

Creating Classified Part Version

Follow the routine to create a part version for the selected classification:

- Click on an applicable classification node in a hierarchy. The table with a list of part versions appears on the right side of the window.
- Find the autorow with asterisk at the end of the list (see Figure 1).
- Enter data into columns: **Id**, **Name**, **Version** and classification attribute if any.

To top

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9.13.4. Creating New Version

Follow the routine to create a new version of a part version:

- Select a part version in a table list.
- Right click on the selected part version to open a context sensitive menu (see Figure 1).

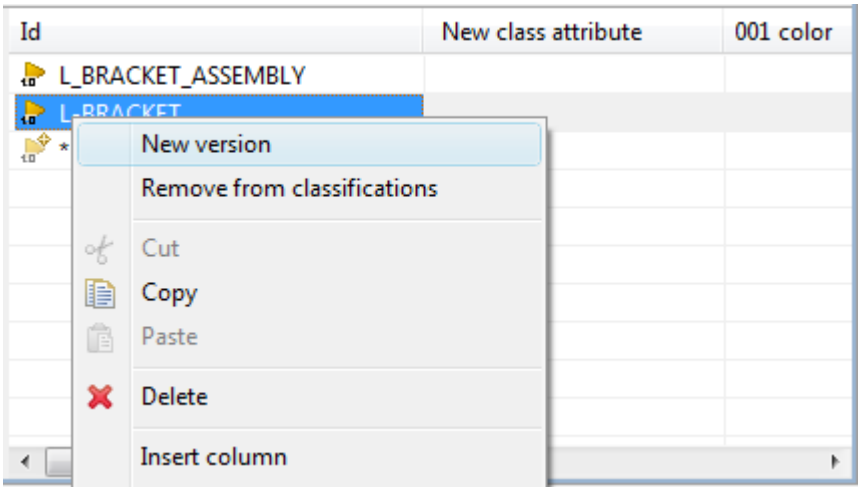

Figure 1. Context sensitive menu to create a new part version

- Select **New version** menu item. A new line with the same data (**Id**, **Name**) appears under the previous part version row. The **Version** column cell is marked with an asterisk (see Figure 2):
  - Enter a number of the version into the **Version** cell.

Note. A new part version will be assigned automatically to the classification, the previous version was assigned to.

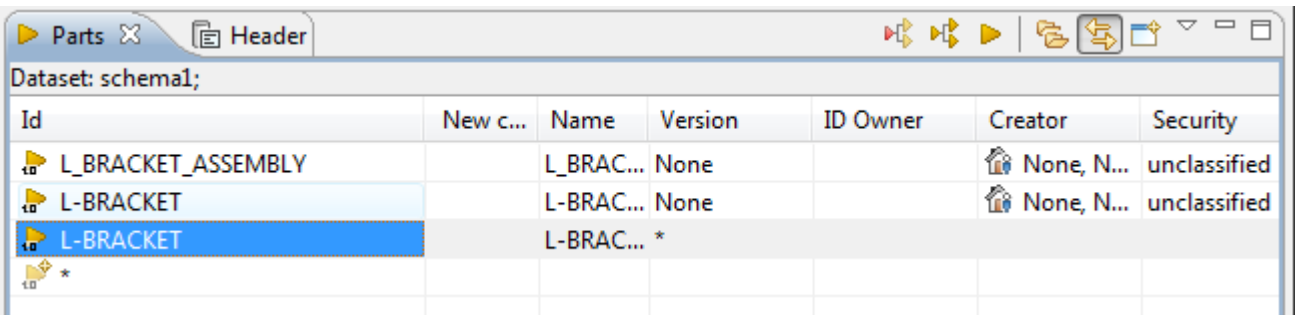

Figure 2. Created new part version row

To top

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9.13.5. Copying and Pasting

The following copy / pasting functionality is implemented in the **Parts** view:

- Copy / paste part version
- Copy / paste part classification object: classification system, classification, class attribute

Copy / Paste Part Version

IDA-STEP presents possibility to copy an applicable part version and paste it not only inside the currently opened **Parts** view, but also into another opened STEP file **Parts** view.
Follow the routine to copy and paste a part version in the **Parts** view:

- Select an applicable part version.
- Right click and select **Copy** menu item in the appeared context sensitive menu or press **CTRL + C** keys.
- Right click on the autorow at the bottom of the table.
- Select **Paste** menu item in the context sensitive menu or press **CTRL + V keys**. The copy of the selected part version appears in the table list (see Figure 1).

Note. The copied part version contains all data of the initial part version (Name, part definitions, components, shapes and etc. see Figure 1).

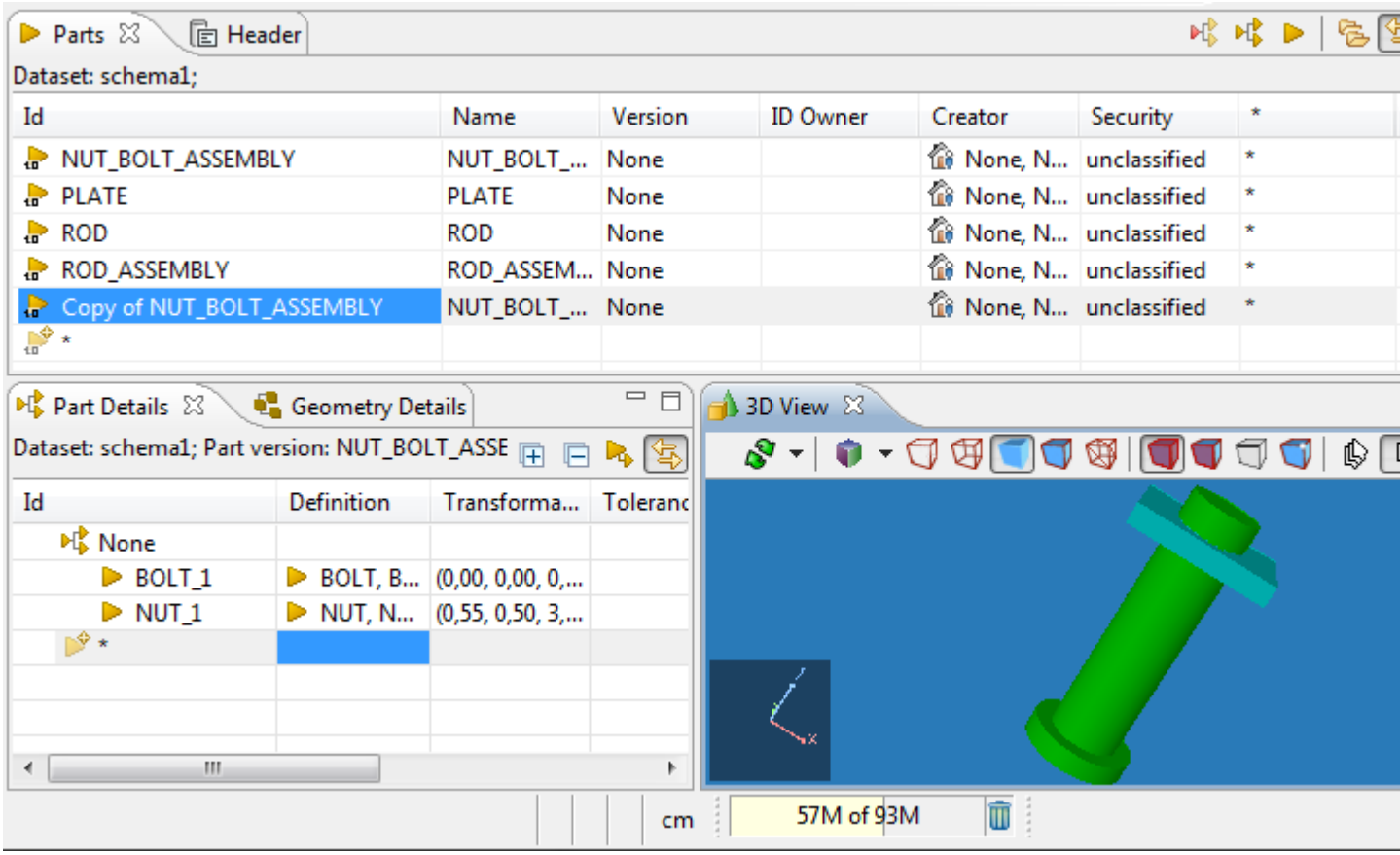

Figure 1. Copy of the part version in Parts view

To top

Copy / paste part classification object

First of all, press **Show classifications** button in the **Parts** view toolbar to see a part classification hierarchy on the left side of the **Parts** view.
Follow the routine to copy and paste an applicable object in the part classification hierarchy:

- Select an applicable object in the part classification hierarchy (classification system, classification or class attribute).
- Right click and select **Copy** menu item in the appeared context sensitive menu or press **CTRL** + **C** keys.
- Select a destination place in the part classification hierarchy.
- Right click and select **Paste** menu item in the appeared context sensitive menu or press **CTRL** + **V** keys. The copy of some part classification object is made with all related objects.

Note. If the destination place is selected not correctly, the pasting is not allowed, i.e. **Paste** menu item is disabled. For example, a classification system can be pasted only as a root node. If you try to paste it on some other object in the hierarchy, simply it is not allowed.

To top

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9.13.6. Editing Data

You can edit entered data in the Parts window:

- Classification system data
- Classification data
- Classification attribute data
- Part version data

Edit Classification System Data

Follow the routine to edit a classification system name:

- Double click on a classification system node in the tree.
- Edit the current or enter a new name.
- Press **Enter** key to turn off edit mode.

Edit Classification Data

Follow the routine to edit a classification name:

- Double click on a classification node in the tree.

- Edit the current or enter a new name.
- Press **Enter** key to turn off edit mode.

Edit Class Attribute Data

Follow the routine to edit a class attribute name:

- Double click on a classification attribute node in the tree. The class attribute name is selected (see Figure 3).
- Edit the current or enter a new name.
- Press **Enter** key to turn off edit mode. A new name is set for the selected class attribute in the tree and in the title of class attribute column in the table list.

Or

- Click on a classification attribute node in the tree. The **Details of classification attribute** page appears on the right side of the **Part Classification** window:
    - Enter the classification attribute identifier into **Id** field.
    - Edit or enter a new name in the **Name** field.
    - Enter descriptive information about the classification attribute in the **Description** field.

To top

Edit Part Data

Follow the routine to edit a part version data:

- Double click on an applicable column of the part version you want to edit the current text or just type text to enter new values:
    - **Id** - edit the current or enter new identifier.
    - **Name** - edit the current or enter new part name.
    - **Version** - edit the current or enter new version identifier.
    - <**class attribute name**> - enter or edit the current value in the field and select an applicable unit from the list.
    - **ID Owner, Creator** - select an applicable value from the drop down list or press the button next to the text field and select an applicable organization from the appeared **Select entity** dialog window.
- Press **Enter** key to go down to the next cell.

Note. If you change an identifier and/or a name of the part version, which has several versions, the changes are applied to all versions of that part.

To top

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9.13.7. Assigning Part to Classification

Follow the routine to assign a part version to an applicable classification:

- Click on a part in a table list You can select:
  - an unclassified part version in a list, which appears after all nodes in the tree are unselected by mouse click on the empty tree area,

  Or

  - a classified part version in a list of a particular classification by clicking on a particular node in a classification hierarchy.
- Drag the selected part version from a table list into the destination classification. When dragging the part version into the tree, the classification that the part version is being dropped into will be selected, also the transparent part icon with an identifier are visible (see Figure 1). Release the mouse button after you reached the destination classification node.

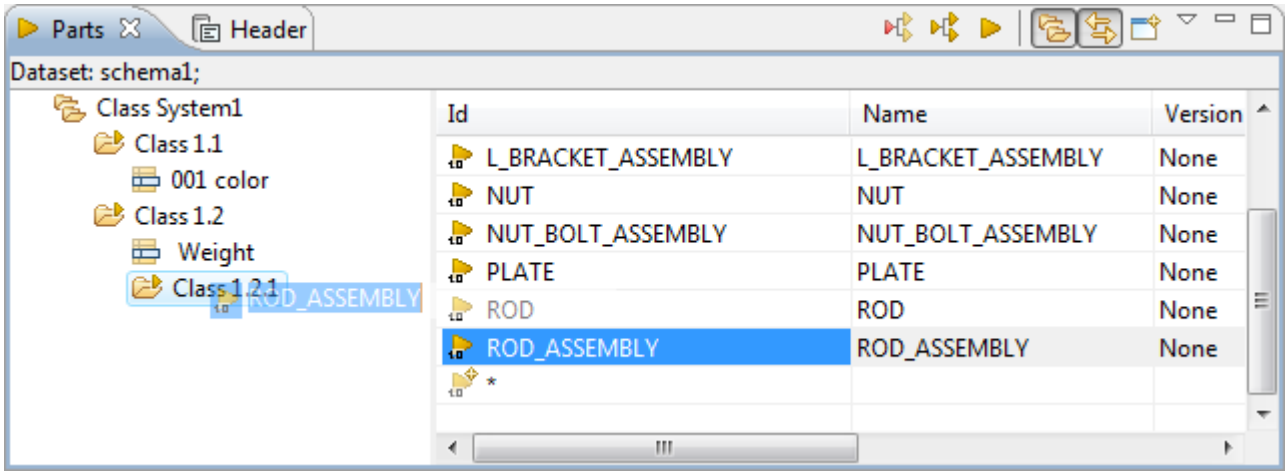

Figure 1. Part 'rod_assembly' dragged into 'Class 1.2.1' classification

- The part version is copied to the selected classification.
- This can also be achieved by copying a part version in a table list, then selecting a destination classification in the tree and pasting.

Note. If a part version has several versions, all versions of this part will be moved to the selected classification.

To top

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9.13.8. Removing Part Version from Classification

You have possibility to remove an applicable part version from the classification it is assigned to:

- Open a list of part versions of the classification, a part should be removed from.

Or

- Open a list of all part versions (classified and not), if you want to remove a part form all classifications it is assigned to.
- Right click on the part version to be removed. The context sensitive menu appears (see Figure 1).

- Select **Remove from classification** menu item.

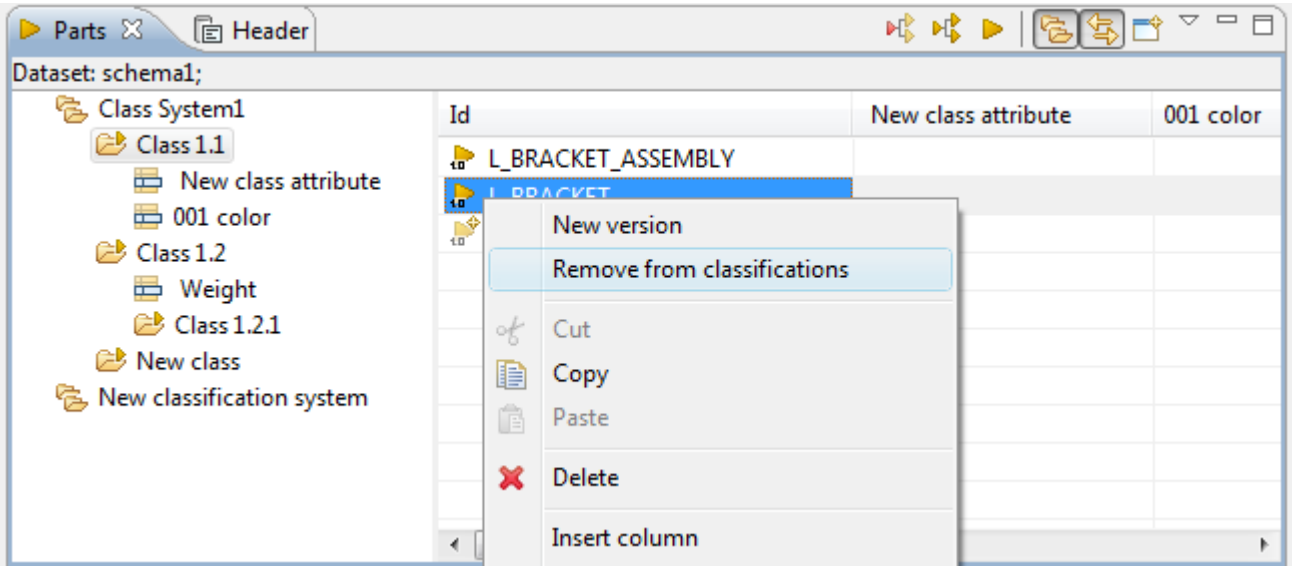

Figure 1. Remove from classification menu item (right). Highlighted classifications, the part is assigned to (left)

Note. If a part has several versions, all versions of this part will be moved from the classification.

To top

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9.13.9. Creating Classification System

Follow the routine to create a new classification system:

- Right click on the empty tree area (left side of the Parts window).
- The context sensitive menu appears (see Figure 1).

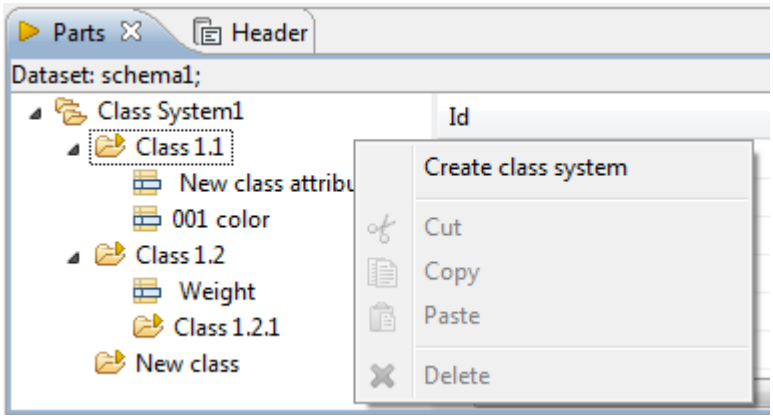

Figure 1. Context sensitive menu, Create class system menu item

- Select **Create class system** menu item.
- A new node named **New classification system** is created in the tree (see Figure 2).

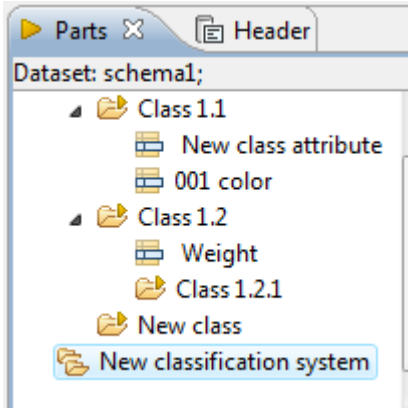

Figure 2. New classification system, named 'New classification system' in the tree

- To change name of the newly created classification system double click on the node and enter an applicable name.

To top

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9.13.10. Creating Classification

You can create new top level classification or sub-classification in the part classification hierarchy.

Creating Top Level Classification

Follow the routine to create a top level classification:

- Right click on the classification system node in the tree. The context sensitive menu appears (see Figure 1).
- Select **Create class** menu item.

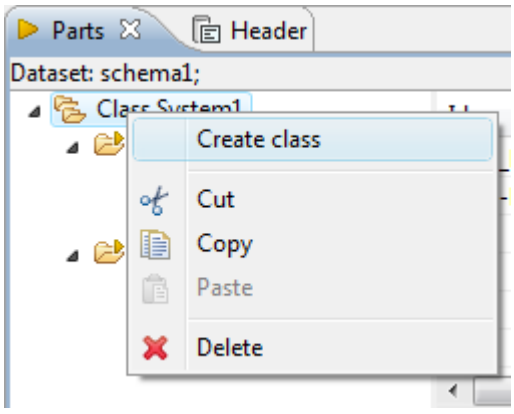

Figure 1. Context sensitive menu, invoked from dataset branch

- A new node named **New class** is created in a hierarchy (see Figure 2).

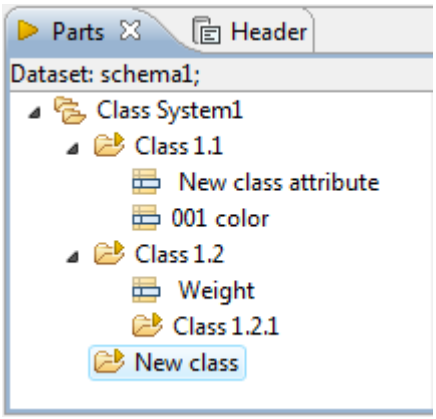

Figure 2. New classification, named 'New class' in the tree

- To change name of the newly created classification double click on the node and enter an applicable name.

To top

Creating Sub-Classification

Follow the routine to create a new sub-classification:

- Right click on a classification name, where a sub-classification should be created. The context sensitive menu appears (see Figure 3).
- Select **Create class** menu item.

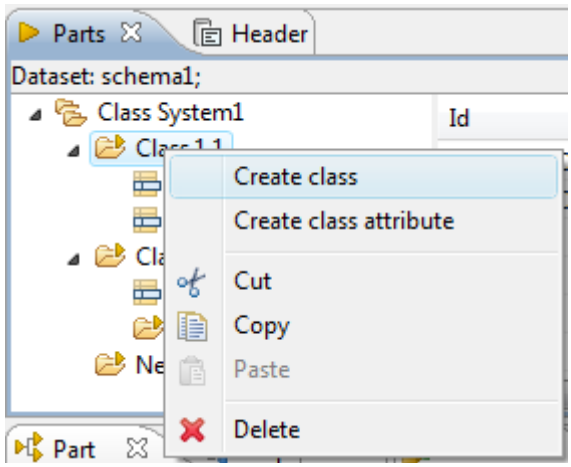

Figure 3. Context sensitive menu to create sub classification

- A new node named **New class** is created. You can change the name as it is described in above chapter.

To top

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9.13.11. Creating Classification Attribute

Follow the routine to crate a new class attribute for the selected classification:

- Right click on an applicable classification node. The context sensitive menu appears (see Figure 1).
- Select **Create class attribute** menu item.

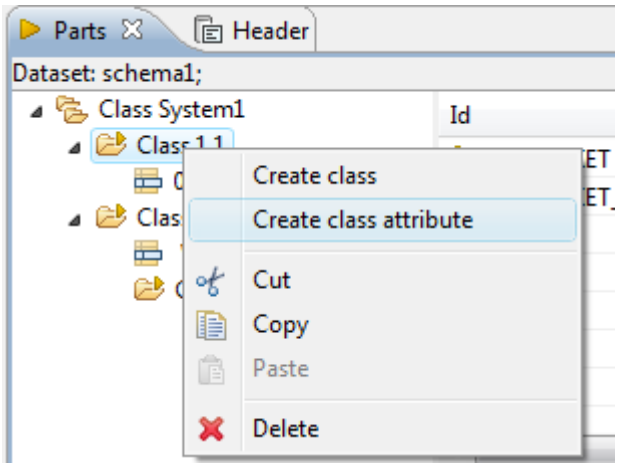

Figure 1. Context sensitive menu to create new class attribute

- The class attribute, named **New class attribute** appears in the hierarchy. Also a new column of the created class attribute appears in the table list of the classification the class attribute is assigned to (see Figure 2).

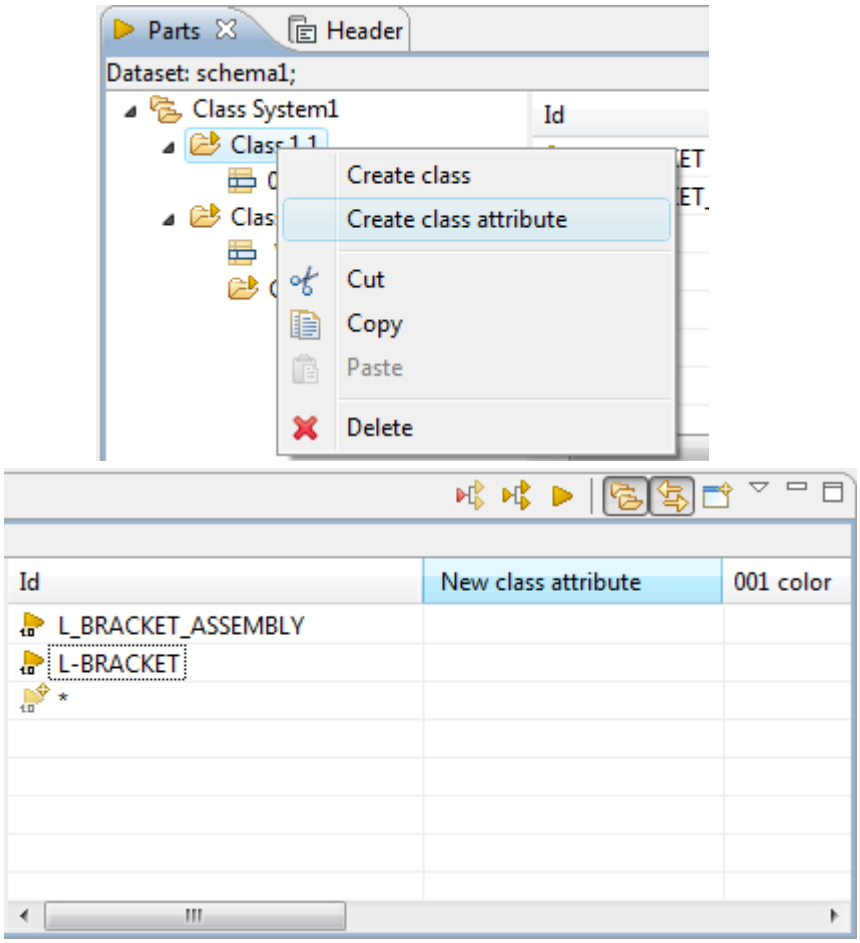

Figure 2. 'New class attribute' in the classification hierarchy (left) and in the table list (right)

- Click on the created class attribute to open the **Details of classification attribute** page (see Figure 3):
  - Enter classification attribute identifier into **Id** field.
  - Edit or enter a new name in the **Name** field.
  - Enter descriptive information about the classification attribute in the **Description** field.

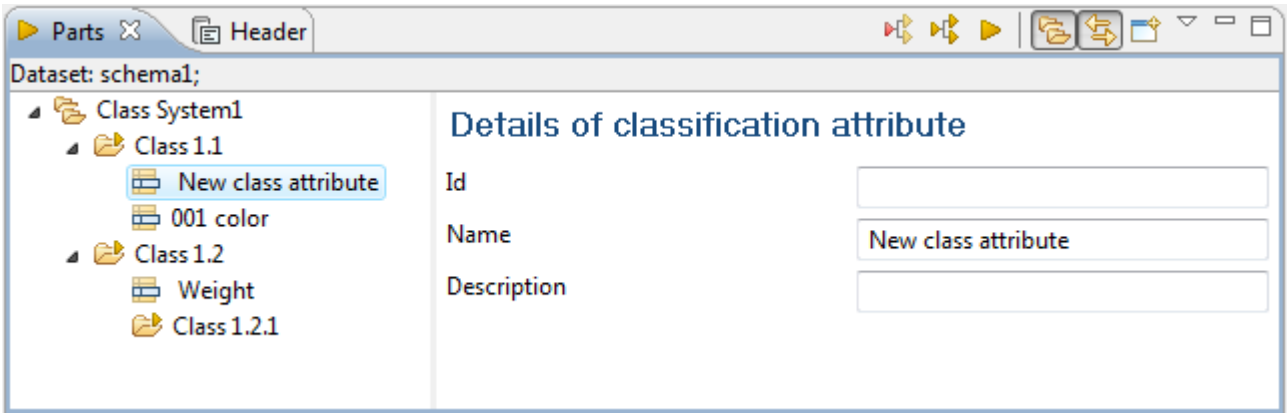

Figure 3. Details of classification attribute page

To top

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9.13.12. Deleting

Follow the routine to delete an incorrect or redundant element in the Parts window:

- Right click on the element you want to remove in the tree / table list:
  - classification system node,
  - classification node,
  - classification attribute node,
  - part in a table list.

- The context sensitive menu appears (see Figure 1).
- Select **Delete** menu item.

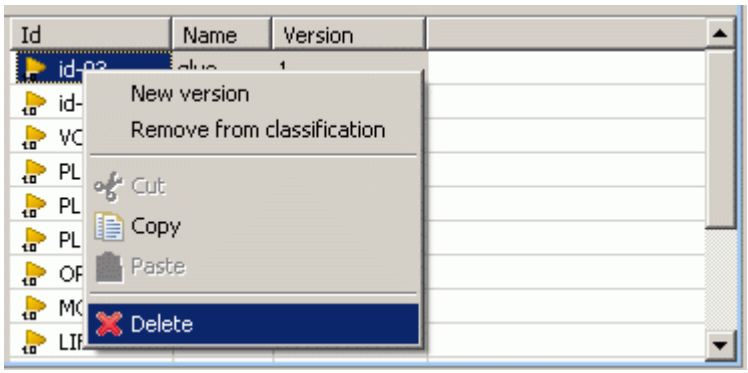

Figure 1. Context sensitive menu, Delete menu item

To top

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9.13.13. Moving

Moving can be performed by dragging and dropping selected objects in the Parts window:

- Moving classification - to change task location in the hierarchy.
- Moving classification attribute - to change the location of the selected classification attribute.
- Move part versions - to assign a part version to a classification or remove a part version form a classification.

To top

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9.14.1. Passage Technologies View

The **Passage technologies** view (available by selecting in the main menu bar **Window -> Show view -> Other** then under **PCB Layer Stack** node) presents a list of passage technologies (see Figure 1). The first column presents the row element - passage technology, other columns contain data of the particular passage technology. The default column perspective contain the following columns:

- **Id** - specifies an identifier of the passage technology.
- **Plating** - specifies if the passage is plated or not.
- **Min size** - specifies the finished passage extent for the passage technology.
- **Min Interior** - specifies the minimum interior extent for the passage technology. This characteristic is available only for a passage, which technology type is *via*.
- **Terminus** - specifies the passage terminus condition for the passage technology.
- **Deposition thickness** - specifies the finished deposition thickness value for the passage technology.
- **Deposition Material**- specifies the passage deposition material identification.
- **Electrical conductivity** - specifies the passage technology conductivity classification for deposition material identification for a passage.
- **Filling Material**- specifies the passage filling material identification for the passage technology.
- **Electrical conductivity** - specifies the passage technology conductivity classification for filling material identification for a passage.

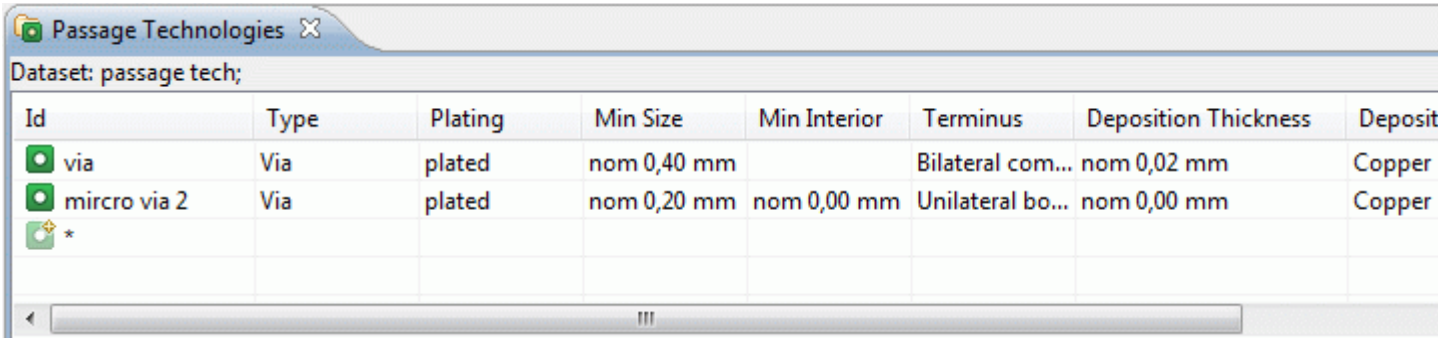

Figure 1. Passage technologies view (default columns)

To top

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9.14.2. Creating Passage Technology

Follow the routine to create a new passage technology element in the **Passage Technologies** view tree table (see Figure 1):

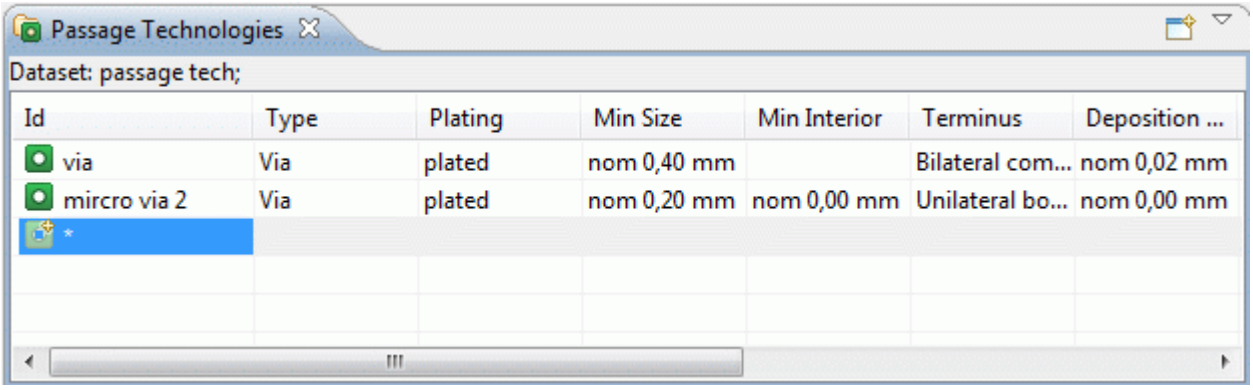

Figure 1. Passage technologies view tree table with highlighted auto row

- Select the auto row in the tree table (see Figure 1 highlighted cell in the **Id** column).
- Click on the selected auto row to enter into the edit mode.
- Enter an identifier of the new passage technology element.
- Press **Enter** key. A new passage technology element appears in the **Passage Technologies** view tree table (see Figure 2). The **Type** column is filled with a default value automatically. The value can be changed together with other passage technology data.

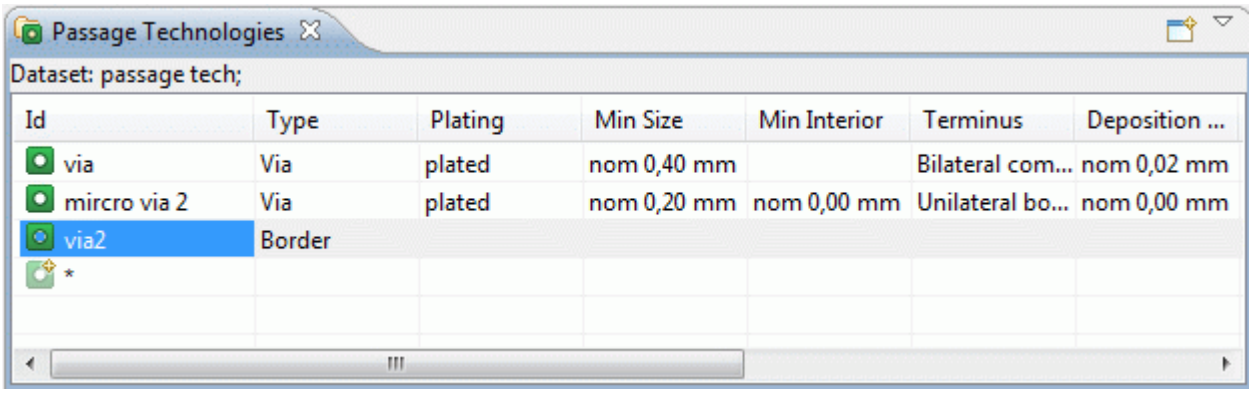

Figure 2. Created passage technology element (highlighted)

To top

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9.14.3. Editing Passage Technology Data

Activate the **Passage Technologies** view. Select an applicable passage technology element in the tree table. You can edit the following:

- Enter a new or edit the current passage technology identifier in the **Id** column.
- Select an applicable value in the **Type** combo box. You can not enter a new value.
- Click with mouse on the **Plated** column to set an applicable value.
- Enter a new or edit the current value of the finished passage extent in the **Min size** column. You can also select a new tolerance value or/and change units.
- Enter a new or edit the current value of the minimum interior extent in the **Min interior** column. You can also select a new tolerance value or/and change units.
- Select a new passage terminus condition value in the **Terminus** column.
- Enter a new or edit the current finished deposition thickness value in the **Deposition thickness** column. You can also select a new tolerance value or/and change units.
- Select or enter a new value of a deposition material identification in the **Deposition Material** column. If the selected deposition material contains the associated electrical conductivity

classification, the **Electrical conductivity** column cell is filled with a predefined value.

- Select an applicable value of the electrical conductivity classification in the **Electrical conductivity** column. The selected value is associated to the current deposition material identification, presented in the **Deposition Material** column.
- Select or enter a new value of a deposition material identification in the **Filling Material** column. If the selected filling material contains the associated electrical conductivity classification, the **Electrical conductivity** column cell is filled with a predefined value.
- Select an applicable value of the electrical conductivity classification in the **Electrical conductivity** column. The selected value is associated to the current filling material identification, presented in the **Filling Material** column.

To top

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9.15.1. Passages View

The **Passages** view (available by selecting in the main menu bar **Window -> Show view -> Other** then under **PCB Layer Stack** node) presents a tree table with a hierarchy of passages, referenced by a particular design stack model, which is identified in the top section of the **Passages** view window (see Figure 1).

Note: The data appears in the **Passages** view only after an applicable design stack model is selected in the **Design Stack Models** view.

First column presents row element - passage. Other columns contain the assigned data for the particular passage element.

The default column perspective contain the following columns:

- **Id** - an identifier of a passage.
- **From** - specifies the top most stratum the passage is passing trough, if the passage orientation is top-bottom, and the bottom most stratum otherwise.
- **To** - specifies the top most stratum the passage is passing through, if the passage orientation is bottom-top, and the top most stratum otherwise.
- **Connected Ring Size** - specifies a size of the connected minimum annular ring of the passage.
- **Unconnected Ring Size** - specifies a size of the unconnected minimum annular ring of the passage.
- **Macro** - specifies an allocated technology of the passage.
- **Type** - specifies a technology type value of the passage. The available values: border, component termination, unsupported and via.
- **Plated** - specifies if the passage is plated or not.
- **Deposition thickness** - specifies the finished deposition thickness value for the passage technology.
- **Min Size** - specifies the finished passage extent for a passage technology.
- **Min Interior** - specifies the minimum interior extent for a passage technology.
- **Terminus Condition** - specifies the passage terminus condition for a passage technology.

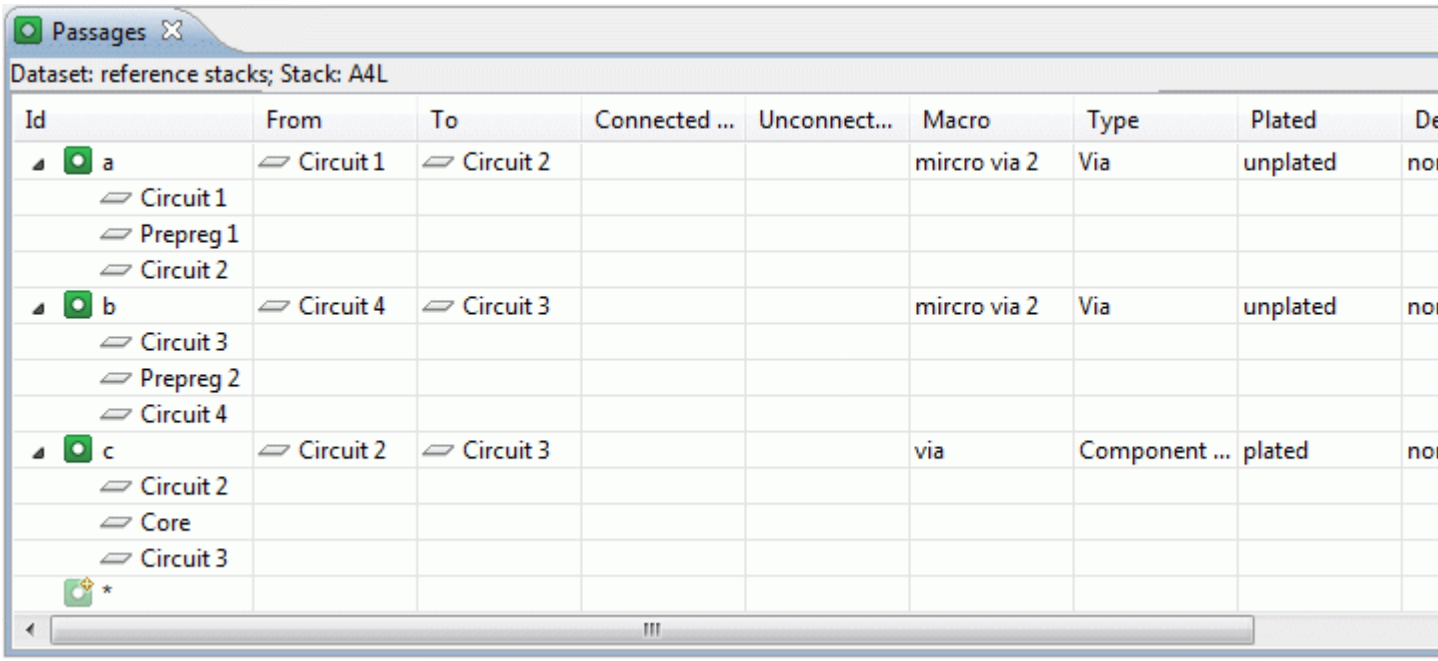

Figure 1. Passages view

To top

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9.15.2. Passages Hierarchy

The Id column in the Passages view presents the hierarchy of passages (see Figure 1):

- **Root node** - the passage identifier.
- **Second level node** - the layer name, the particular passage is passing through.

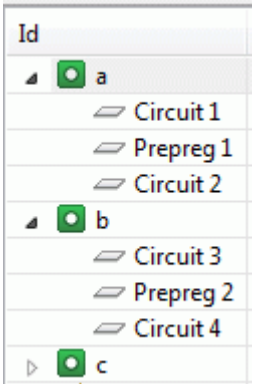

Figure 1. The hierarchy of passages

To top

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9.15.3. Creating Passage

Follow the routine to create a new passage in the **Passages** view tree table (see Figure 1):

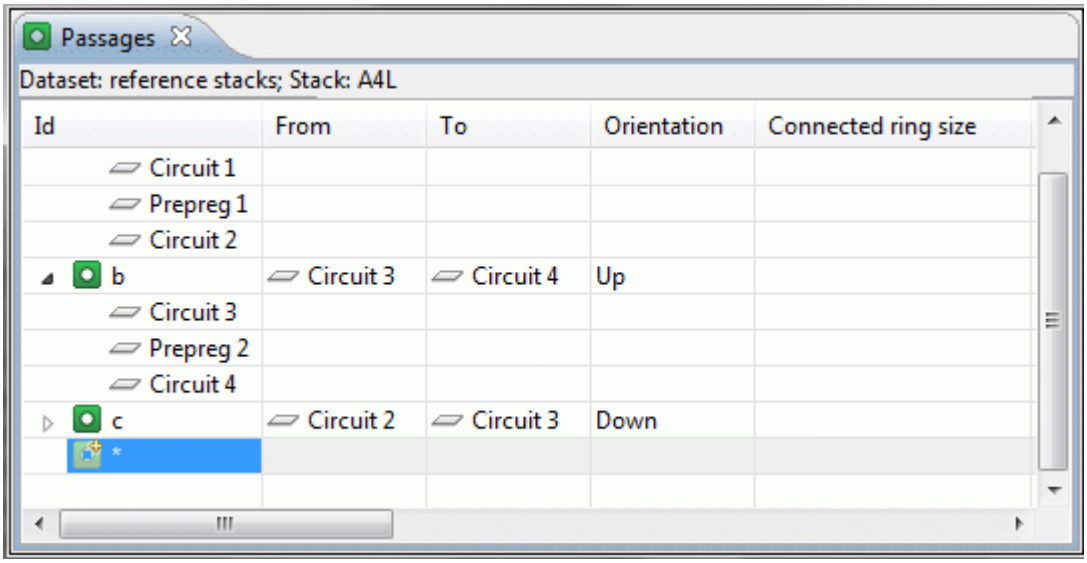

Figure 1. Passages view with highlighted autorow

- Select the autorow in the table (see Figure 1 highlighted cell in **Id** column).
- Click on the selected autorow to enter in to the edit mode.
- Enter an identifier for the new passage element.
- Press **Enter** key. A new record appears in the **Passages** view tree table (see Figure 2).

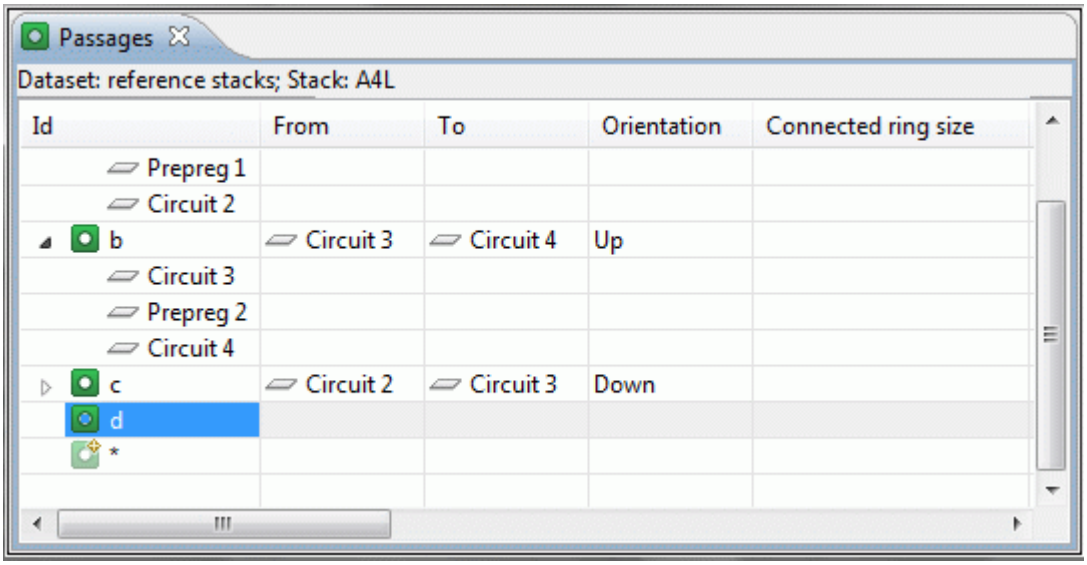

Figure 2. Created passage element in the tree table

To top

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9.15.4. Editing Passage Data

Activate the **Passages** view. Select an applicable passage in the tree table. You can edit the following:

- Enter an identifier of a passage into the **Id** column.
- Specify the starting layer the passage is passing through in the **From** column by dragging an applicable layer from the **Stratums** view.
- Specify the last layer the passage is passing through in the **To** column by dragging an applicable layer from the **Stratums** view.
- Set the orientation value (down or up) of the passage in the **Orientation** column by clicking with

mouse button.

- Enter a size of the connected minimum annular ring of the passage in the **Connected ring size** column.
- Enter a size of the unconnected minimum annular ring of the passage in the **Unconnected ring size** column.
- Select an allocated technology of the passage in the **Macro** column combo box.

Note. If you select an existing value (coming from **Passage Technologies** view) in the **Macro** combo box, then some columns are filled with data automatically.

- Select a technology type value of the passage in the **Type** column combo box.
- Specify if the passage is plated or not by clicking with mouse in the **Plated b**
- Enter the finished deposition thickness value for the passage technology into the **Deposition thickness** column.
- Enter the finished passage extent for a passage technology into the **Min size** column.
- Enter the minimum interior extent for a passage technology into the **Min interior** column.
- Select the passage terminus condition for a passage technology in the **Terminus condition** column combo box.

To top

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9.16.1. Persons and Organizations

Organizations captures information concerning organizations, persons and positions associated with those organizations along with useful relationships to experience and qualifications.
The **Persons and Organizations** window is split into the following areas (see Figure 1):

- Organization Hierarchy
- Detailed Information
  - Persons
  - Organizations
  - Positions
  - Address Details

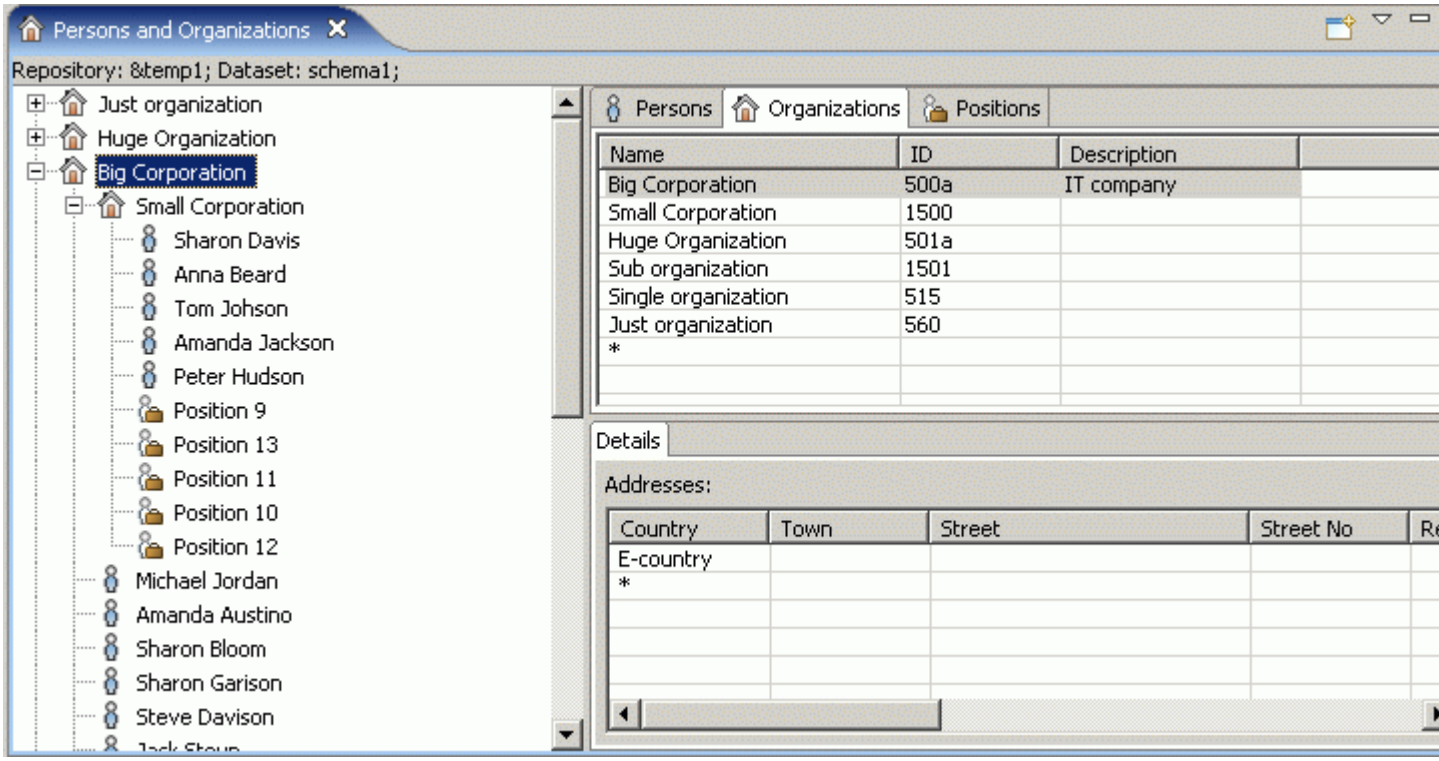

Figure 1. Organization Editor view

Organization Hierarchy

The hierarchy (tree) of organizations (see Figure 2), available for the selected dataset, is presented as a tree. The top level organization may have sub organizations, persons and positions situated beneath. Each sub organization also may have their own persons and positions beneath. The same person may be associated to many organizations - the same person may be included into several branches. The position can be assigned only to one organization.

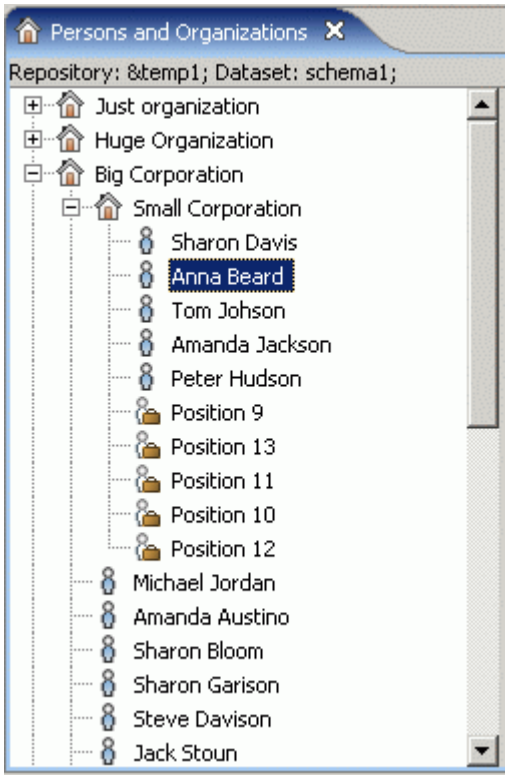

Figure 2. Hierarchy of organizations

To top

Detailed Information

Detailed and editable information is presented in the special tabbed panels on the right section of the **Persons and Organizations** window. When you click on an element in the organizations tree, the panels are synchronized to show the selected element (see Figure 3).

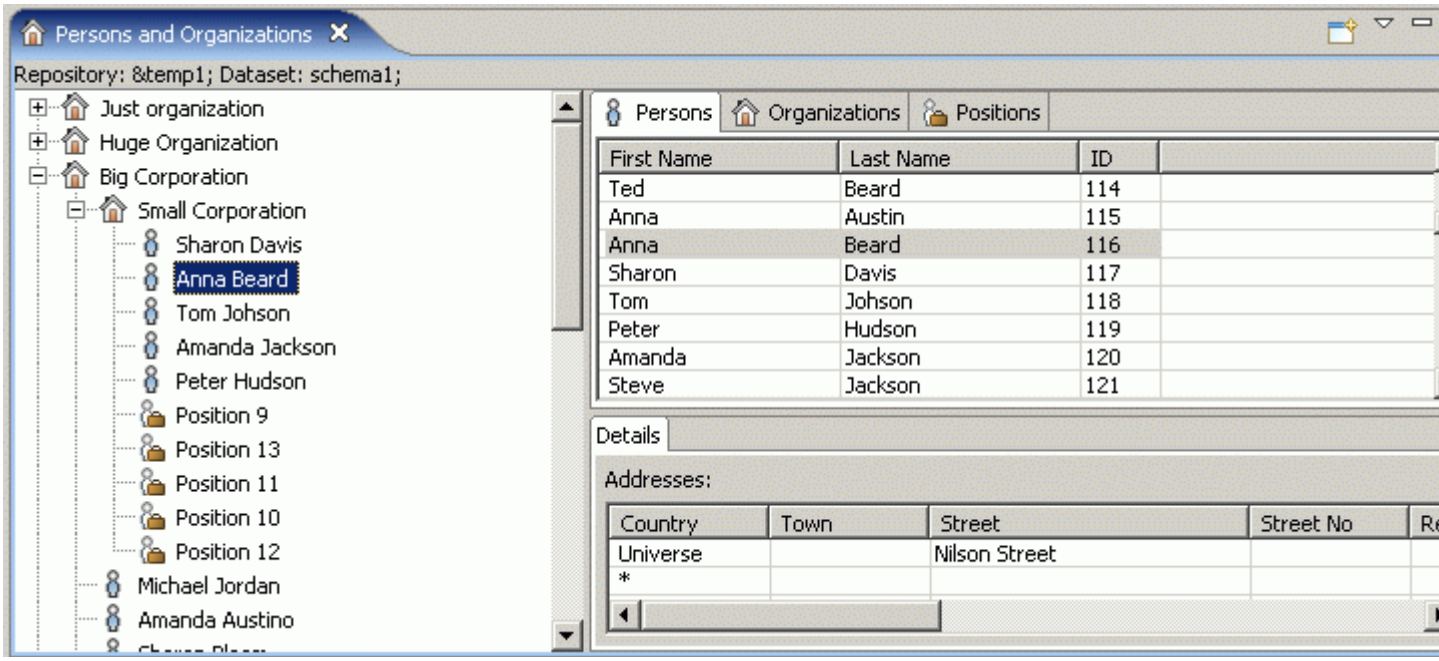

Figure 3. Selected person in the organizations tree is highlighted in the **Persons** tabbed panel

Persons

Click on **Persons** tab to activate a panel. The **Persons** panel presents a list of all available persons and their details: first name, last name and identifier.
Click on an applicable person in the table to view detailed address information.

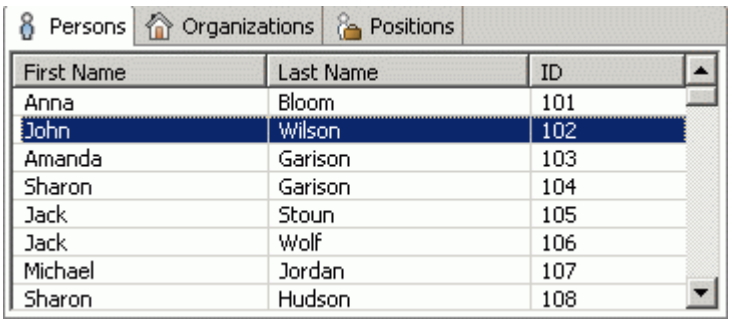

Figure 4. Persons tabbed panel

Organizations

Click on **Organizations** tab to activate a panel. The **Organizations** panel presents a list of all available organizations and their details: identifier, name and description.
Click on an applicable organization in the table to view detailed address information.

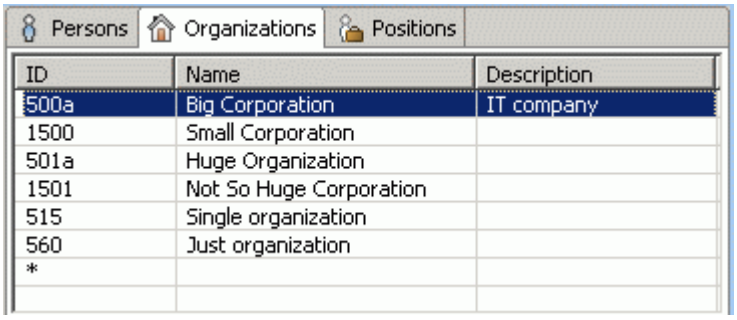

Figure 5. Organizations tabbed panel

Positions

Positions present the data that identify an individual's position in an organization and the relationships between positions and organizations. Within an organization individuals hold a discreet position which confers their authority and their place, or rank, in relation to others within that organization.
Click on **Positions** tab to activate a panel. The **Positions** panel presents a list of all available positions and their details: identifier, name, description and organization the position is assigned to. Click on an applicable position in the table to view detailed address information.

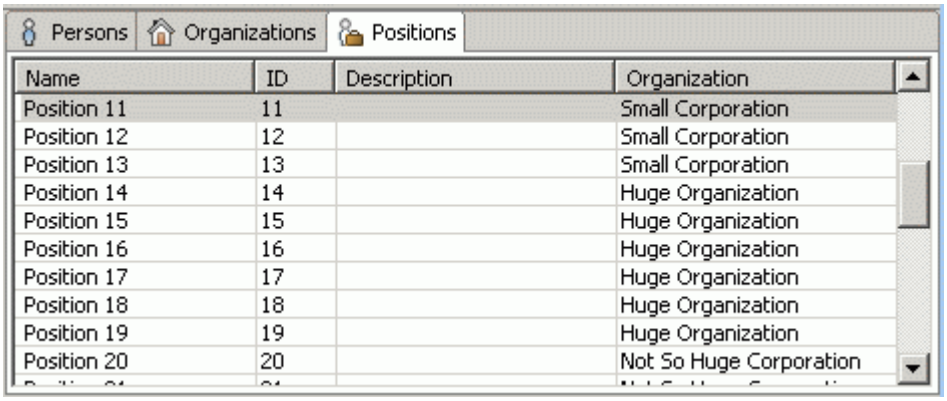

Figure 6. Positions tabbed panel

Address Details

When you click on an element in the panels, described above, the **Address details** panel is synchronized to show the addresses of the selected element (person, organization or position). The **Addresses** table presents the list of available addresses (see Figure 7) of the element (person, organization or position) selected in a particular tabbed panel.

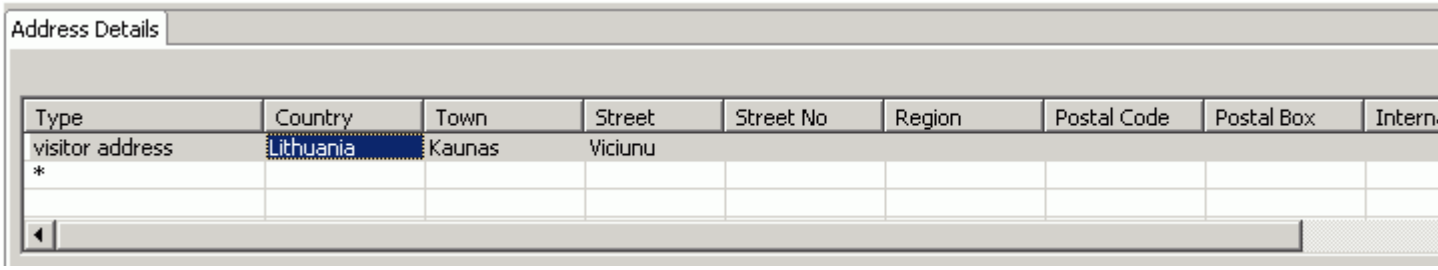

Figure 7. Address Details tabbed panel

To top

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9.16.2. Editing Person Data

Activate the **Persons** panel to edit data.You can make the following actions for the selected person in the table:

- Enter new or edit the first name in the **First Name** column.
- Enter new or edit the last name in the **Last name** column.
- Enter new or edit an identifier in the **ID** column.

To top

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9.16.3. Editing Organization Data

Activate the **Organizations** panel to edit data. You can make the following actions for the selected organization in the table:

- Enter new or edit an identifier in the **ID** column.
- Enter new or edit an organization name in the **Name** column.
- Enter new or edit descriptive representation about the organization in the **Description** column.

To top

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9.16.4. Editing Position Data

Activate the **Positions** panel to edit data. You can make the following actions for the selected position in the table:

- Enter new or edit an identifier in the **ID** column.
- Enter new or edit an organization name in the **Name** column.
- Enter new or edit descriptive representation about the position in the **Description** column.

To top

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9.16.5. Editing Address Data

Activate **Address details** panel to edit address data. You can make the following actions for the selected

address in the table:

- Enter or select the address type from the list in the **Type** filed:
    - Click on the **Type** field.
    - Enter the address type.

    Or

    - Select a type in the drop down list.
- Enter new or edit a country in the **Country** column.
- Enter new or edit a town in the **Town** column.
- Enter new or edit a street in the **Street** column,
- Enter new or edit a street number in the **Street number** column.
- Enter new or edit a region in the **Region** column.
- Enter new or edit a postal code in the **Postal code** column.
- Enter new or edit a postal box data in the **Postal box** column.
- Enter new or edit an internal location data in the **Internal location** column.
- Enter new or edit a telephone number in the **Telephone number** column.
- Enter new or edit a facsimile number in the **Facsimile number** column.
- Enter new or edit a telex number in the **Telex number** column.
- Enter new or edit an e-mail address in the **e-mail** column.

To top

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9.16.6. Creating New Element

You can add a new element (person, organization, position, address) into a particular table:

- Activate a panel with an applicable data:
    - Persons - to add a new person;
    - Organizations - to add a new organization;
    - Positions - to add a new position;
    - Address details - to add a new address.
- Select the autorow (the line, which is at the end of the list in a table, marked with an asterisk) (see Figure 1).

Eve	Johson	163
Ted	Hudson	164
*		

Figure 1. Table with autorow

- Enter particular data for the element you want to create: person, organization, position or address.

To top

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9.16.7. Copying and Pasting

You can copy and paste organizations, persons or positions in the Persons and Organizations window. You can copy elements in the tables, from a table to the Organizations tree, from the tree to table and in the Organizations tree:

- Copy Paste Person
- Copy Paste Organization
- Copy Paste Position

Copy Paste Person

Follow the routine to copy and paste a person:

- In the table of the **Persons** panel.
  - Activate the **Persons** panel.
  - Right click on an applicable person in the table list. A context sensitive menu appears (see Figure 1).
  - Select **Copy Person** menu item.

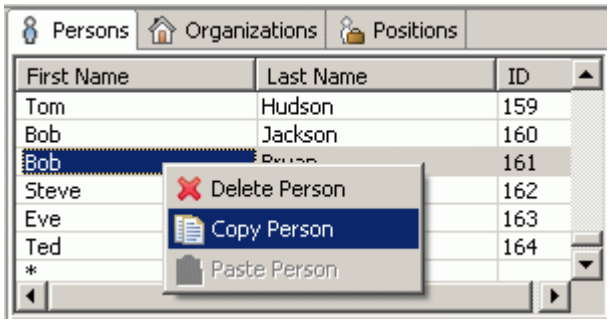

Figure 1. Context sensitive menu in Persons panel, Copy Person

  - Right click on the auto row (cell with an asterisk) at the bottom of the table list. A context sensitive menu appears (Figure 2).
  - Select **Paste Person** menu item.

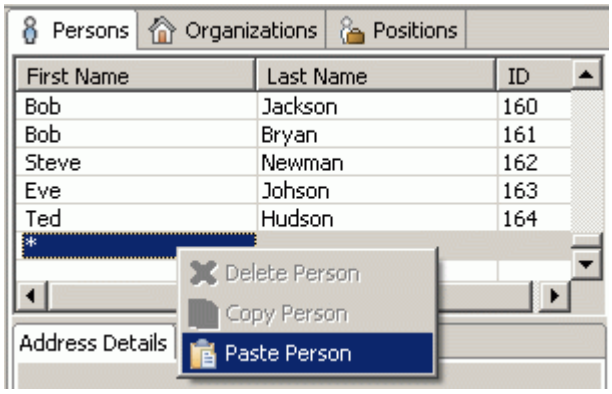

Figure 2. Context sensitive menu in Persons panel, Paste Person

- From the table in the **Persons** tabbed panel to the Organizations tree:
  - Right click on an applicable person in the table list.
  - Select **Copy Person** menu item.
  - Right click on an applicable organization node in the Organizations tree.
  - Select **Paste** menu item.

- From the Organizations tree to the table in the **Persons** tabbed panel:
  - Right click on an applicable person node in the tree.
  - Select **Copy** menu item.
  - Right click on the auto row (cell with an asterisk) at the bottom of the table list in the **Persons**.
  - Select **Paste Person** menu item.

- In the Organizations tree:
  - Right click on an applicable person node in the tree.
  - Select **Copy** menu item.
  - Right click on an applicable organization node in the Organizations tree.
  - Select **Paste** menu item.

To top

Copy Paste Organization

Follow the routine to copy and paste an organization:

- In the **Organizations** panel.
  - Activate the **Organizations** panel.
  - Right click on an applicable organization in the table list. A context sensitive menu appears (see Figure 3).

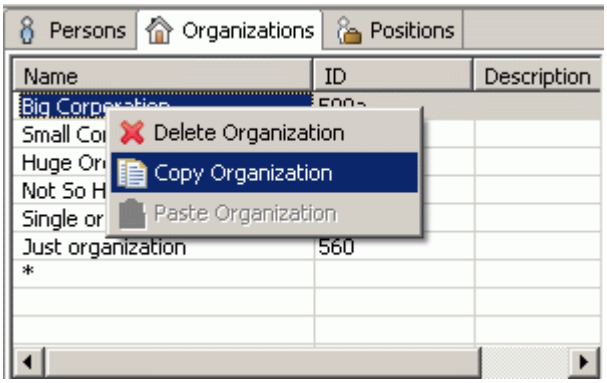

Figure 3. Context sensitive menu in Organizations panel, Copy Organization

  - Select **Copy Organization** menu item.
  - Right click on the auto row (cell with an asterisk) at the bottom of the table list. A context sensitive menu appears (see Figure 4).
  - Select **Paste Organization** menu item.

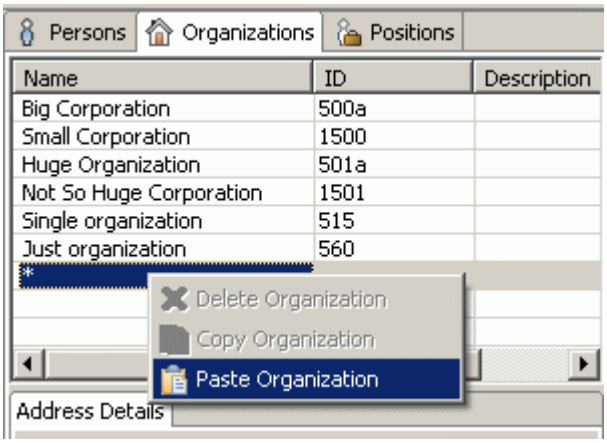

Figure 4. Context sensitive menu in Organizations panel, Paste Organization

- From the table in the **Organizations** tabbed panel to the Organizations tree:
  - Right click on an applicable organization in the table list.
  - Select **Copy Organization** menu item.
  - Right click on the tree area or on an applicable organization node in the Organizations tree.
  - Select **Paste** menu item.

- From the Organizations tree to the table in the **Organizations** tabbed panel:
  - Right click on an applicable organization node in the tree.
  - Select **Copy** menu item.
  - Right click on the auto row (cell with an asterisk) at the bottom of the table list in the **Organizations** tabbed panel.
  - Select **Paste Organization** menu item.

- In the Organizations tree:
  - Right click on an applicable organization node in the tree.
  - Select **Copy** menu item.
  - Right click on the tree area or on an applicable organization node in the Organizations tree.
  - Select **Paste** menu item.

To top

Copy Paste Position

Follow the routine to copy and paste a position:

- In the **Positions** panel:
  - Activate the **Positions** panel.
  - Right click on an applicable position in the table list. A context sensitive menu appears (see Figure 5).
  - Select **Copy Position** menu item.

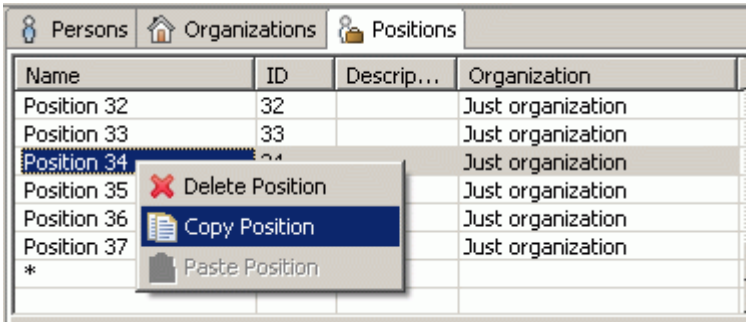

Figure 5. Context sensitive menu in Positions panel, Copy Position

  - Right click on the auto row (cell with an asterisk) at the bottom of the table list. A context sensitive menu appears (see Figure 6).
  - Select **Paste Position** menu item.

Figure 6. Context sensitive menu in Positions panel, Paste Position

- From the table in the **Positions** tabbed panel to the Organizations tree:
  - Right click on an applicable position in the table list.
  - Select **Copy Position** menu item.
  - Right click on an applicable organization node in the Organizations tree.
  - Select **Paste** menu item.

- From the Organizations tree to the table in the **Positions** tabbed panel:
  - Right click on an applicable position node in the tree.
  - Select **Copy** menu item.
  - Right click on the auto row (cell with an asterisk) at the bottom of the table list in the **Positions** tabbed panel.
  - Select **Paste Position** menu item.

- In the Organizations tree:
  - Right click on an applicable position node in the tree.
  - Select **Copy** menu item.
  - Right click on the tree area or on an applicable organization node in the Organizations tree.
  - Select **Paste** menu item.

To top

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9.16.8. Deleting

You can delete redundant or incorrect elements from the dataset in the Persons and Organizations window (table list and tree):

- Delete Person
- Delete Organization
- Delete Position

Delete Person

Follow the routine to delete a person

- From the table list:
  - Activate **Persons** panel.
  - Right click on the person, you want to delete, in the table list. A context sensitive menu appears (see Figure 1).
  - Select **Delete Person** menu item.

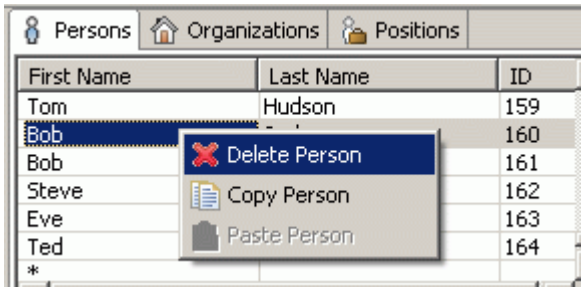

Figure 1. Context sensitive menu, Delete Person

- From the Organizations tree:
  - Right click on a person node in the organizations tree on the left side of the window.
  - Select **Delete** menu item.

To top

Delete Organization

Follow the routine to delete an organization

- From the table list:
  - Activate **Organizations** panel:
  - Click on the tab on the right side of the window.
  - Right click on the organization, you want to delete, in the table list. A context sensitive menu appears (see Figure 2).
  - Select **Delete Organization** menu item.

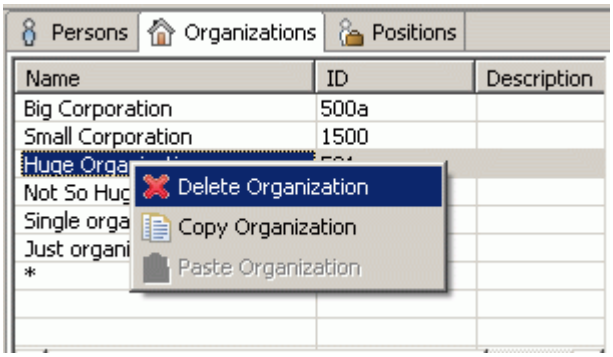

Figure 2. Context sensitive menu, Delete Organization

- From the Organizations tree:
  - Right click on a organization node in the organizations tree on the left side of the window.
  - Select **Delete** menu item.

To top

Delete Position

Follow the routine to delete a position

- From the table list:
  - Activate **Positions** panel:
  - Click on the tab on the right side of the window.
  - Right click on the position, you want to delete, in the table list. A context sensitive menu appears (see Figure 3).
  - Select **Delete Position** menu item.

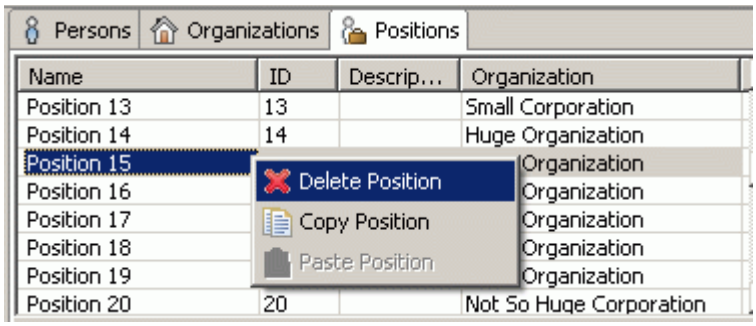

Figure 3. Context sensitive menu, Delete Position

- From the Organizations tree:
  - Right click on a position node in the organizations tree on the left side of the window.
  - Select **Delete** menu item.

To top

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9.16.9. Moving Element

The following movings can be performed by dragging and dropping in the Persons and Organizations window:

- Moving of the selected table element into the organizations tree;
- Moving of nodes in the organizations tree.

Moving Table Element into Tree

Follow the routine to move the table element into the organizations tree:

- Click on an applicable element in a particular tabbed panel:
  - Persons - to select a person;
  - Organizations - to select an organization;
  - Positions - to select a position.
- Drag and drop the element onto the destination node in the organizations tree. The destination node is highlighted on the tree (see Figure 1).

Or

- The same result can be achieved by copying and pasting the selected element.

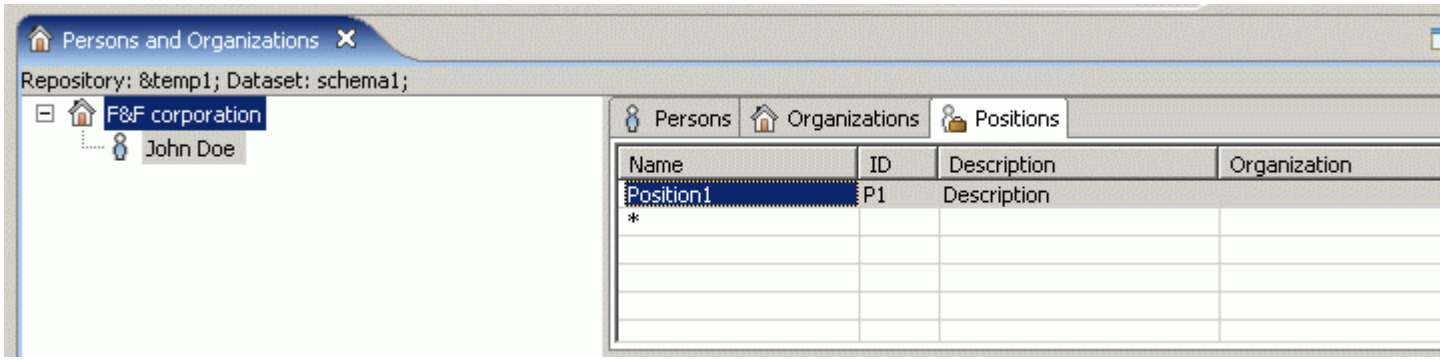

Figure 1. Position 'Position1' is dragged onto 'F&F corporation' node in the tree

To top

Moving Tree Nodes

You can change an organization hierarchy, by moving nodes into applicable positions:

- Click on the node in the organizations tree.
- Drag and drop it in the desired destination place.

Or

- The same result can be achieved by cutting and pasting the selected element.

You can move an applicable organization, a person or a position onto the desired organization node in a hierarchy.

To top

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9.16.10. Removing from Organization

There is a possibility to remove an element (sub-organization, person or position) from an organization hierarchy:

- Right click on an element node you want to remove (sub-organization, person or position). A context sensitive menu appears (see Figure 1).
- Select **Remove from Organization** menu item.

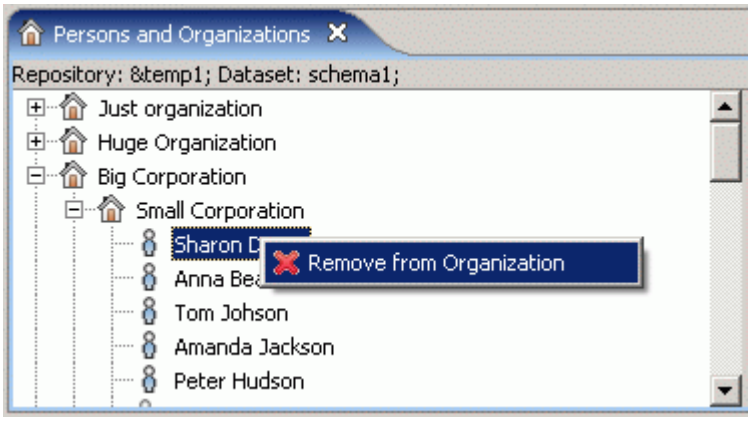

Figure 1. Context sensitive menu, Remove from organization

- The selected element is removed from the organizations hierarchy, but it is still present in the table list of a particular panel (**Organizations**, **Persons** or **Positions**).

To top

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9.17.1. Products View

A *Product as individual* records the identification of an existing or potential future artefact - an actual product whose properties can only be known by observation or by derivation from observations.
The **Products** view presents a list of product as individual instances together with assigned properties in the table list (see Figure 1). The default column perspective contains the following columns:

- **Id** - specifies an identifier of the *product as individual* version.
- **Name** - specifies the word or group of words by which the *product as individual* is referred to.
- **Description** - specifies additional information about the *product as individual*.
- **Design** - specifies a part, which is the design of the *product as individual*.
- **In Service Date** - specifies a date, when the *product as individual* was started to use.
- **Out of Service Date** - specifies the expiration date of the *product as individual*.

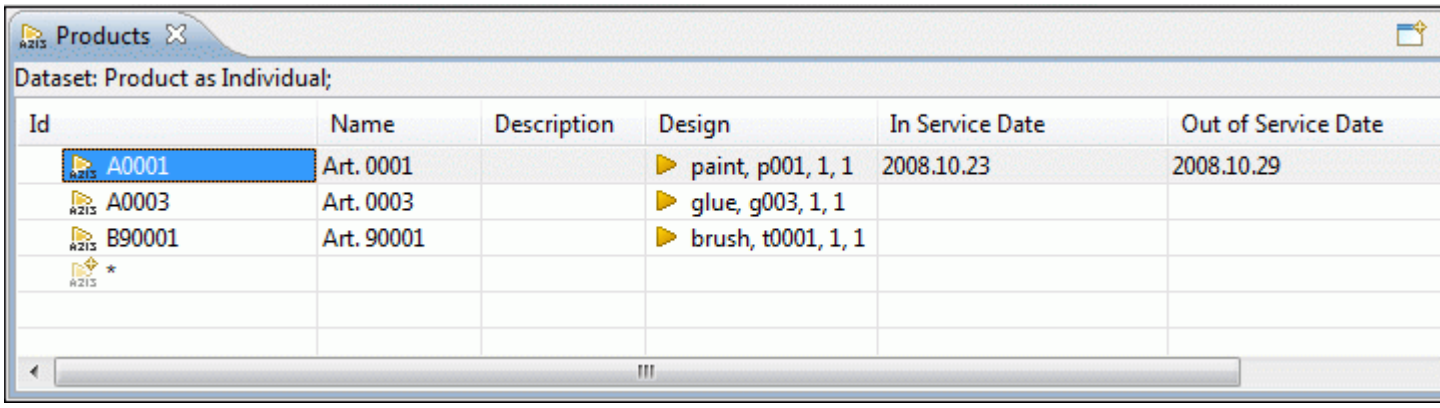

Figure 1. Products view (default columns)

To top

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9.17.2. Creating Product

Follow the routine to create a new product as individual element in the **Products** view table list (see Figure 1).

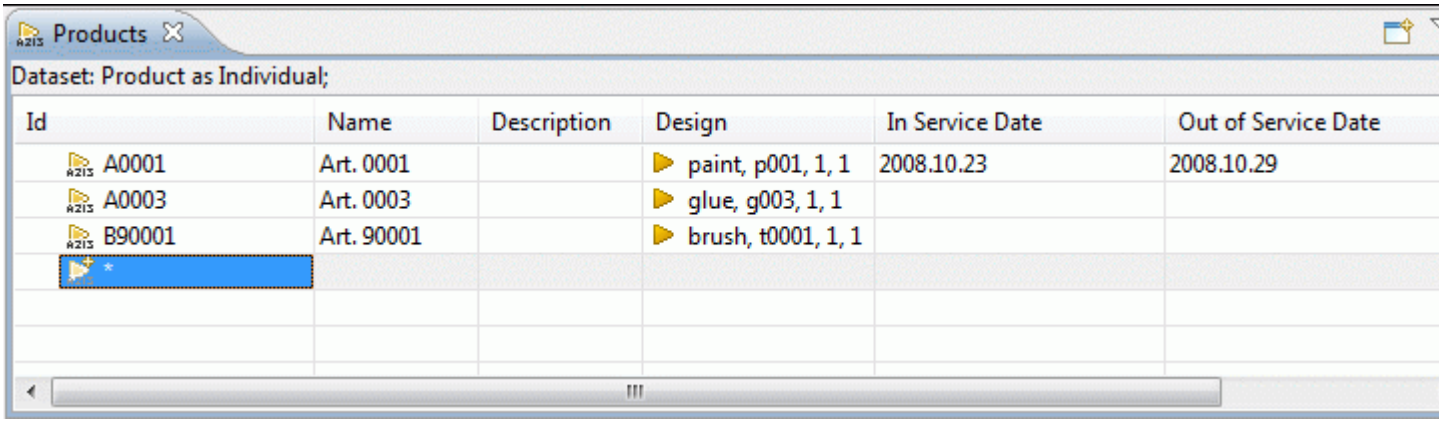

Figure 1. Products view table with highlighted auto row

- Select the auto row in the table (see Figure 1 highlighted cell in the **Id** column).
- Enter an identifier of the new product as individual element.
- Press **Enter** key. A new product as individual element appears in the **Products** view table (see Figure 2).

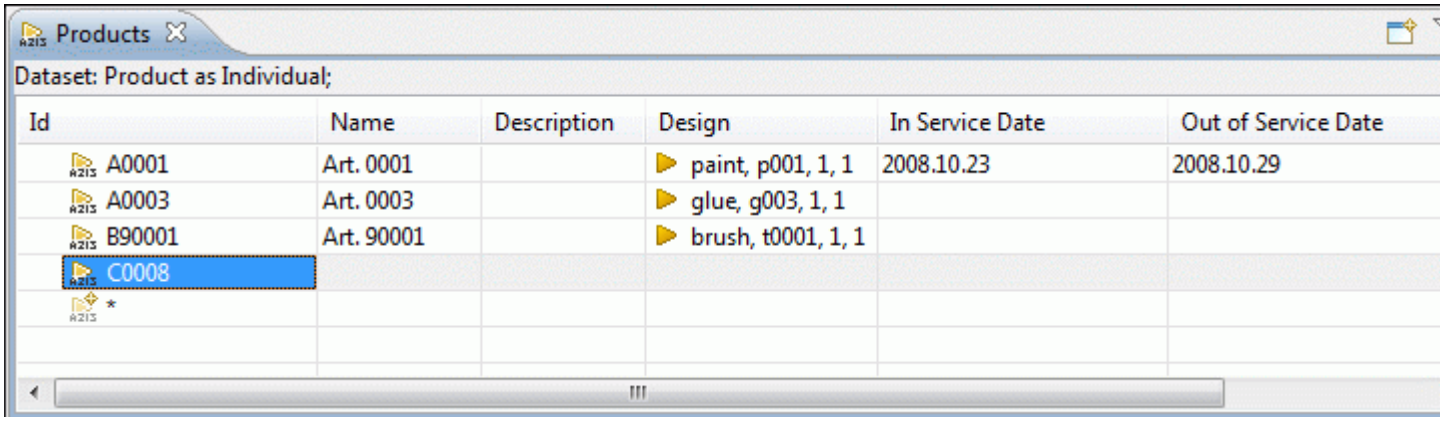

Figure 2. Created product as individual element (highlighted)

To top

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9.17.3. Editing Product Data

Activate the **Products** view. Select an applicable product as individual element in the table list. You can edit the following:

- Enter a new or edit the current product as individual identifier in the **Id** column.
- Enter the text, which defines the product as individual name into the **Name** column.
- Enter additional information about the product as individual in the **Description** column.
- The **Design** column contains the link to a part, which is a design of the current product as individual. The value of this column can be changed only by copying and pasting a particular

part element form the **Part Classification** view (if available).
- Enter a new date by typing value or select it from the pop up table in the **In Service Date** column.
- Enter a new date by typing value or select it from the pop up table in the **Out of Service Date** column.

To top

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9.17.4. Removing Product

Activate the **Products** view. Follow the routine to remove a particular product as individual element form the table:

- Select a product as individual element in the table list.
- Right click on the **Id** column.
- Select the **Delete** menu item in the appeared context sensitive menu.

The selected product as individual element is removed from the **Products** view table list.

To top

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9.18.1. States View

A *State* is the mode of being in which something does or could exist or existed for a period of time.
The **States** view presents a hierarchy of assemblies/components with the applied states (see Figure 1). The state of the parent assembly may contain a sub state, which is the state of the child assembly component (see Figure 1 - *State 1* is a state of the parent assembly and *State 2* is a state of the child assembly component).
The default column perspective contain the following columns:

- **Name** - presents a hierarchy of assemblies/components with the applied states.
- **Description** - specifies the word or group of words by which the *state* is referred to.
- **State type** - the purpose of the association of the *state* with a product.

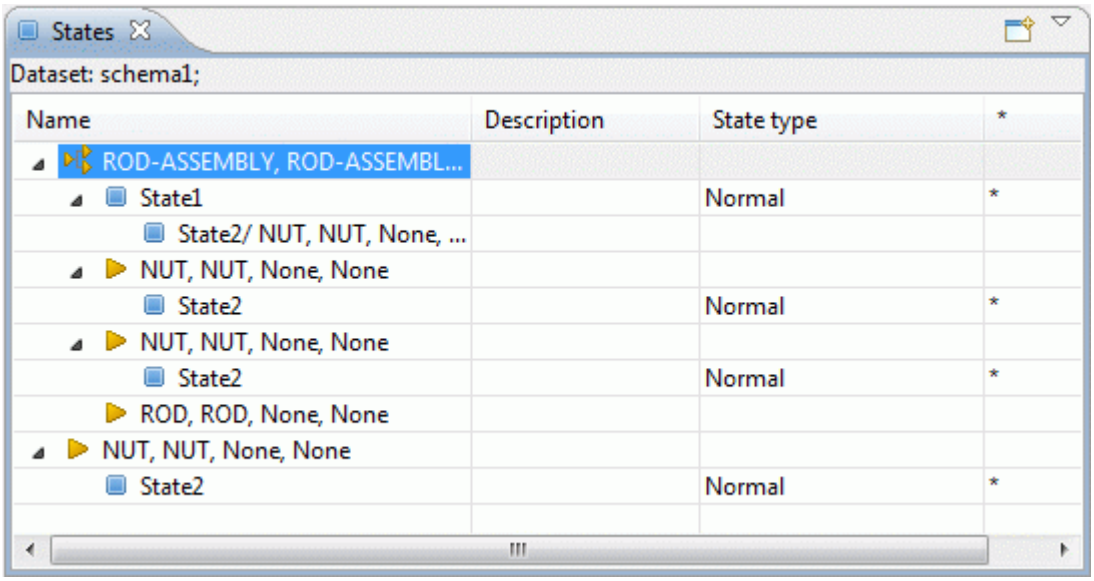

Figure 1. States view (default columns)

To top

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9.18.2. Creating State

Follow the routine to create a new state for the selected element in the **States** view:

- Right click on an applicable element (assembly or assembly component).
- Select the **New state** menu item in the appeared context sensitive menu (see Figure 1).

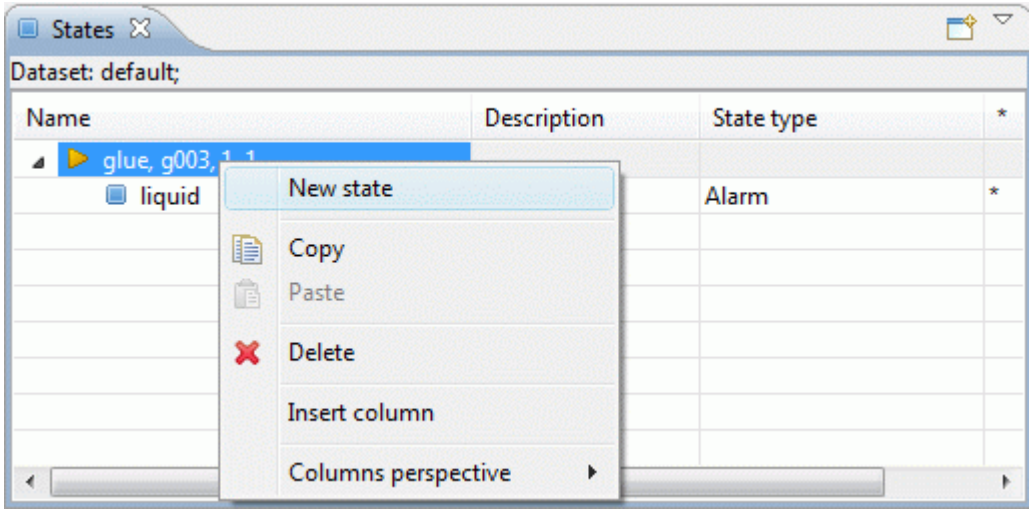

Figure 1. Context sensitive menu with selected New state menu item

A new node named 'New state' appears under the selected element (see Figure 2).

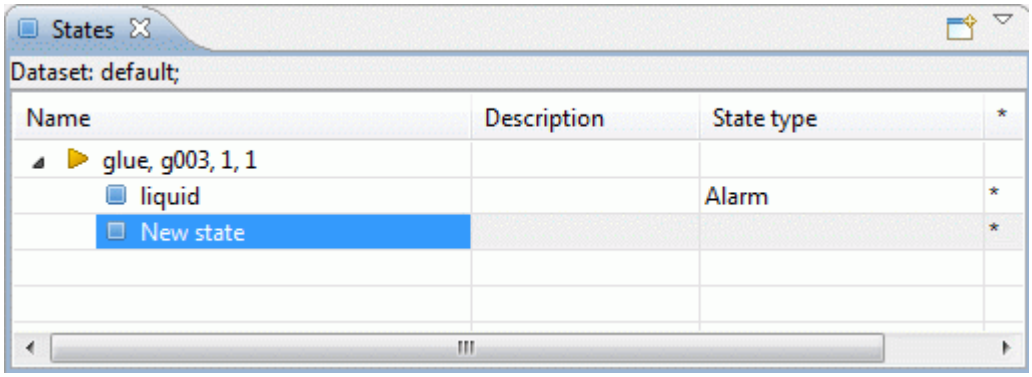

Figure 2. Created new state

To top

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9.18.3. Adding Sub State

Follow the routine to add a sub state for the parent state:

- First of all there should exist a tree of assembly components with their own states (see Figure 1).

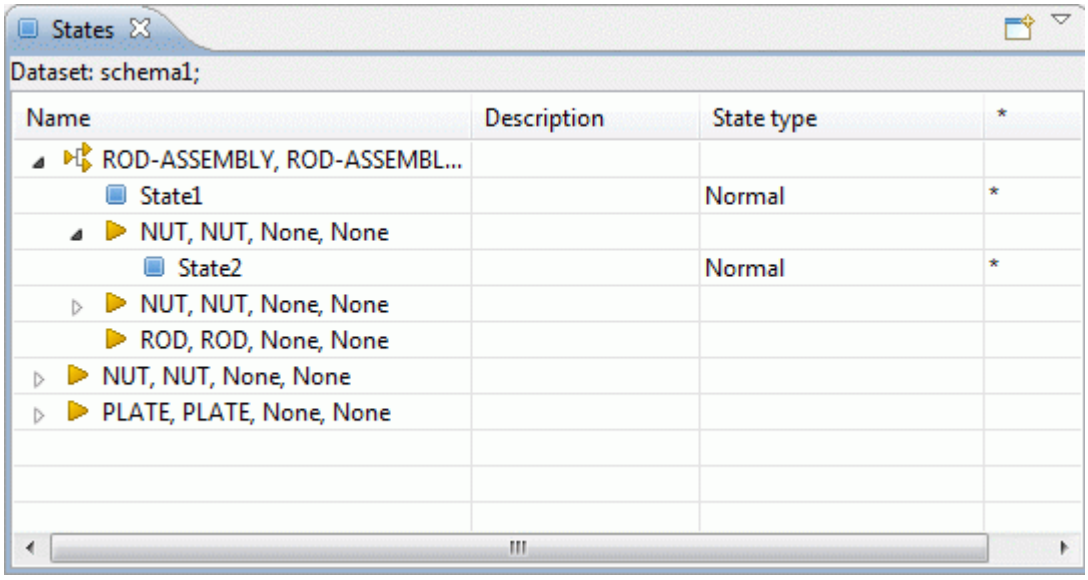

Figure 1. The tree with assembly components with their own states (State 1 and State 2)

- Select a lower level state in the tree (e.g. 'State 2' in Figure 1).
- Drag it and drop on the higher level state (e.g. 'State 1' in Figure 1). The lower level state becomes a sub state of the higher level state (see Figure 2).

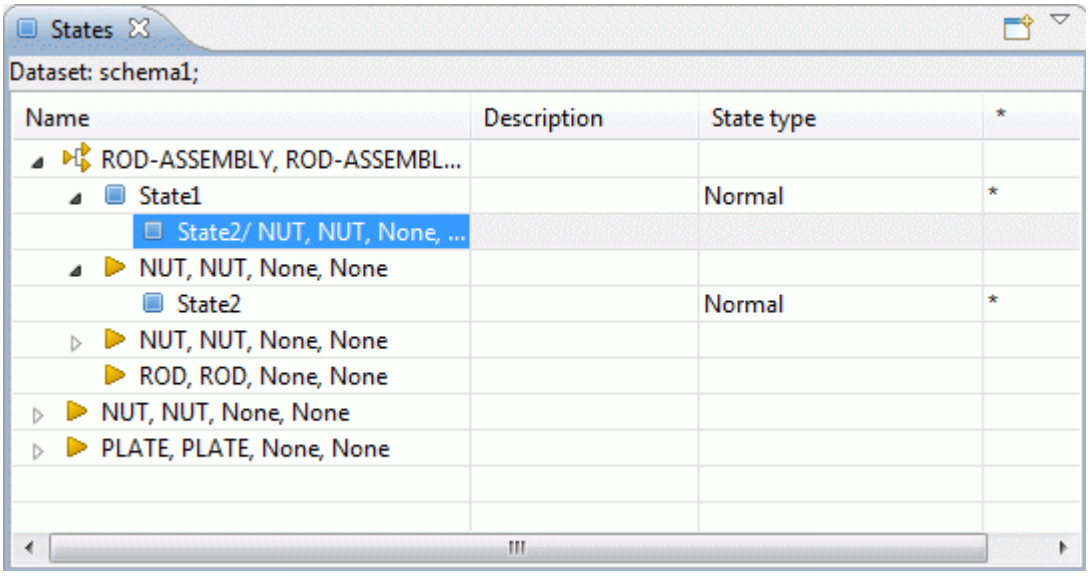

Figure 2. Sub state (selected)

To top

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9.18.4. Editing State Data

Activate the **States** view. Select an applicable state in the table list. You can edit the following:

- Enter the text, which defines the state name into the **Name** column.
- Enter additional information about the state in the **Description** column.
- Enter a new value or select predefined value from the combo box in the **State type** column.

To top

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9.18.5. Removing

Activate the **States** view. Follow the routine to remove an applicable element form the tree table:

- Select an element (assembly/ part/ state) in the tree table.
- Right click on the **Name** column.
- Select the **Delete** menu item in the appeared context sensitive menu.

The selected element is removed from the **States** view tree table.
To top

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9.19.1. Stratum Technologies View

The **Stratum technologies** view (available by selecting in the main menu bar **Window -> Show view -> Other** then under **PCB Layer Stack** node) presents a list of stratum technologies (see Figure 1). The first column presents the row element - stratum technology, other columns contain data of the particular stratum technology. The default column perspective contain the following columns:

- **Name** -specifies the word or group of words by which the stratum technology is referred to.
- **Purpose** - specifies the purpose of the current stratum technology. The predefined values are: build-up material, coating, core material, embedded passive capacitor dielectric, embedded passive resistor, generic layer, glue, gluemask, lands only, pastemask, power or ground, signal, silkscreen, soldermask or solderpaste.
- **Thickness** - specifies the stratum technology stratum thickness for a layer.
- **Min feature** - specifies the stratum technology minimum finished feature size for a layer.
- **Min Spacing** - specifies the stratum technology minimum finished feature spacing for a layer.
- **Material** - specifies the stratum technology material designation.
- **Electrical conductivity** - specifies the stratum technology conductivity classification for material identification for a layer.

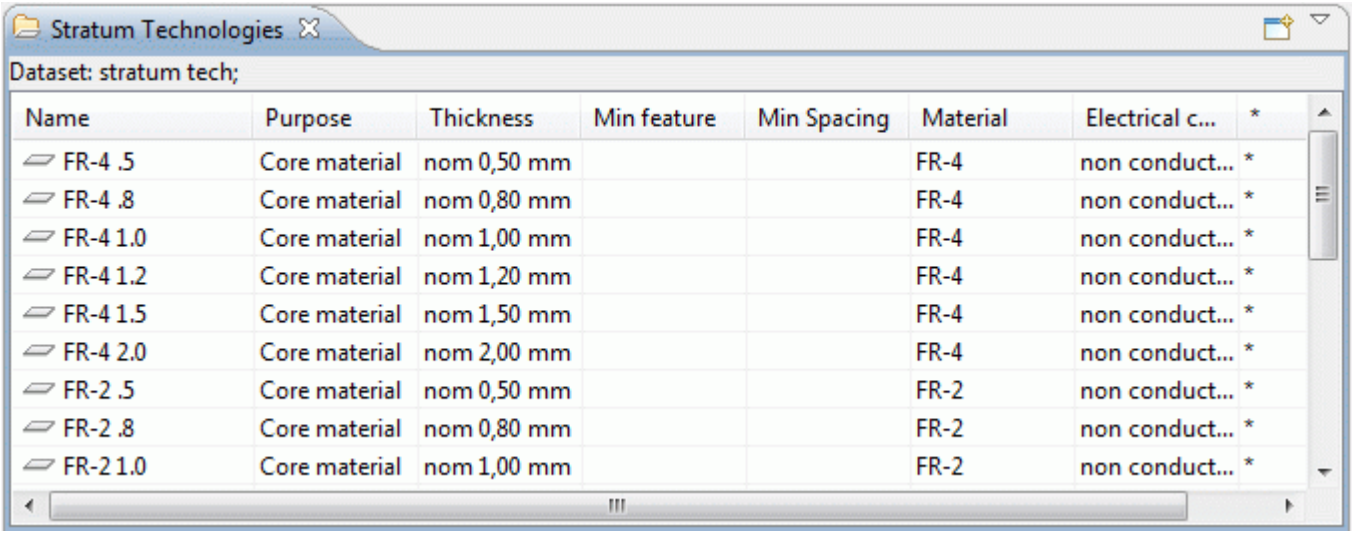

Figure 1. Stratum technologies view

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9.19.2. Creating Stratum Technology

Follow the routine to create a new stratum technology in the **Stratums Technologies** view tree table (see Figure 1):

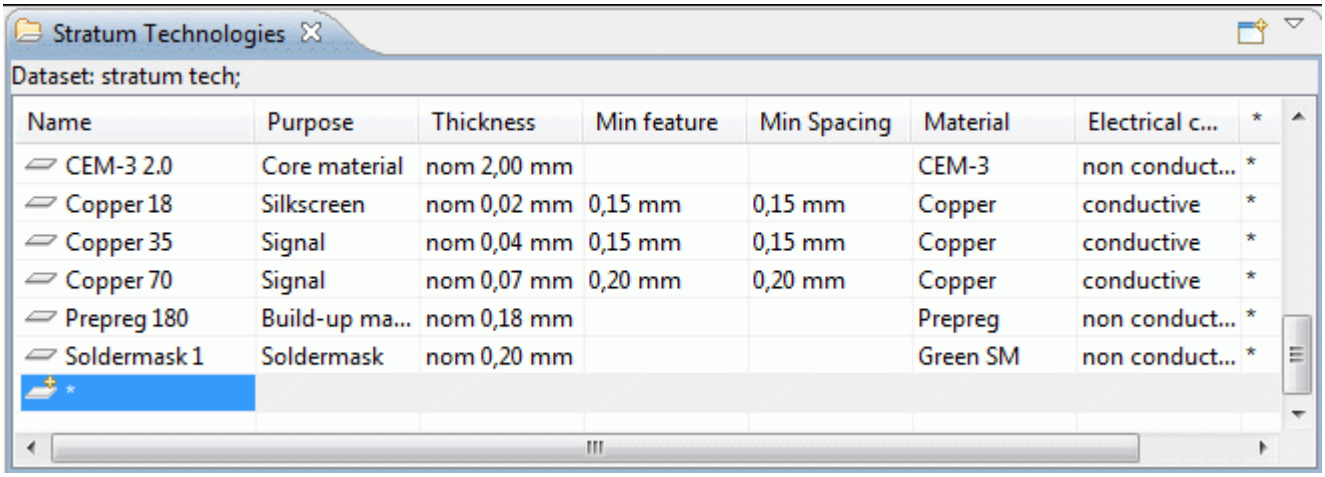

Figure 1. Stratum Technologies view (with highlighted autorow)

- Select the auto row in the table (see Figure 1 highlighted cell in the **Name** column).
- Click on the selected auto row to enter in to the edit mode.
- Enter a name for the new stratum technology.
- Press **Enter** key. A new record appears in the **Stratum Technologies** view tree table (see Figure 2). The **Purpose** column is filled with default value, which you can change by selecting a new value from the list.

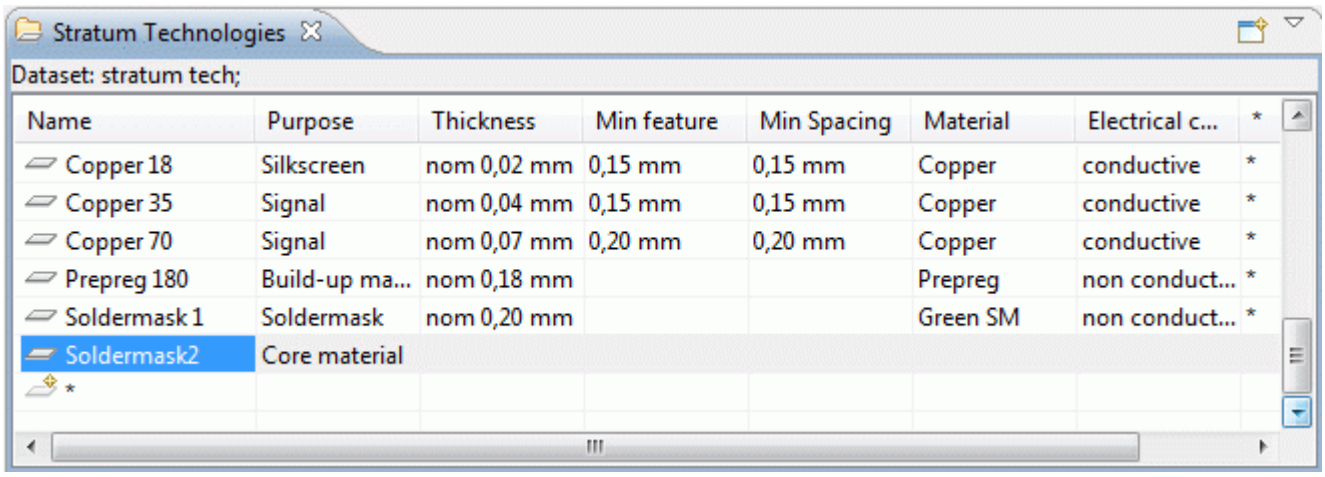

Figure 2. Created new stratum technology element (highlighted)

To top

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9.19.3. Editing Stratum Technology Data

Activate the **Stratum Technologies** view. Select an applicable stratum technology element in the tree table. You can edit the following:

- Enter a new or edit the current value in the **Name** column.
- Select another value in a combo box of the **Purpose** column. You can not enter a new value, just select an existing one.
- Enter a thickness value into the **Thickness** column. You can also select a new tolerance value or/and change units.
- Enter a value of the minimum finished feature size into the **Min Feature** column. You can also change units by selecting a new unit from the combo box.
- Enter a value of the minimum finished feature spacing into the **Min Spacing** column. The units

can be changed by selecting an applicable value in the drop down list.

- Select or enter a new value of a material designation into the **Material** column.
- Select an applicable value of the electrical conductivity classification in the **Electrical conductivity** column. The selected value is associated to the current material designation, presented in the **Material** column.

To top

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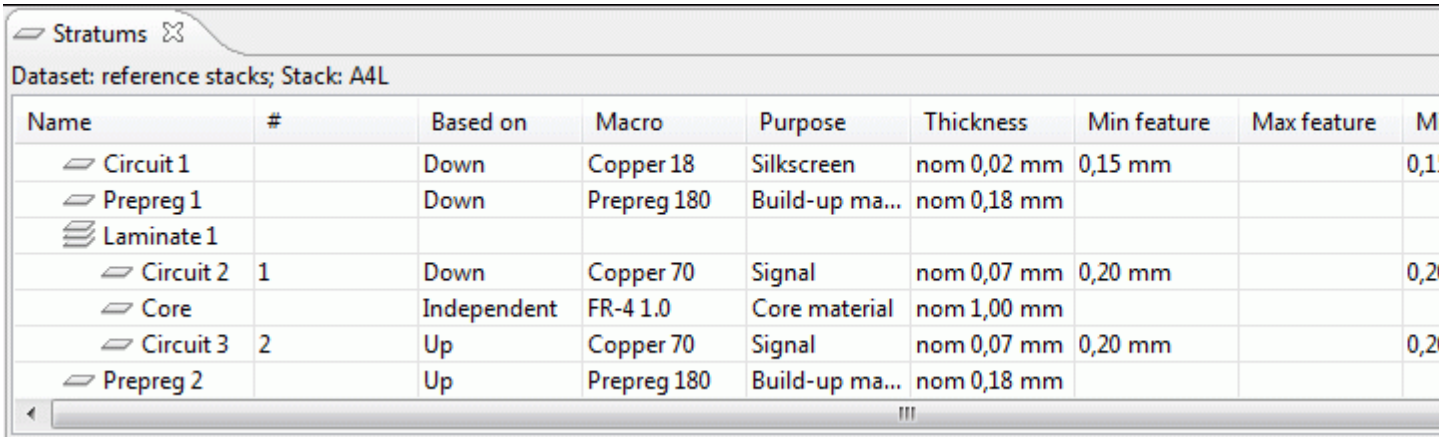
9.20.1. Stratums View

The **Stratums** view (available by selecting in the main menu bar **Window -> Show view -> Other** then under **PCB Layer Stack** node) presents a tree table with a hierarchy of layers, referenced by a particular design stack model, which is identified in the top section of the **Stratums** view window (see Figure 1).

Note: The data appears in the **Stratums** view only after an applicable design stack model is selected in the **Design Stack Models** view.

The default perspective contains the following columns:

- **Name** - specifies the word or group of words by which the layer is referred to.
- **#** - specifies the stratum technology occurrence number for the layer. The value depend on the selected *Purpose* of the layer.
- **Based on** - specifies the base of stratum technology occurrence for the layer. The possible values are: down, independent and up.
- **Macro** - specifies the stratum technology occurrence definition for the layer.
- **Purpose** - specifies the purpose of the current layer. The predefined values are: build-up material, coating, core material, embedded passive capacitor dielectric, embedded passive resistor, generic layer, glue, gluemask, lands only, pastemask, power or ground, signal, silkscreen, soldermask and solderpaste.
- **Thickness** - specifies the stratum technology stratum thickness for the layer.
- **Min feature** - specifies the stratum technology minimum finished feature size for the layer.
- **Max feature** - specifies the stratum technology maximum feature size requirement for the layer.
- **Min Spacing** - specifies the stratum technology minimum finished feature spacing for the layer.
- **Electrical conductivity** - specifies the stratum technology conductivity classification for material identification for the layer.



Name	#	Based on	Macro	Purpose	Thickness	Min feature	Max feature	M
Circuit 1		Down	Copper 18	Silkscreen	nom 0,02 mm	0,15 mm		0,1
Prepreg 1		Down	Prepreg 180	Build-up ma...	nom 0,18 mm			
Laminate 1								
Circuit 2	1	Down	Copper 70	Signal	nom 0,07 mm	0,20 mm		0,2
Core		Independent	FR-4 1.0	Core material	nom 1,00 mm			
Circuit 3	2	Up	Copper 70	Signal	nom 0,07 mm	0,20 mm		0,2
Prepreg 2		Up	Prepreg 180	Build-up ma...	nom 0,18 mm			

Figure 1. Stratums view

To top

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9.20.2. Stratums hierarchy

The **Name** column in the **Stratums** view tree table presents the hierarchy of layers (see Figure 1):

- **Root node** - the layer or sequential group name.
- **Second and lower level nodes** - layers or sequent elements grouped into particular sequential group.

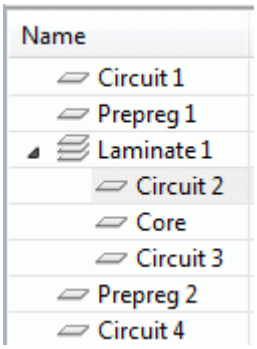

Figure 1. Stratums hierarchy

To top

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9.20.3. Creating Layer

Follow the routine to create a new layer in the **Stratums** hierarchy (see Figure 1):

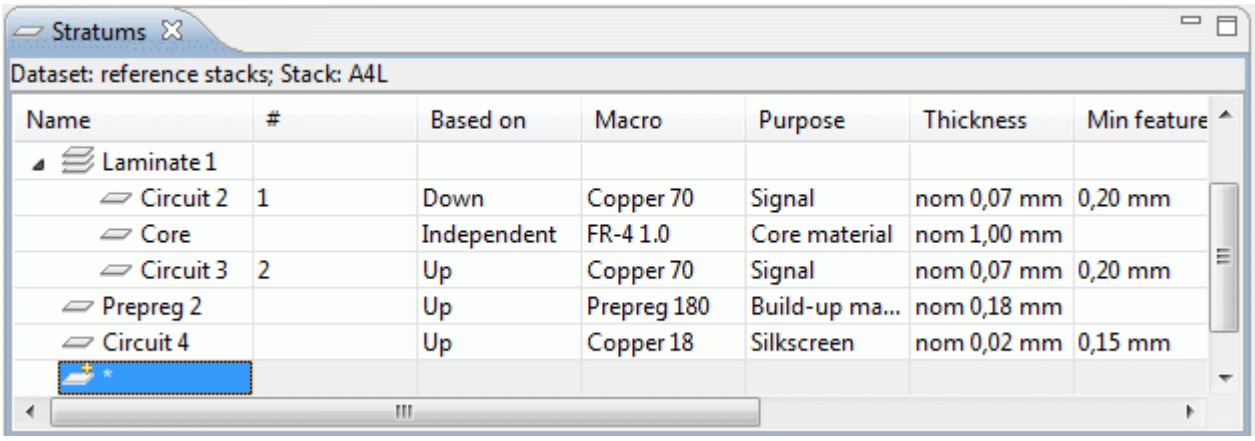

Figure 1. Stratums view, hierarchy of layers with highlighted autorow

- Select an autorow (highlighted node in Figure 1).
- Click on the selected autorow to enter into the edit mode.
- Enter a name for a new layer.
- Press **Enter** key.

If you want to include a new layer into a particular sequential group:

- Right click on a sequential group node.
- Select the **Insert element** menu item in the context sensitive menu (see Figure 2).

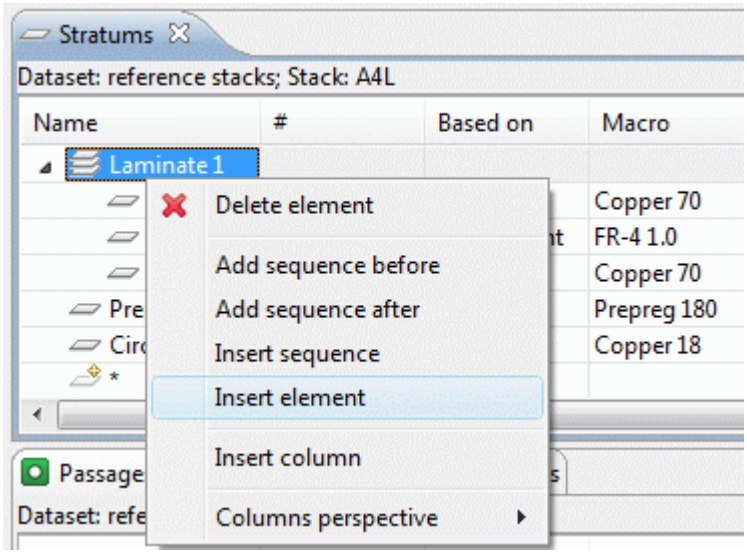

Figure 2. Context sensitive menu, selected Insert element menu item

- The autorow appears under the selected sequential group node. Follow the previously described steps to create a new layer.

The created layer node appears in the **Startums** tree table.

To top

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9.20.4. Creating Sequential Group

There are several possibilities to add a new sequential group in the **Stratums** hierarchy:

- Add sequential group before the selected element (layer, sequential group);
- Add sequential group after the selected element (layer, sequential group);
- Insert sequential group as a child for the selected sequential group.

Add sequence before

Follow the routine to add a sequential group before the selected element:

- Select a layer or a sequential group node in the tree table.
- Right click and select the **Add sequence before** menu item in the appeared context sensitive menu (see Figure 1).
- A new node, named '*New group*' appears before the selected element node.

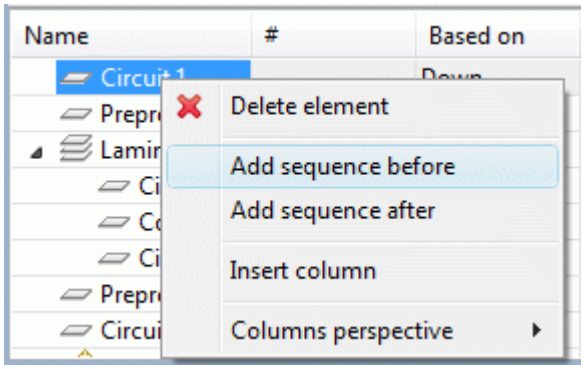

Figure 1. Context sensitive menu with selected *Add sequence before* menu item

To top

Add sequence after

Follow the routine to add a sequential group after the selected element:

- Select a layer or a sequential group node int the tree table.
- Right click and select the **Add sequence after** menu item in the appeared context sensitive menu (see Figure 2).
- A new node, named '*New group*' appears after the selected element node.

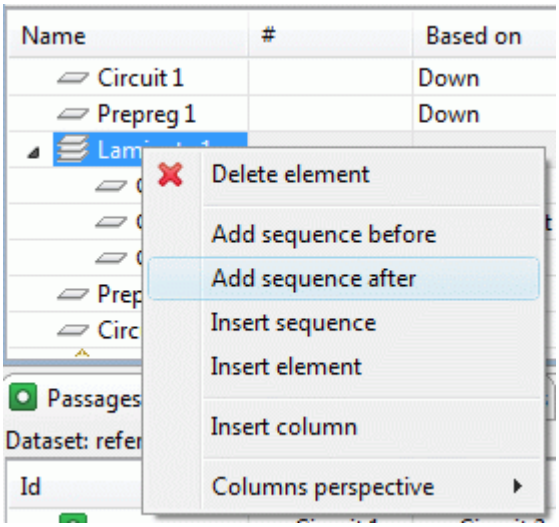

Figure 2. Context sensitive menu with selected *Add sequence after* menu item

To top

Insert sequence

Follow the routine to insert a new sequential group under the selected sequential group node:

- Select a sequential group node in the table tree.
- Right click and select the Insert sequence menu item in the appeared context sensitive menu (see Figure 3).
- A new node, named '*New group*' appears beneath the selected sequence group node.

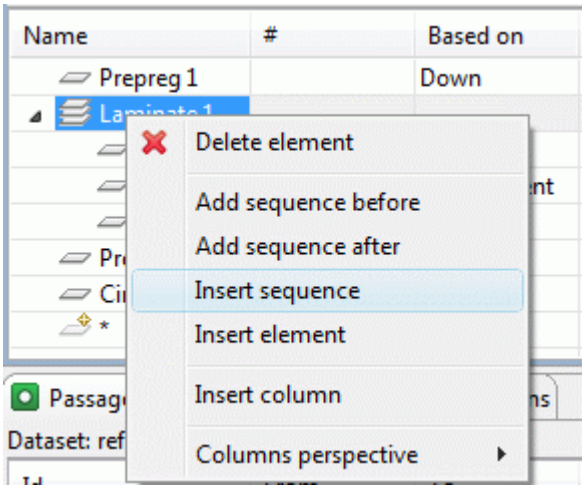

Figure 3. Context sensitive menu with selected *Insert sequence* menu item

To top

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9.20.5. Changing Layer Location

Follow the routine to change the selected layer position in the **Stratums** hierarchy:

- Select an applicable element (layer or sequential group of layers) node in the table tree.
- Drag the selected node and drop it in the desired destination place. The possible destination place is marked with the horizontal line (see Figure 1).
- The selected object appears in the selected destination place.

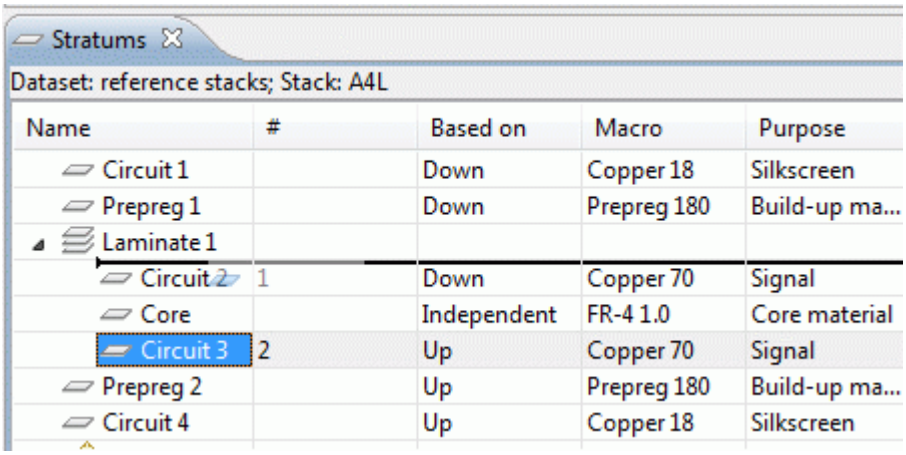

Figure 1. Selected layer is dragged to the destination place, which is marked as horizontal line

To top

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9.20.6. Moving Autorow to Root

After inserting an element into a sequential group, the autorow remains under the sequential group node

(see Figure 1). To insert a root element, the autorow must appear as a root node in the tree table. This can be achieved by using function *Move to root*.

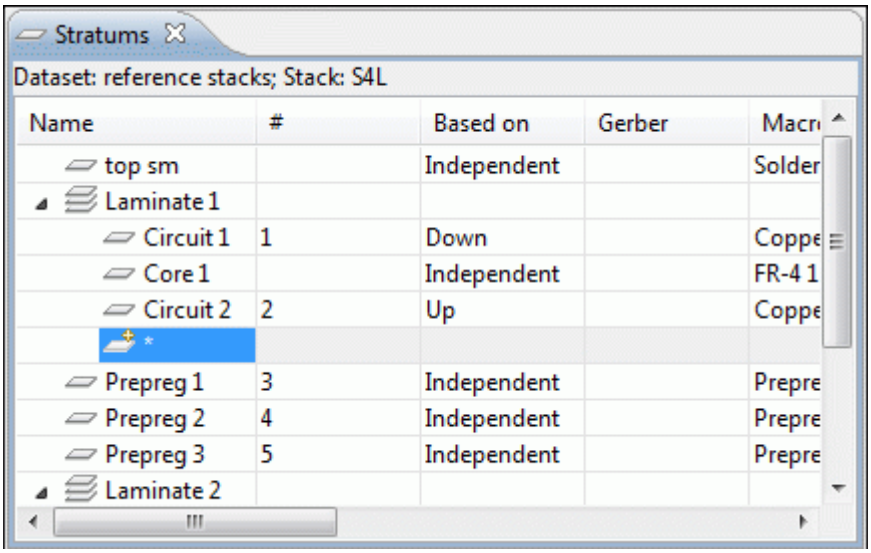

Figure 1. Stratums hierarchy with autorow under the sequential group node

- Right click on the autorow in the tree table. A context sensitive menu appears.
- Select **Move to root** menu item (see Figure 2).

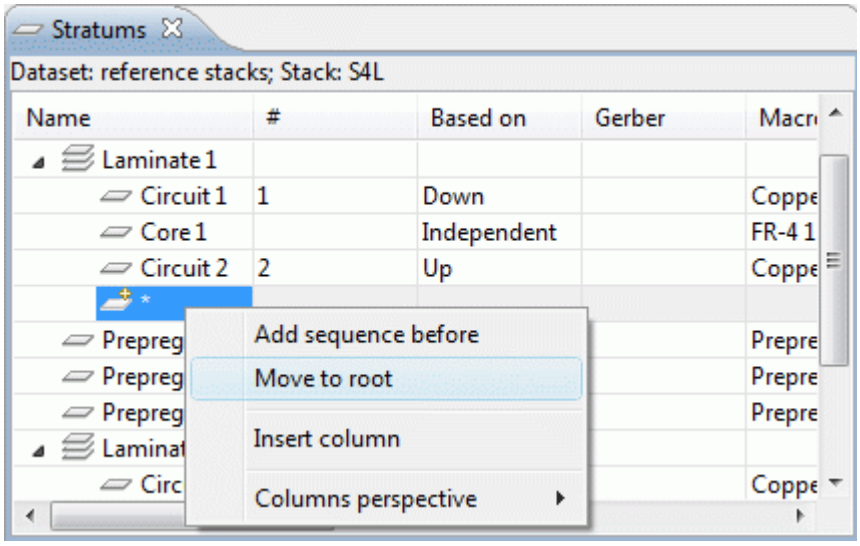

Figure 2. Context sensitive menu with selected Move to root menu item

Right after the above mentioned menu item is selected, the autorow appears as a root node in the tree table (see Figure 3).

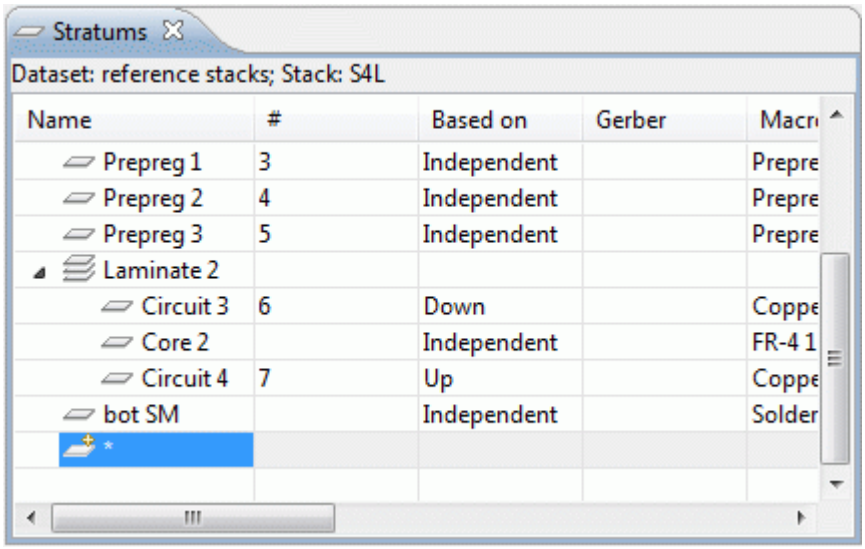

Figure 3. Autorow as root node in the tree table

To top

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9.20.7. Editing Layer Data

Activate the **Stratums** view. Select and applicable layer in the tree table. You can edit the following:

- Enter the text, which defines the layer name into the **Name** column.
- Select the value of the base of stratum technology occurrence for the layer in the **Based on** column r/o combo box.
- Select the stratum technology occurrence definition in the **Macro** column combo box. Also you can enter a new value into this field. A new stratum technology element will be created.

  Note. If you select an existing value (coming from **Stratum Technologies** view) in the **Macro** combo box, then some columns are filled with data automatically.

- Select the purpose for the layer in the **Purpose** column r/o combo box.
- Enter the thickness value of the current layer into the **Thickness** column.
- Enter the stratum technology minimum finished feature size value for the layer into the **Min feature** column.
- Enter the stratum technology maximum feature size requirement value for the layer into the **Max feature** column.
- Enter the stratum technology minimum finished feature spacing value for the layer **Min spacing** column.
- Select an applicable electrical conductivity value in the **Electrical conductivity** column r/o combo box.

To top

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9.21.1. Stratums Cross-section View

The **Stratums Cross-section** view (available by selecting in the main menu bar **Window -> Show view -> Other** then under **PCB Layer Stack** node) presents the visual 2D presentation of objects, defined in **Stratums** and **Passages** views for the particular design stack model, selected in the **Design Stack Models** view (see Figure 1). The referenced design stack model is identified in the top section of the view window. The layers and passages are presented on the left side of the view window.

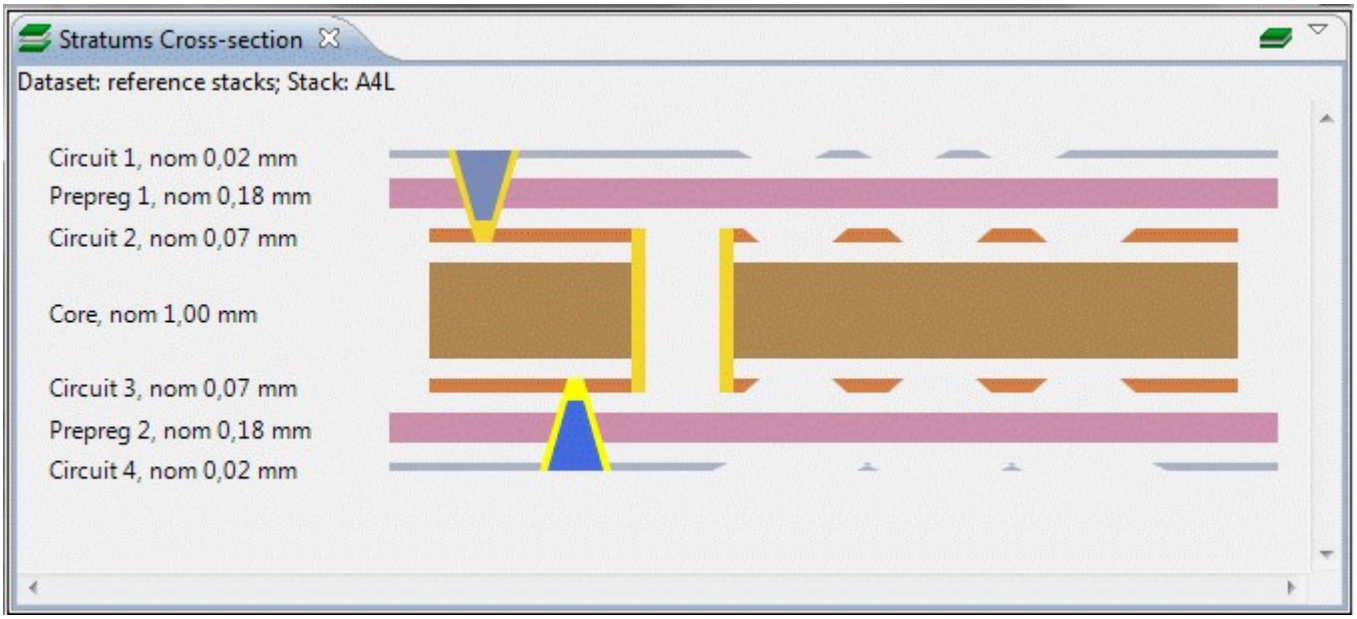

Figure 1. Stratums cross-section view

The **Stratums Cross-section** view has it's own tool bar on the top right corner of the page.

Button icon	Description
	Toggle Collapsed/Exploded view
	Open menu to select color schema

To top

Menu

There is a possibility to change the color schema for the stratums cross-section view by using Click on the button on the top right corner of the **Stratums Cross-section** view window.

To top

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9.21.2. Toggling Collapsed/Exploded View

By default the objects in the **Stratums Cross-section View** are presented in exploded view - each

layer is presented separately (see Figure 1).

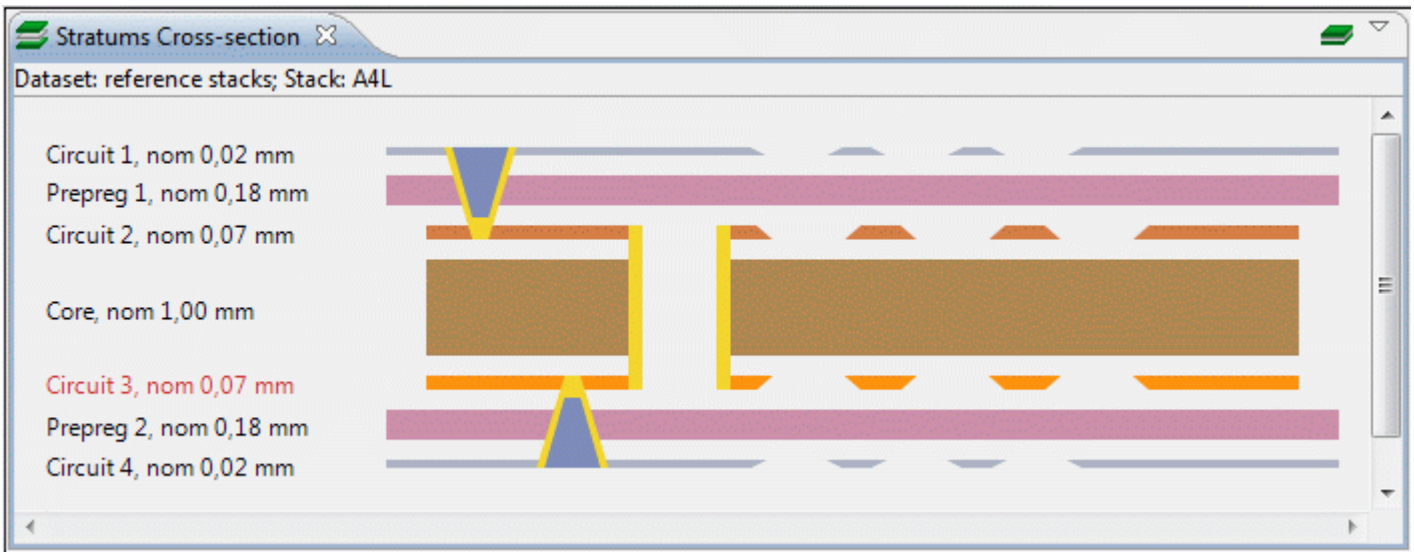

Figure 1. Stratums Cross-section exploded view

To toggle the collapsed view (see Figure 2) click on the **Toggle Collapsed/Exploded view** button in the tool bar.

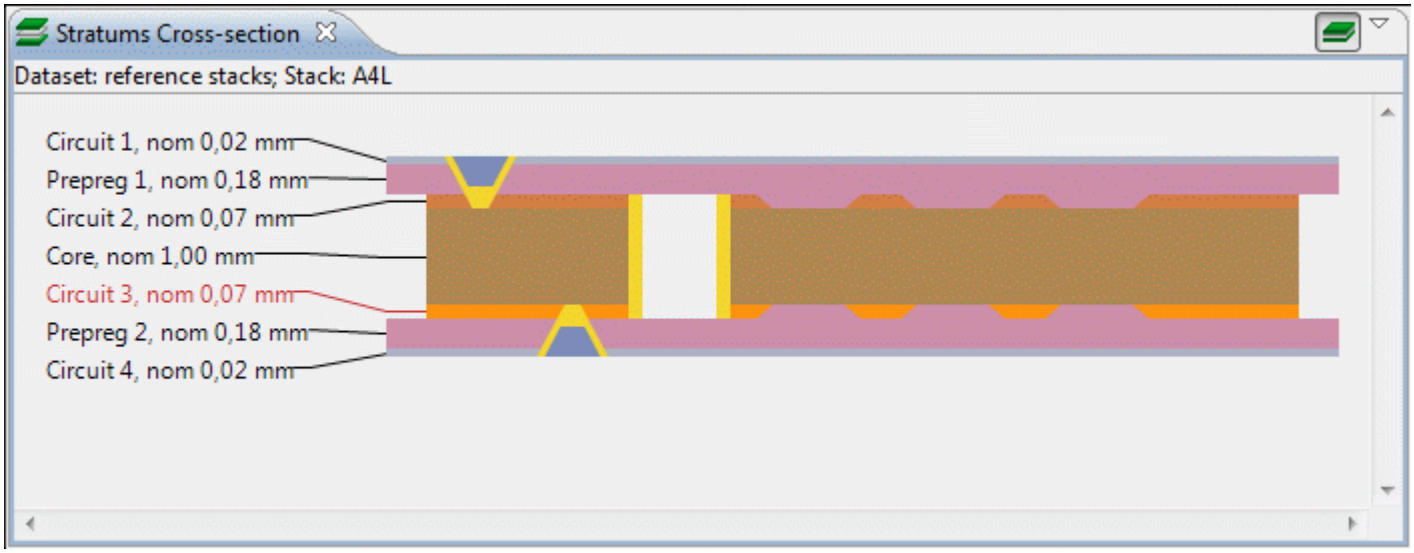

Figure 2. Stratums Cross-section collapsed view

To top

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9.21.3. Changing Color Schema

There is a possibility to change the color of components, text and background for in the **Stratums cross-section** view.
To change the color schema directly in the **Stratums Cross-section** view click on the button on the top right corner of the **Stratums Cross-section** view window. The following schemas are available by default: Default (see Figure 1), Red highlighting (see Figure 2), Darklighting (see Figure 3) and Greyscale (see Figure 4). The default colors for each available schema are set in the **Color Schemas** preferences (select in the main menu **Window -> Preferences -> Stratums Cross-section -> Color Schemas**). A new color schema can be created in the **Color Schemas** preferences page.

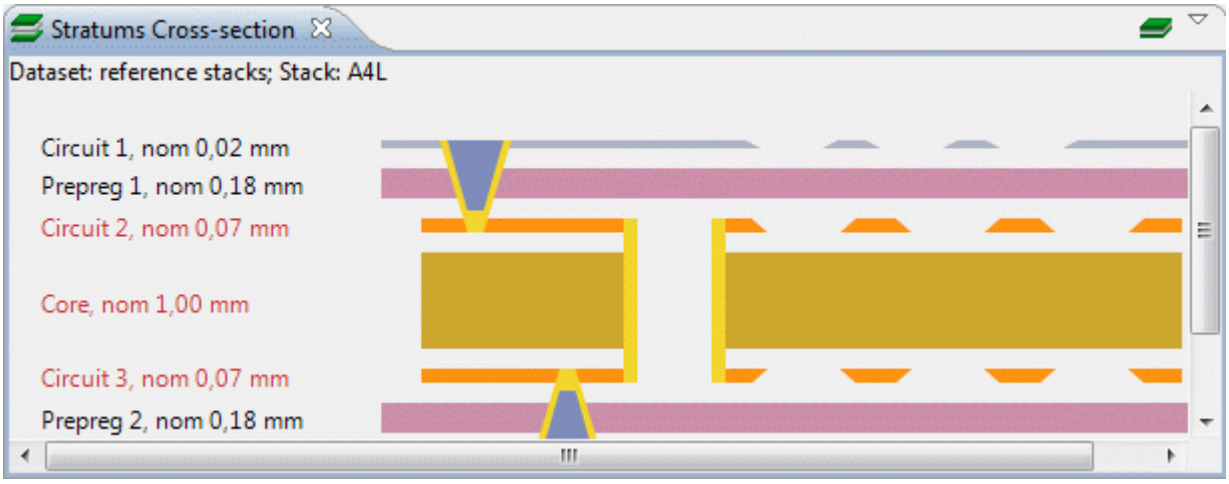

Figure 1. Default color schema view with highlighted components

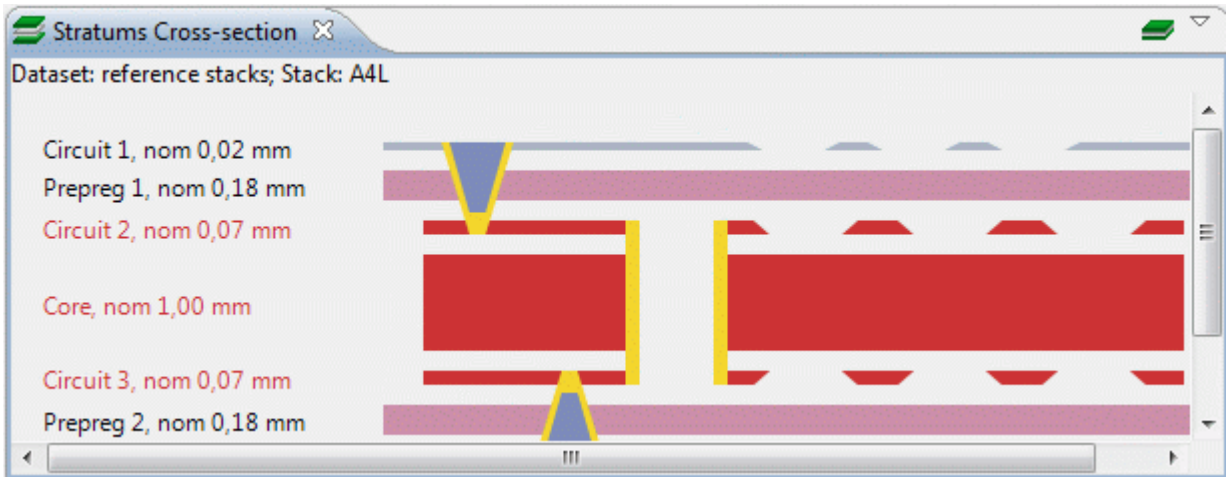

Figure 2. Red highlighting view with highlighted components (red color)

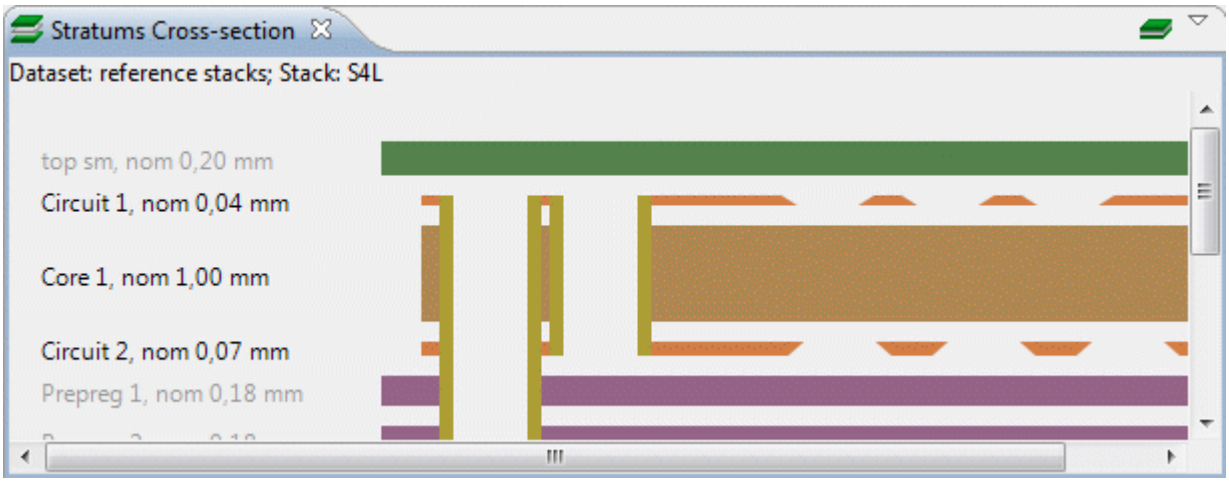

Figure 3. Darklighting view with highlighted components (dark color)

Figure 4. Greyscale view with highlighted components (black color)

To top

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9.21.4. Highlighting Components

You can highlight an applicable layer or a group of layers or a passage in the **Stratums Corss-section** view (see Figure 1). A name and a thickness parameters of the selected layer are highlighted together a 2D presentation.

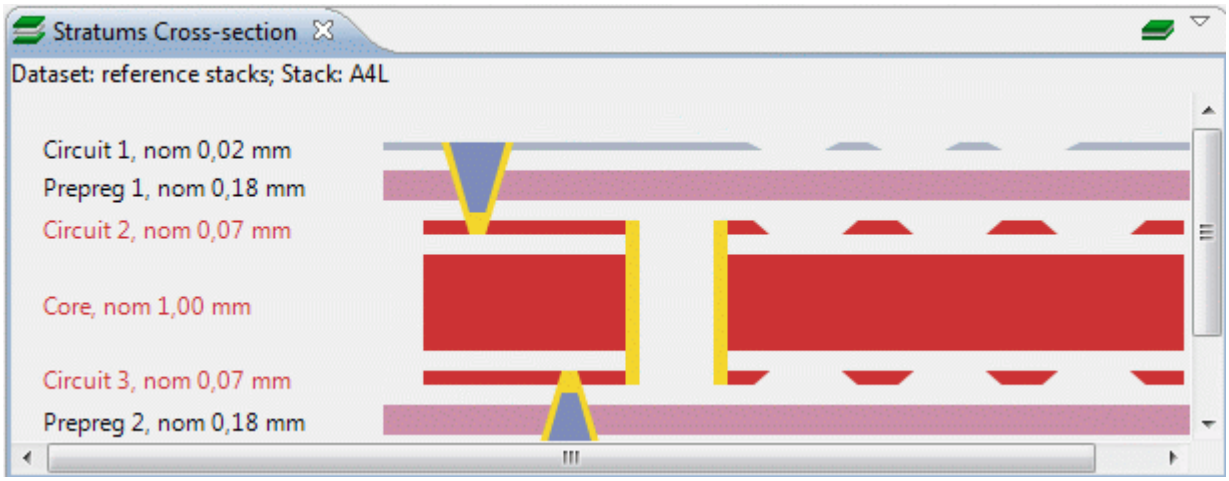

Figure 1. Stratums Cross-section view with highlighted group of layers (red color)

To top

Highlighting layer or group of layers

Follow the routine to highlight an applicable layer or group of layers in the **Stratums Corss-section** view:

- Activate the **Stratums** view on the screen.
- Click on an applicable layer or group of layer row in the tree table.
- The particular layer or group of layers is highlighted in the Stratums Cross-section view (see Figure 2 and Figure 3).

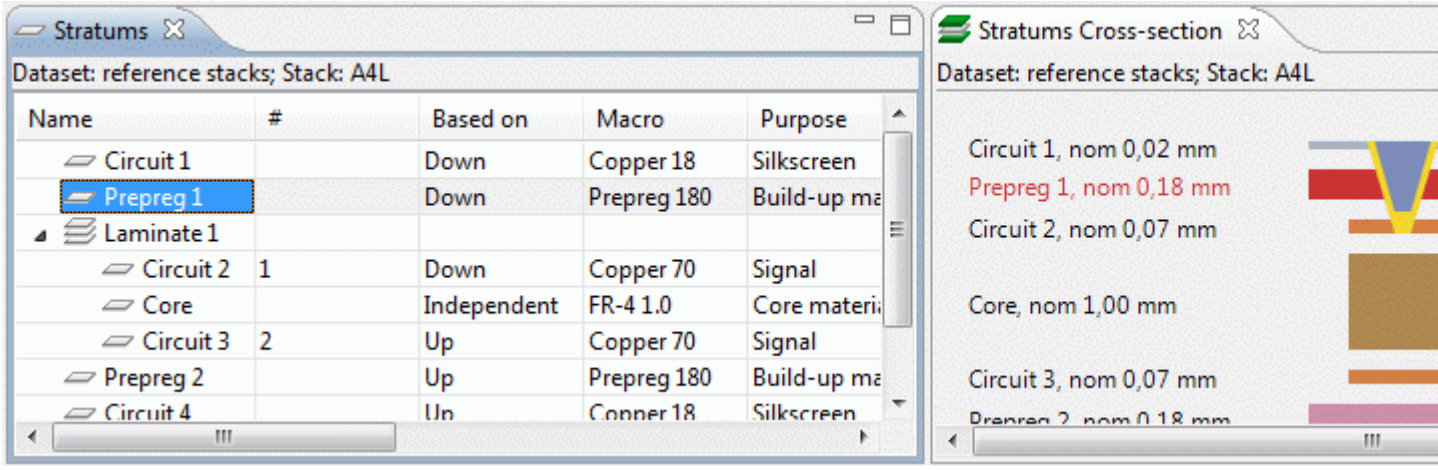

Figure 2. Stratums view with selected layer and Startums Cross-section view with the highlighted layer

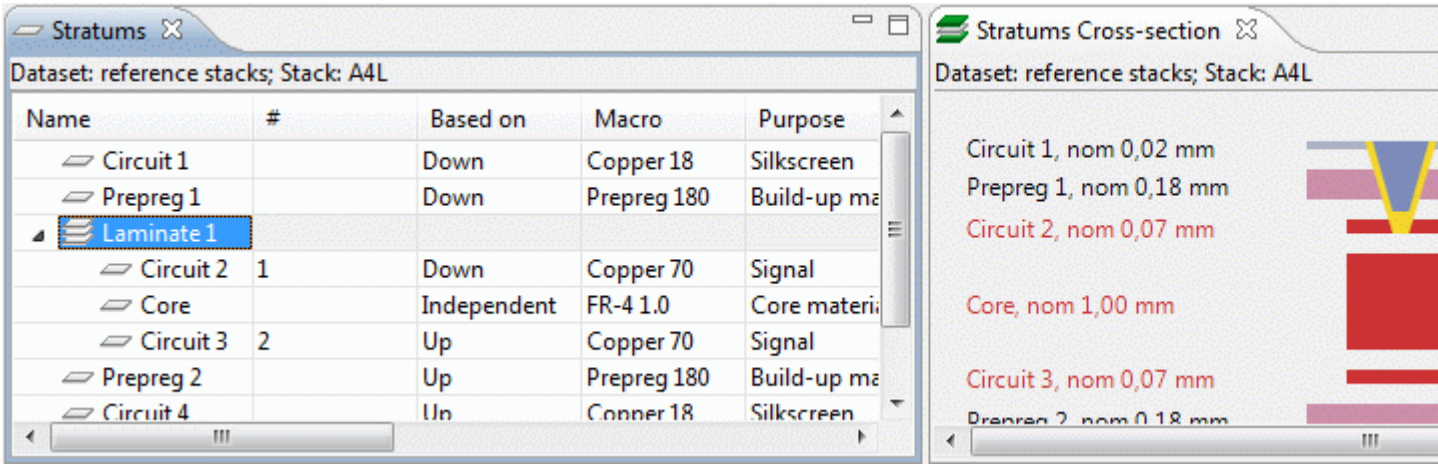

Figure 3. Stratums view with selected sequential group of layers and Startums Cross-section view with the highlighted layers

To top

Highlighting passage

Follow the routine to highlight an applicable passage in the **Stratums Corss-section** view:

- Activate the **Passages** view on the screen.
- Click on an applicable passage row in the tree table.
- The particular passage is highlighted in the Stratums Cross-section view (see Figure 4).

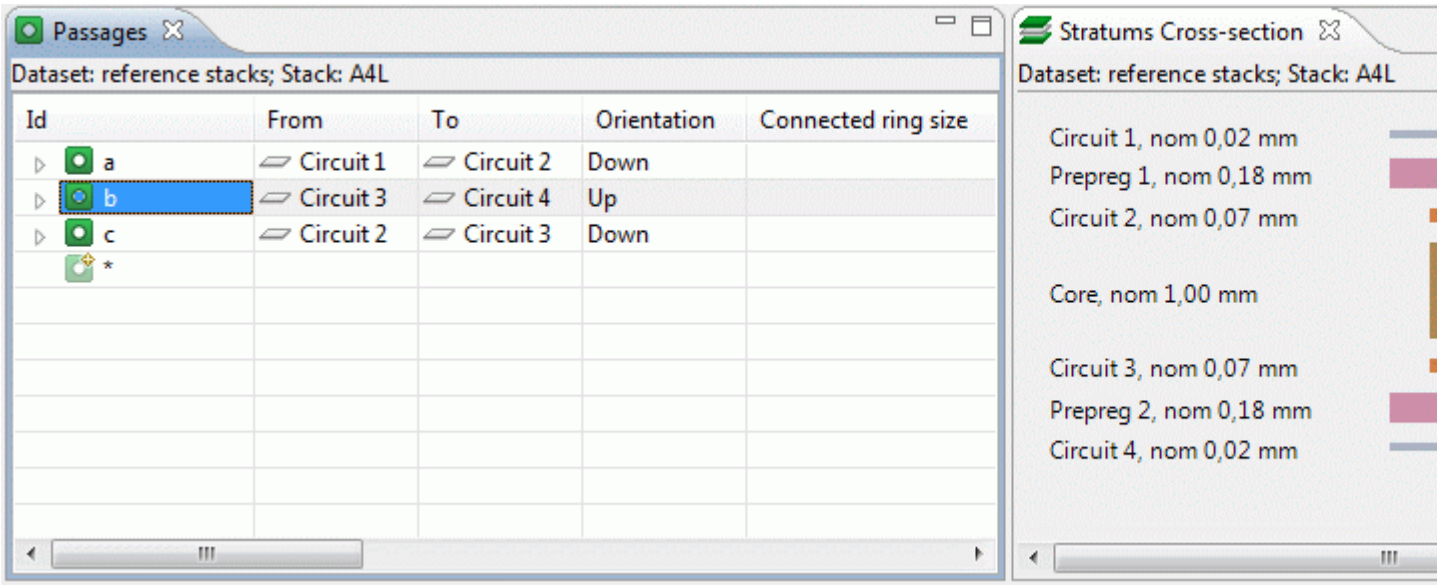

Figure 4. Stratums view with selected passage and Startums Cross-section view with the highlighted passage (red color)

To top

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9.22.1. Tasks

A *task* represents data that specifies task methods, that is the method specified for a particular activity. The task method is the combination of something to be achieved and , optionally, a way to achieve it. The same result can be achieved through more than one method. A task method can be defined as simple text or as a processible structure using repeats, sequences and decision points. A task method can be associated with the resources required to do it.

The **Tasks** window presents a tree table with a task hierarchy, which is available for the selected dataset (see Figure 1).

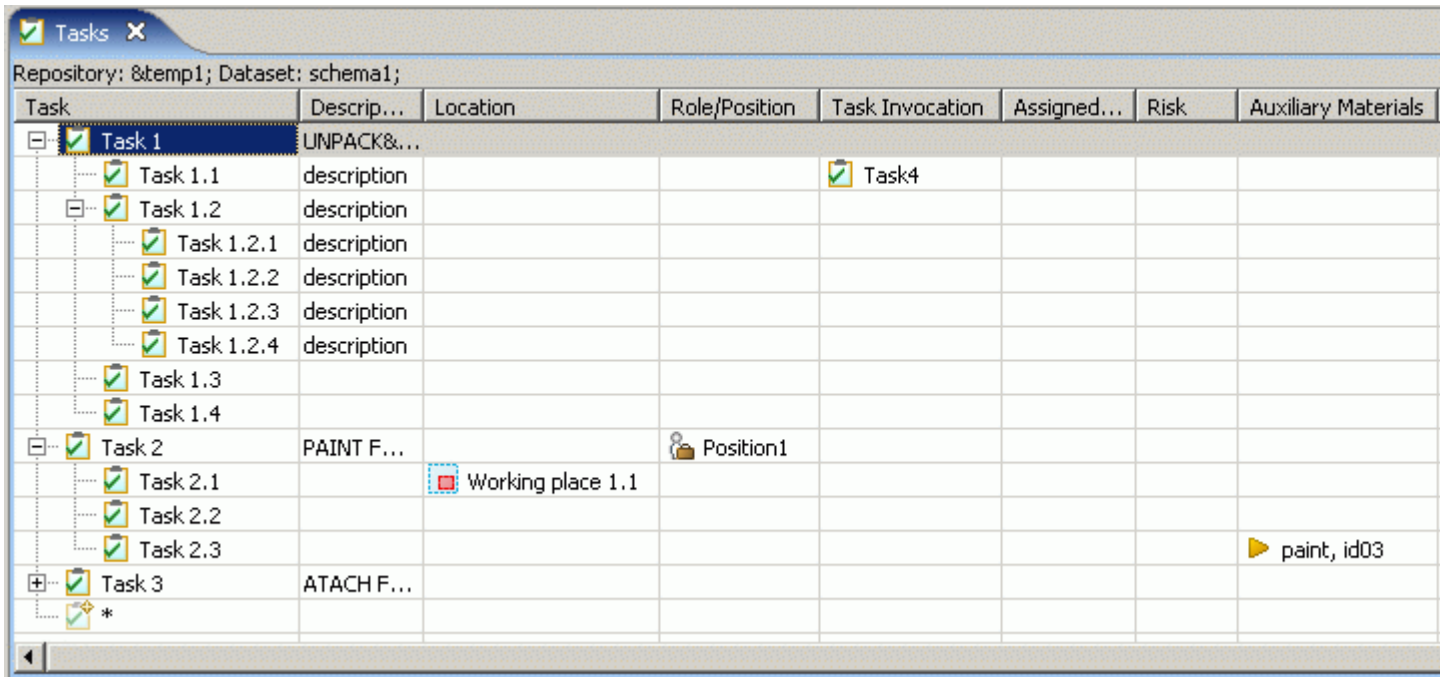

Figure 1. Tasks window

To top

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9.22.2. Task Hierarchy

The **Task** column in the Tasks window presents the hierarchy of tasks (see Figure 1):

- **Root nodes** - tasks, that are independent from each other.
- **Second and further level nodes** (branches and leaves) - sub tasks, which have be performed in a particular order. The next task can be started only after the previous one was finished.

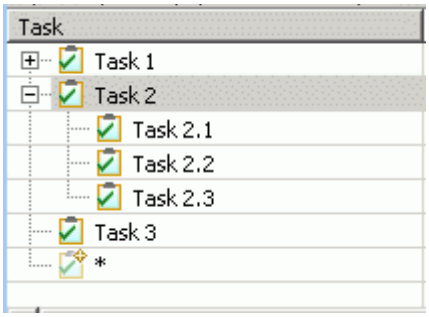

Figure 1. Task hierarchy

To top

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9.22.3. Editing Task Data

Activate a Tasks window to edit data. Select a task you want to edit in a task hierarchy. You can perform the following editing actions:

- Enter the text, which defines the task name into the **Task** column.
- Enter the text into the **Description** column.
- Assign zonal breakdown in the **Location** column.
- Assign role or position in the **Role/Position** column.
- Assign the referenced task in the **Task Invocation** column.
- Select the documents to be associated with the selected task by using the **Browse** button in the **Assigned documents** column.
- <...TBD ...> in the **Risk** column.
- Assign auxiliary materials in the **Auxiliary materials** column.
- Enter the value and select the unit from the combo box in the **Duration** column.
- Assign tools in the **Tools** column.
- Enter the value and select the unit from the combo box in the **Adjustment** column.
- Assign product or part in the **Product/Part** column.

To top

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9.22.4. Creating Task

- Creating Top Level Task
- Creating Task Element
  - Add Task Before
  - Add Task After

There is a possibility to create a new task in a Task hierarchy.

Creating Top Level Task

You can create a new top level task by using the autorow:

- Ensure that the autorow is in the right position - the last row in the tree table, if it is not - drag and drop it at the end of the tree table.
- Enter data into the autorow.
- Press **Enter** key. The new task is created as the root node in the tree.

To top

Creating Task Element

You can add an element for a created task:

- Drag and drop the autorow onto the preferable task.

Or

- Right click on a parent task and select **Add child** menu item in the appeared context sensitive menu.
- Enter data into the autorow.
- Press **Enter** key. The new task is added.

Note. The task will be added at the end of the tree branch list (the same level node).
You can add a task into particular position - before or after the selected task

Adding Task Before

Follow the routine to add a task before the selected task:

- Drag and drop the autorow into an applicable position.

Or

- Right click on a task and select **Add before** menu item in the appeared context sensitive menu.
- Enter data into the autorow.
- Press **Enter** key. The new task is inserted before the selected task.

Adding Task After

Follow the routine to add a task after the selected task:

- Drag and drop the autorow into an applicable position.

Or

- Right click on a task and select **Add after** menu item in the appeared context sensitive menu.
- Enter data into the autorow.
- Press **Enter** key. The new task is inserted after the selected task.

To top

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9.22.5. Moving

You can move an element by dragging and dropping the selected tree table row:

- **Moving task** - to change task location in a hierarchy. The same result can be achieved by cutting and pasting an element.
- **Moving auto row** - to create a new task in the particular place of the task hierarchy.

To top

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9.22.6. Removing

There is a possibility to remove a redundant or an incorrect task from a task hierarchy (tree table). **Note**: you can remove only those tasks, which have no tasks beneath (child nodes). Follow the routine to remove a task form the tree table:

- Right click on a task node in the **Task** column.

Or

- Right click on the any column of an applicable task in the tree table.

- The context sensitive menu appears.
- Select **Erase task** menu item.

To top

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9.22.7. Assigning Task Invocation

There can exist some leaf tasks, which are of the same performance as one of the root tasks in a task hierarchy. To avoid repetition of information, those top level tasks may be referenced by leaf tasks.
You can reference a task from the separate **Tasks** window, if it is opened in the **TDE** window.
Note: You can reference only the top (root) level task. Also you can assign a referenced task only for the task, which is a leaf element. After a task becomes an invocation task, it can not be assigned as a child element to other tasks.
Follow the routine to assign the selected top level task to a leaf task:

- Click on the top (root) level task in a task hierarchy. The selected task is highlighted.
- Drag the selected task onto the **Task Invocation** column of an applicable leaf task. A transparent icon and name of the top level task appears while dragging (see Figure 1).

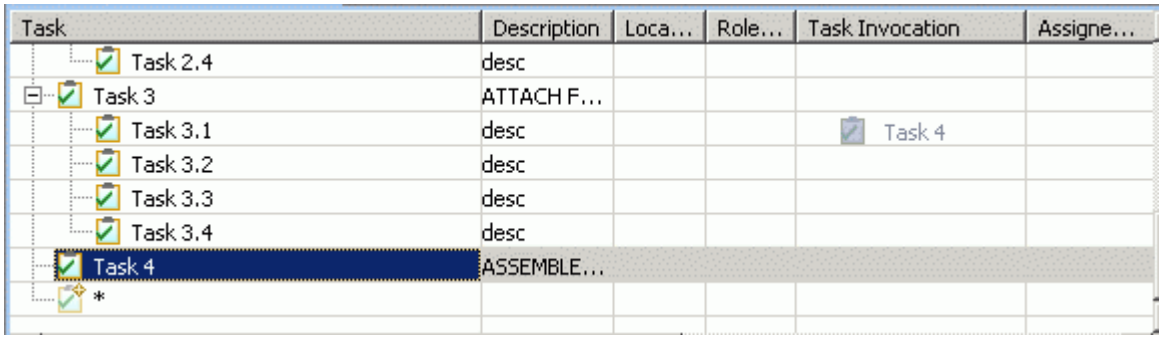

Figure 1. Top level task 'Task 4' dragged to leaf task 'Task 3.1'

- Release the mouse button when you have the selected task in the desired location. The task you've selected is moved to the **Task Invocation** column (see Figure 2).

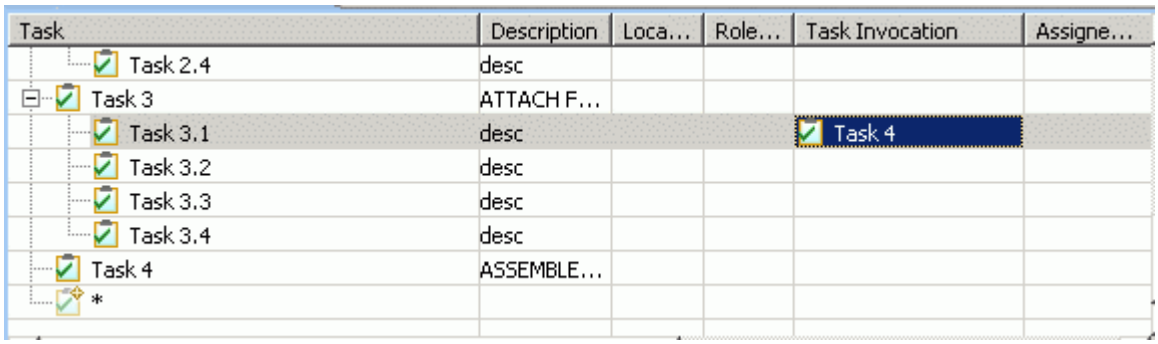

Figure 2. Top level task 'Task 4' assigned to leaf task 'Task 3.1'

You can remove the assigned task from the **Task Invocation** column:

- Right click on the referenced task name in the **Task Invocation** column. The context sensitive menu appears.

- Select **Delete** menu item. The task disappears from the cell.

To top

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9.22.8. Assigning Zonal Breakdown

You can set a zonal breakdown for the selected task in the **Location** column:

- Ensure that Tasks and Zonal Breakdowns windows are opened (see Figure 1).

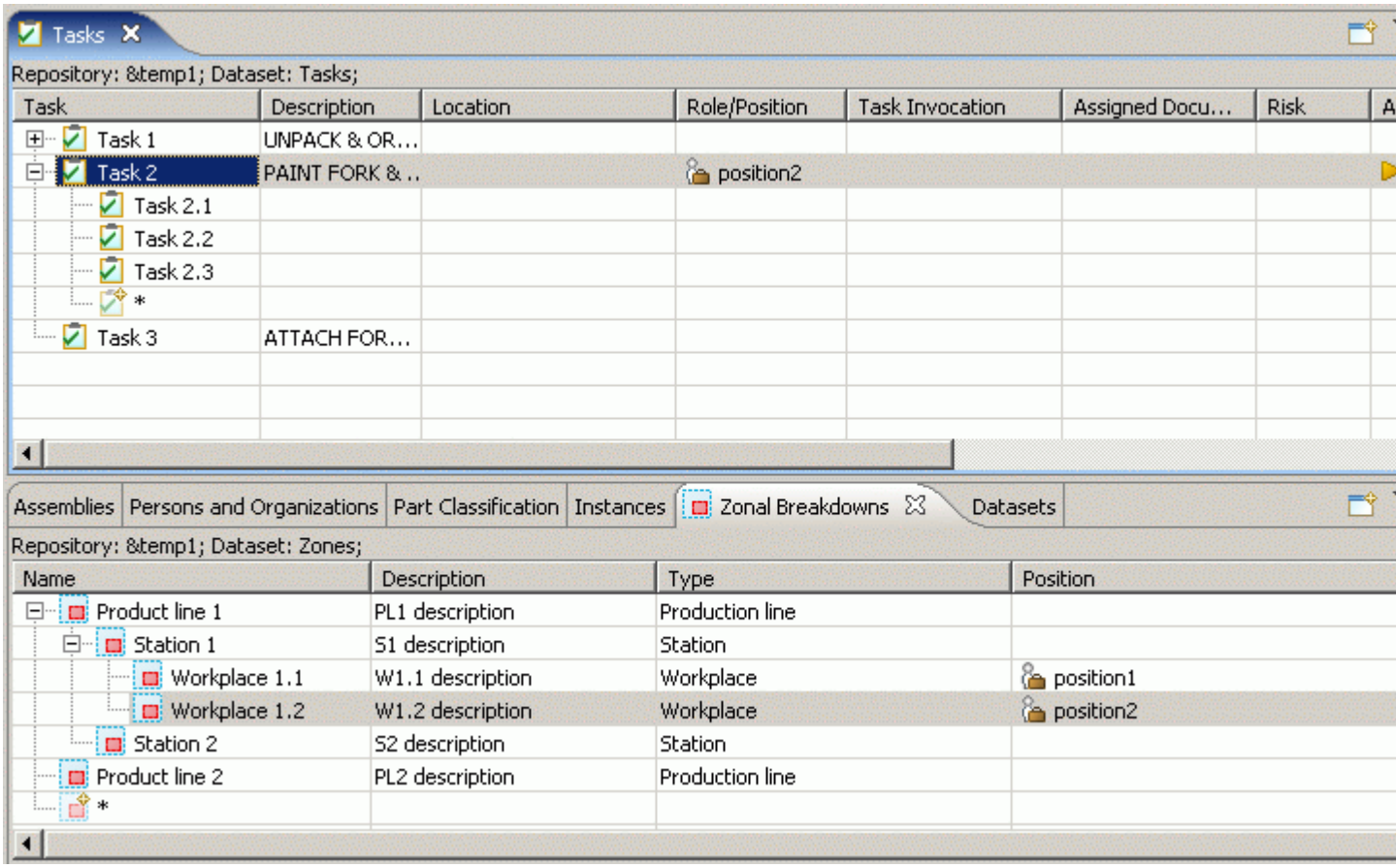

Figure 1. Tasks and Zonal Breakdowns windows in the TDE

- Click on an applicable zonal breakdown in the **Zonal Breakdowns** window. The selected zonal breakdown is highlighted.
- Drag the selected zonal breakdown onto the **Location** column of an applicable task in the **Tasks** window. A transparent icon and name of the zonal breakdown appears while dragging (see Figure 2).

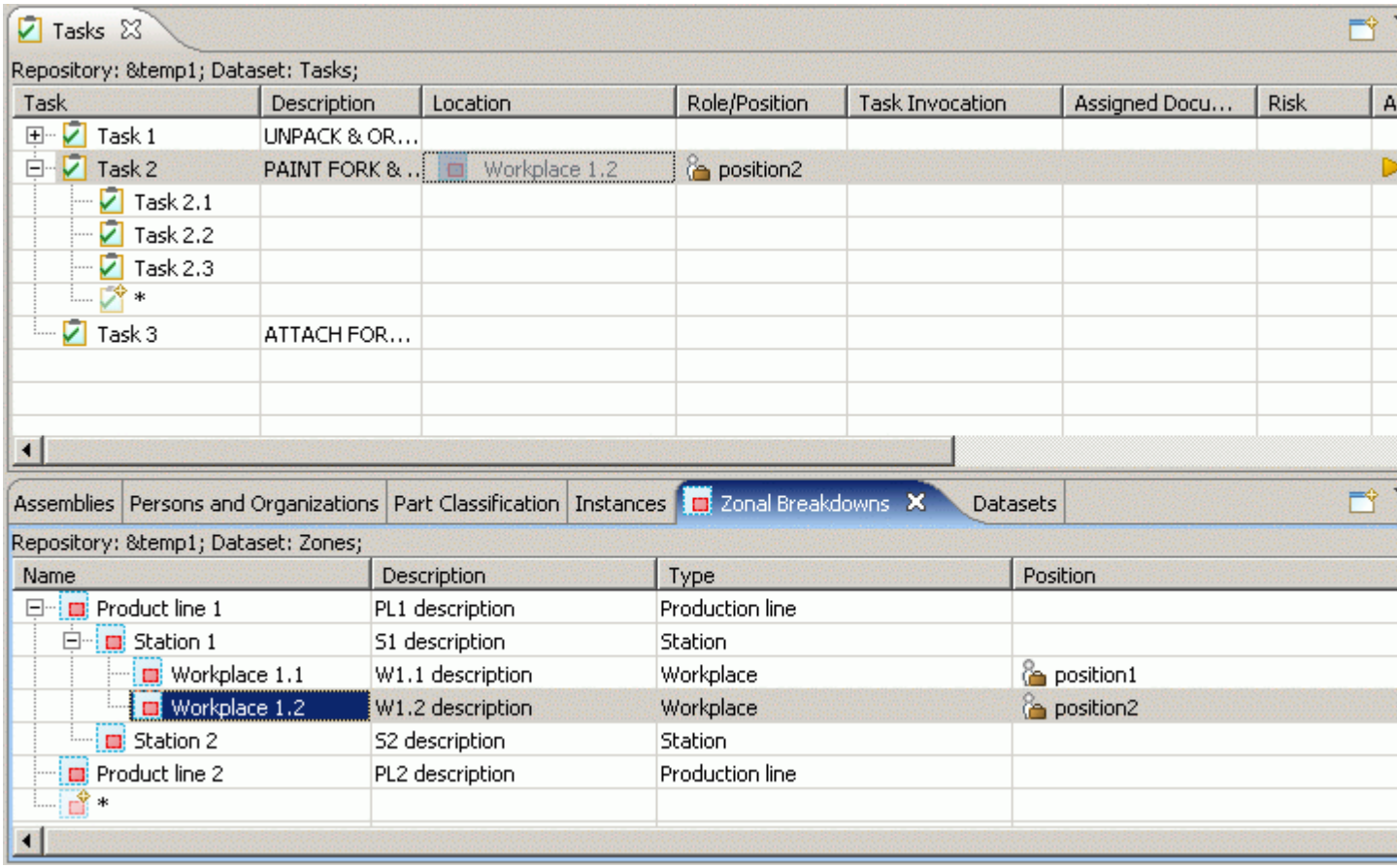

Figure 2. Zonal breakdown 'Workplace_1.2' dragged onto Task 'Task 2'

- Release the mouse button when you have the selected zonal breakdown in the desired location. The zonal breakdown you've selected is moved to the **Location** column of the desired task in the **Tasks** window tree table (see Figure 3).

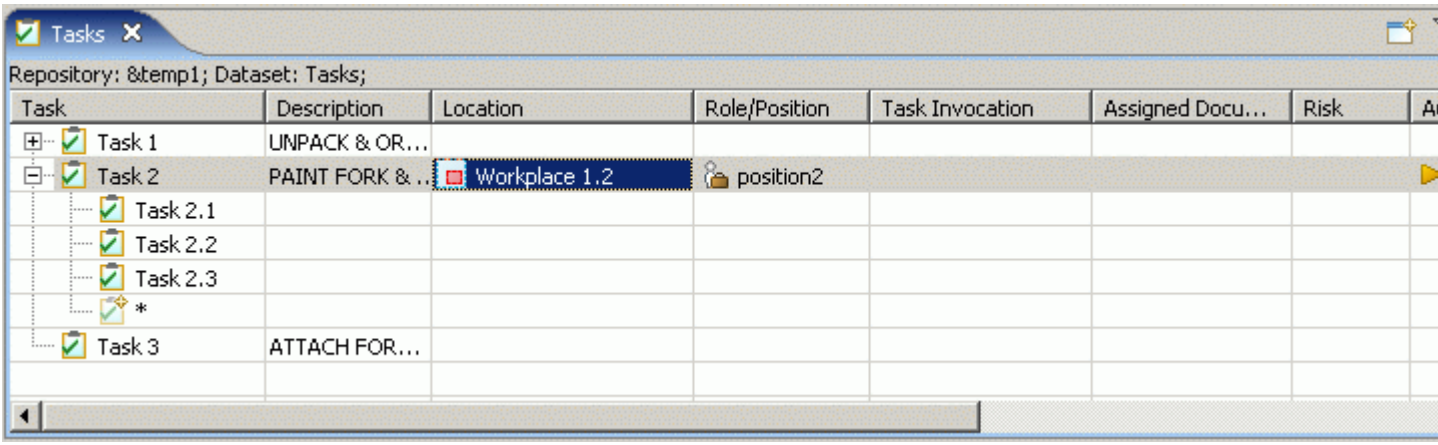

Figure 3. Zonal breakdown 'Workplace_1.2' assigned to Task 'Task 2'

You can remove an assigned zonal breakdown from the **Location** column in the **Tasks** window:

- Right click on a referenced zonal breakdown name in the **Location** column. The context sensitive menu appears.
- Select **Delete** menu item. The zonal breakdown disappears from the cell.

To top

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9.22.9. Assigning Part

You can set a part for the selected task in the **Product/Part** column:

- Ensure that Tasks and Part Details windows are opened (see Figure 1).

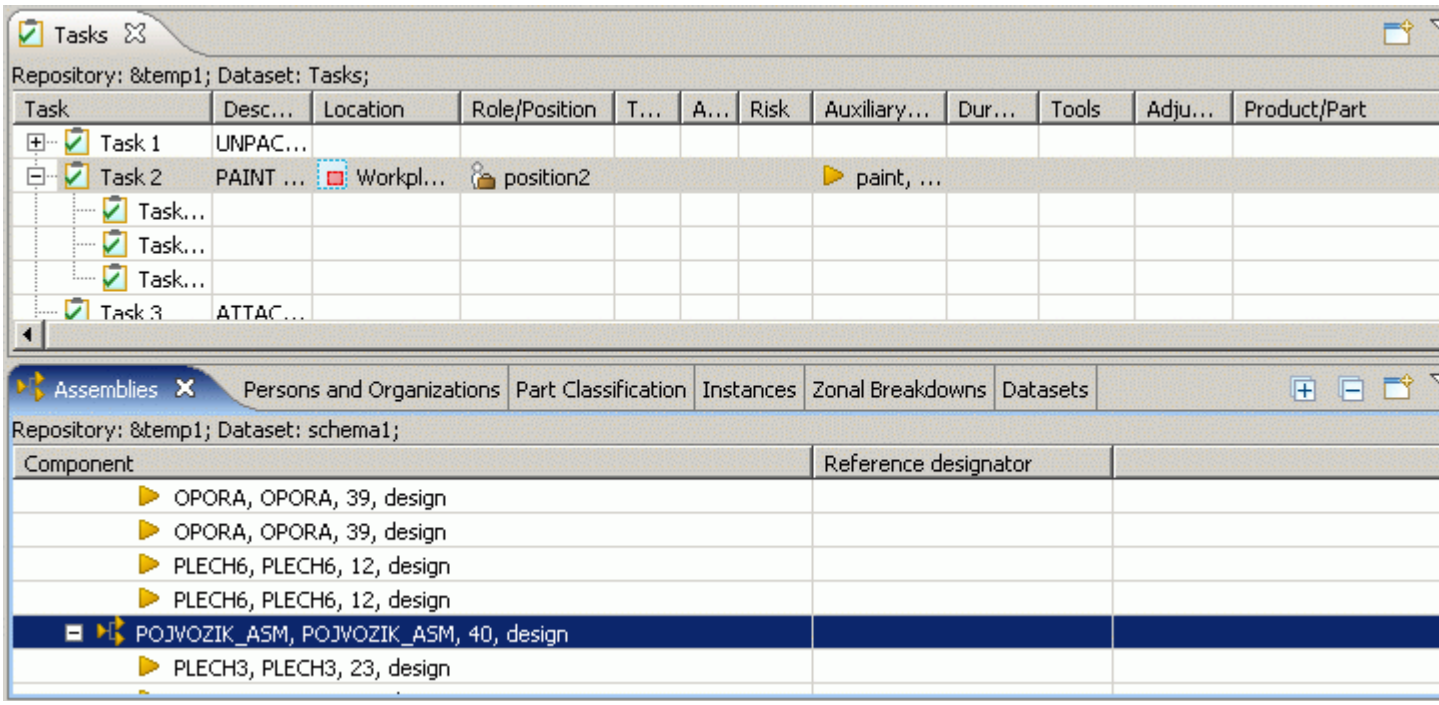

Figure 1. Tasks and Part Details window in TDE

- Click on an applicable object in the **Part Details** window. The selected object is highlighted.
- Drag the selected object onto the **Product/Part** column of an applicable task in the **Tasks** window. A transparent icon and name of the selected object appears while dragging (see Figure 2).

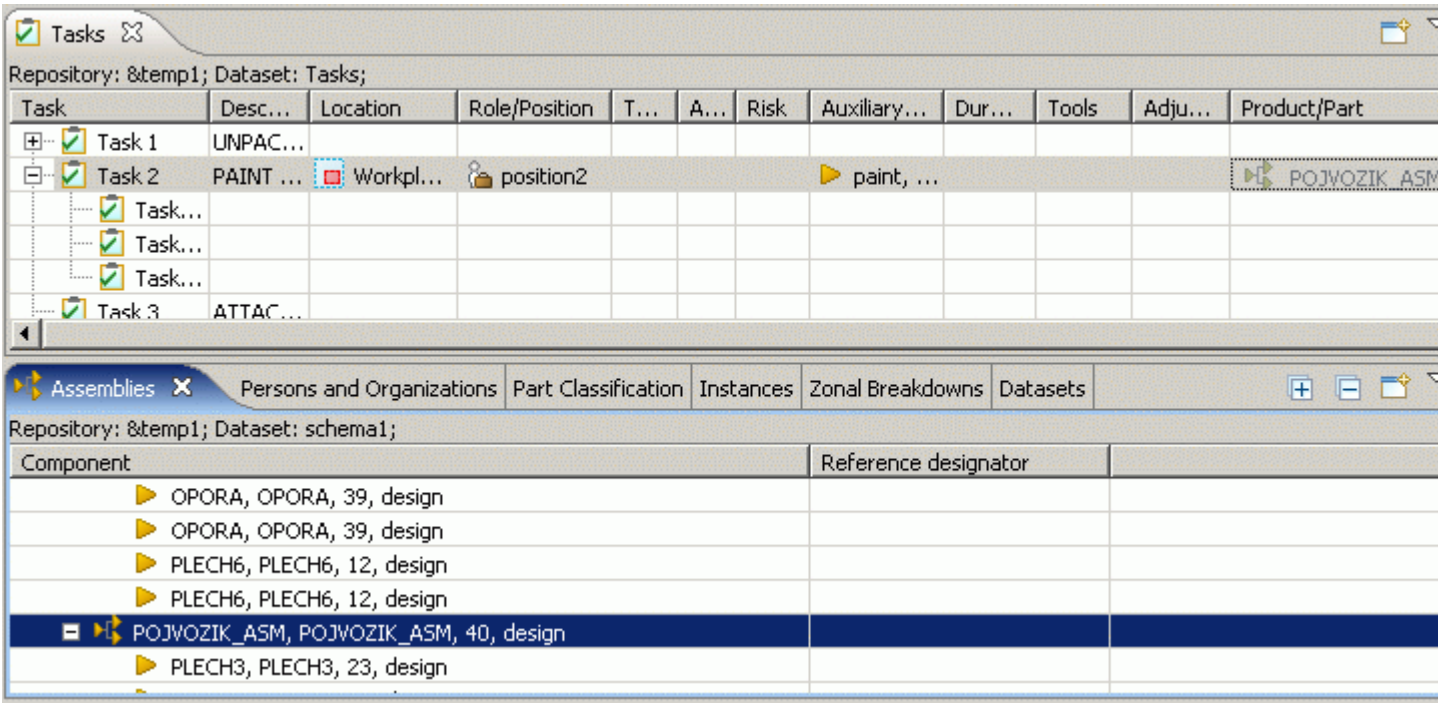

Figure 2. Selected part dragged onto task

- Release the mouse button when you have the selected object in the desired location. The object

you've selected is added as a part to the **Product/Part** column of the desired task in the **Tasks** window tree table (see Figure 3).

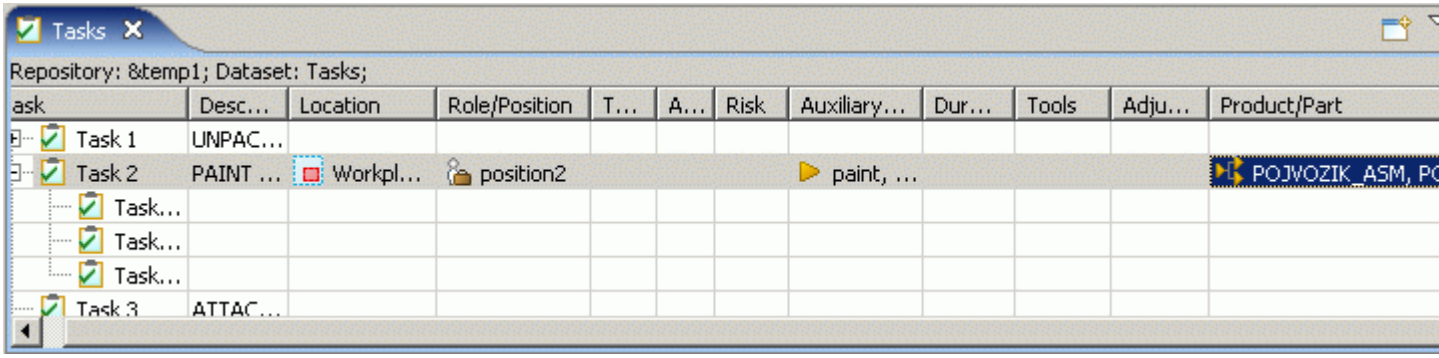

Figure 3. Selected part assigned to task

You can remove an assigned part from the **Product/Part** column in the **Tasks** window:

- Right click on a referenced part name in the **Product/Part** column. The context sensitive menu appears.
- Select **Delete** menu item.

To top

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9.22.10. Assigning Role/Position

You can set a position for the selected task in the **Role/Position** column:

- Ensure that Tasks and Persons and Organizations windows are opened (see Figure 1).

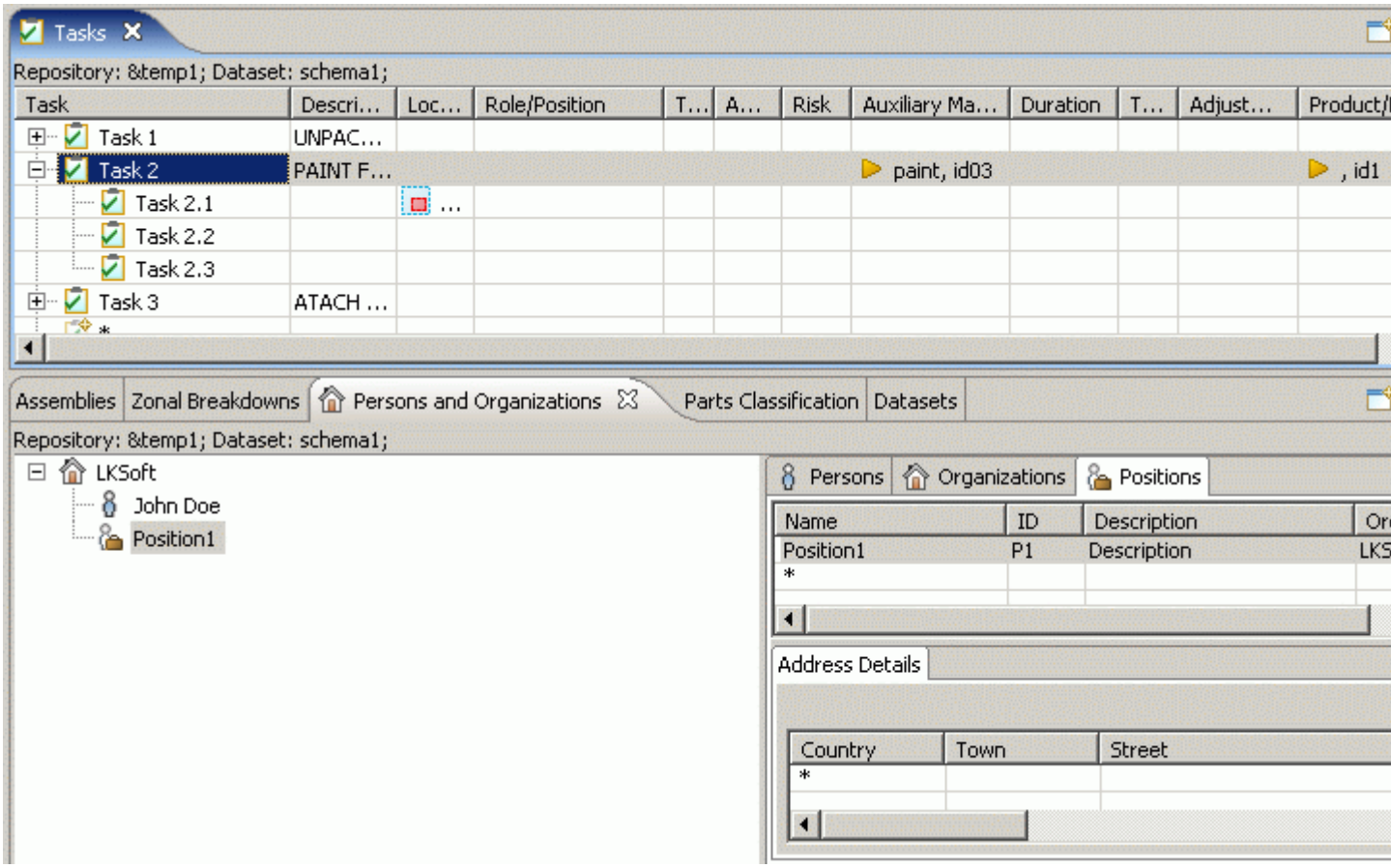

Figure 1. Tasks and Persons and Organizations windows in TDE

- Click on an applicable position in the organizations tree (on the left side of the **Persons and Organizations** window) or in a table of the **Positions** tabbed panel (on the right of the **Persons and Organizations** window). The selected object is highlighted in the tree and in the table list (see Figure 1).
- Drag the selected object onto the **Role/Position** column of an applicable task in the **Tasks** window. A transparent icon and name of the selected object appears while dragging (see Figure 2).

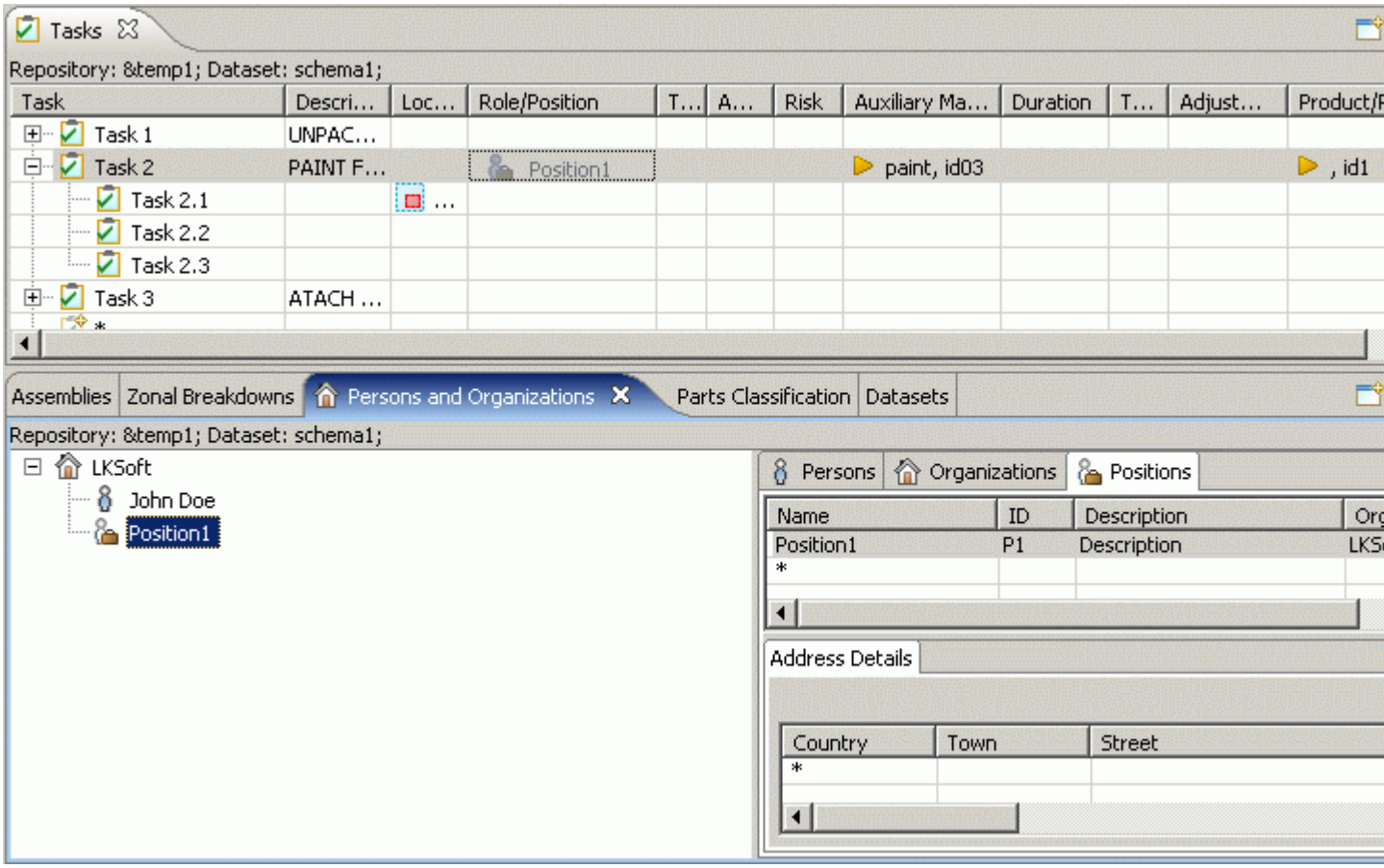

Figure 2. Selected position dragged onto task

- Release the mouse button when you have the selected object in the desired location. The object you've selected is added to the **Role/Position** column of the desired task in the **Tasks** window tree table (see Figure 3).

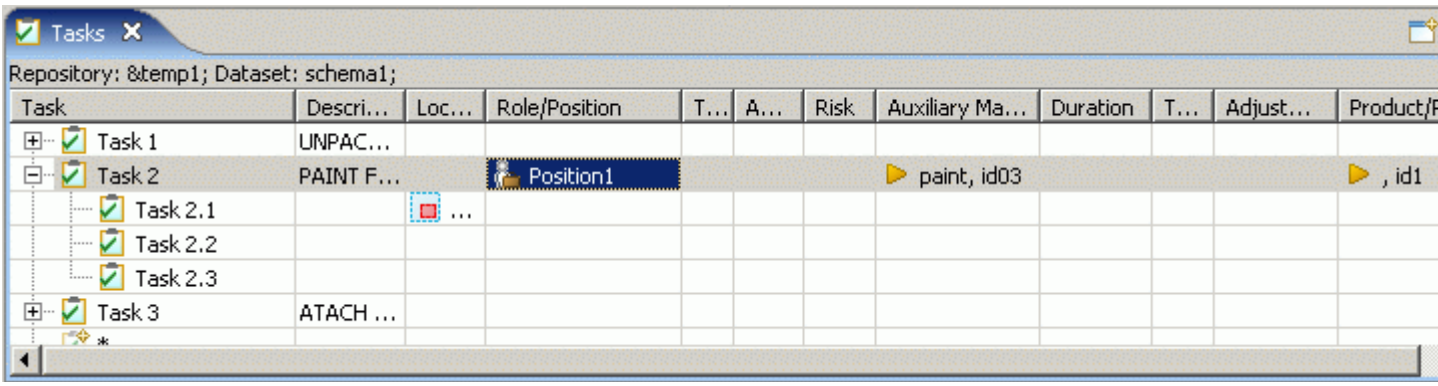

Figure 3. Selected position assigned to task

You can remove an assigned position from the **Role/Position** column in the **Tasks** window:

- Right click on a referenced position name in the **Role/Position** column. The context sensitive menu appears.
- Select **Delete** menu item.

To top

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9.22.11. Assigning Auxiliary Materials

You can set an auxiliary material for the selected task in the **Auxiliary Materials** column:

- Ensure that Tasks and Parts Classification windows are opened (see Figure 1).

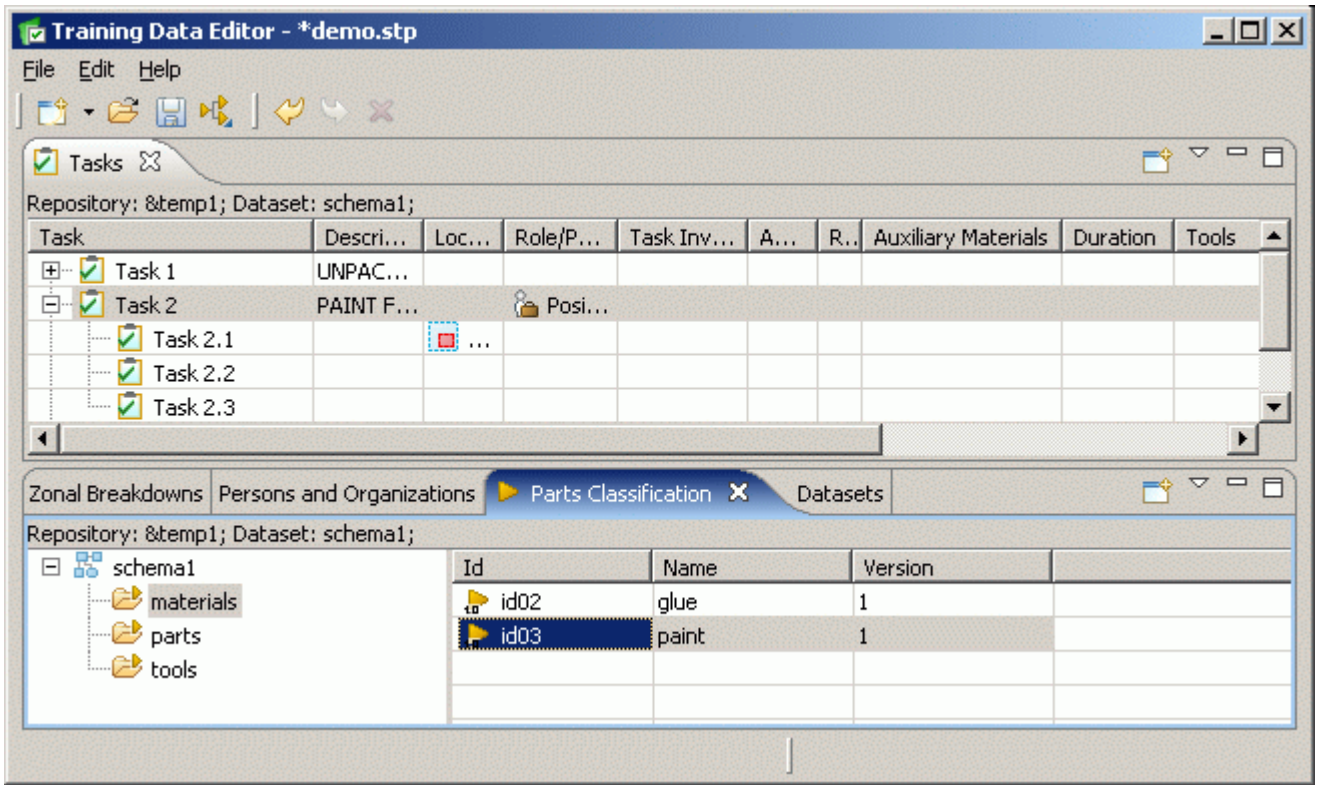

Figure 1. Task and Parts Classification windows in TDE

- Click on an applicable part in the table (on the right side of the **Parts Classification** window), which should be added as an auxiliary material for the selected task. The selected object is highlighted in the table list (see Figure 1).
- Drag the selected object onto the **Auxiliary Materials** column of an applicable task in the **Tasks** window. A transparent icon and name of the selected object appear while dragging (see Figure 2).

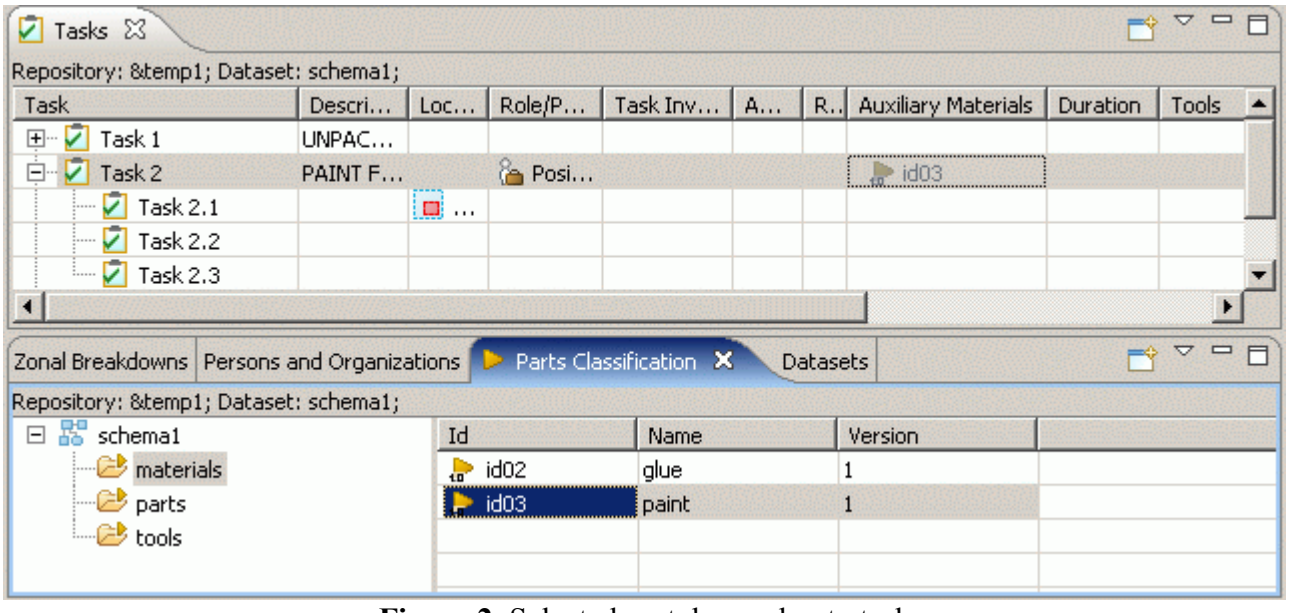

Figure 2. Selected part dragged onto task

- Release the mouse button when you have the selected object in the desired location. The object

you've selected is added to the **Auxiliary Materials** column in the **Tasks** window tree table (see Figure 3).

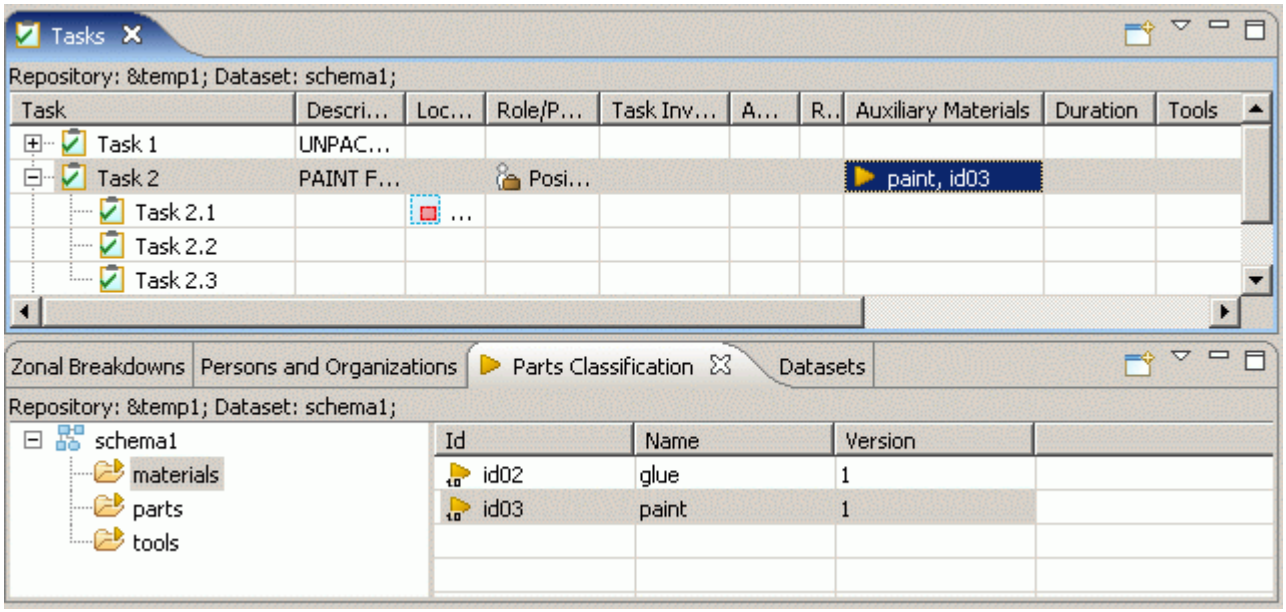

Figure 3. Selected auxiliary material assigned to task

You can remove the assigned auxiliary material from the **Auxiliary Materials** column in the **Tasks** window:

- Right click on an auxiliary material name in the **Auxiliary Materials** column. The context sensitive menu appears.
- Select **Delete** menu item.

To top

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9.22.12. Assigning Tools

You can set a tool for the selected task in the **Tools** column:

- Ensure that Tasks and Parts Classification windows are opened (see Figure 1).

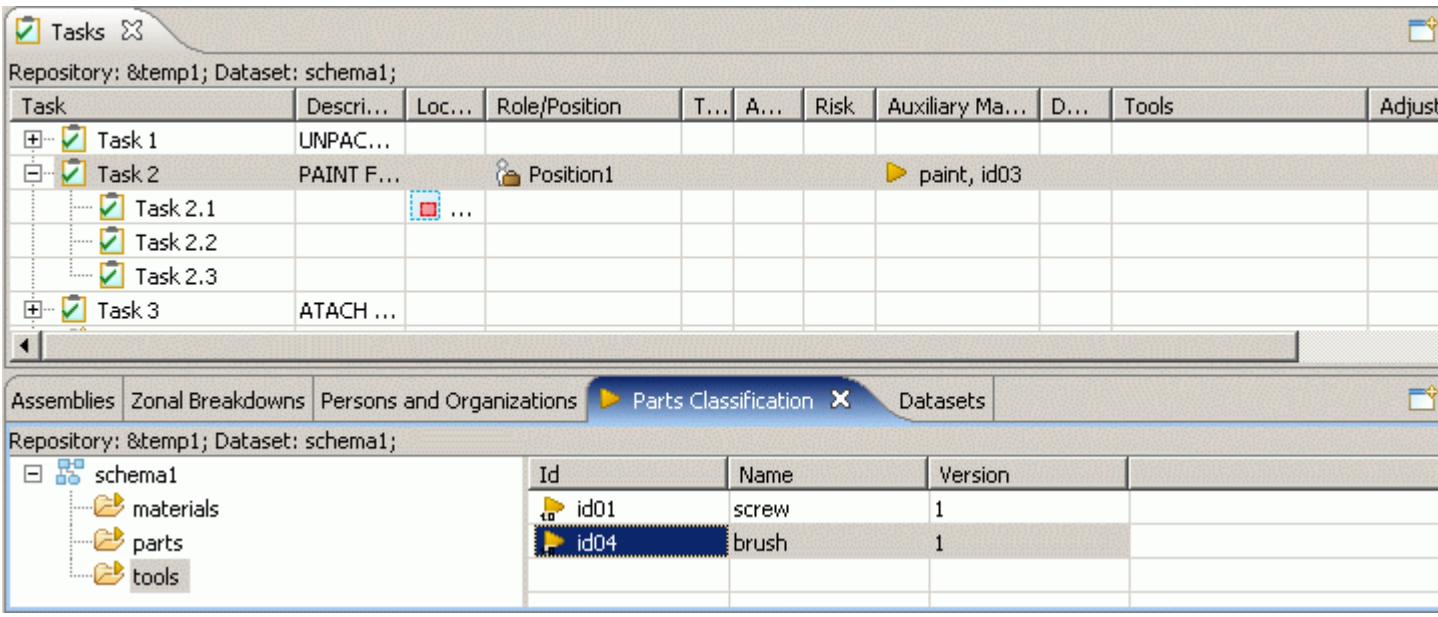

Figure 1. Task and Parts Classification window on the workbench

- Click on an applicable part in the table (on the right of the **Parts Classification** window), which should be added as a tool for the selected task. The selected object is highlighted in the tree or in the table list (see Figure 1).
- Drag the selected object onto the **Tools** column of an applicable task in the **Tasks** window. A transparent icon and name of the selected object appears while dragging (see Figure 2).

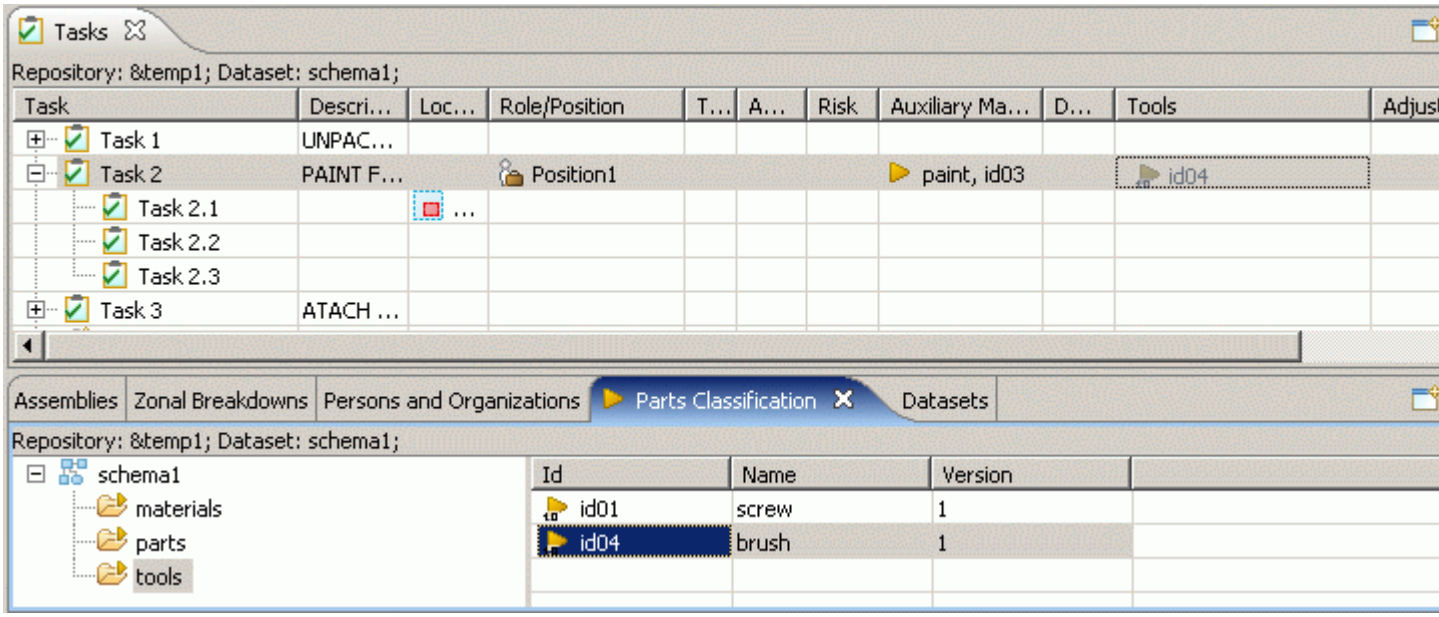

Figure 2. Selected part dragged onto task

- Release the mouse button when you have the selected object in the desired location. The object you've selected is added to the **Auxiliary Materials** column of the desired task in the **Tasks** window tree table (see Figure 3).

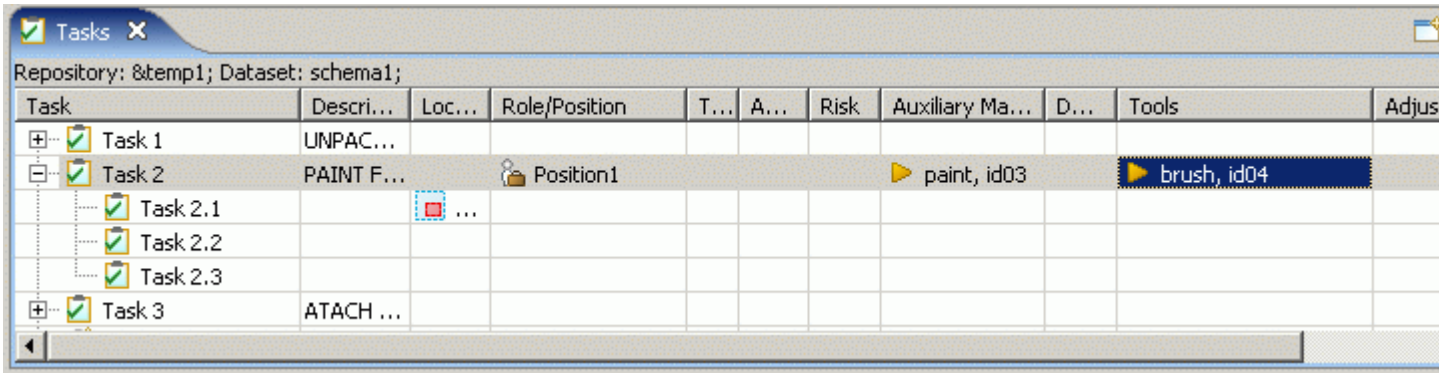

Figure 3. Selected auxiliary material assigned to task

You can remove an assigned tools from the **Tools** column in the **Tasks** window:

- Right click on a referenced tool name in the **Tools** column. The context sensitive menu appears.
- Select **Delete** menu item.

To top

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9.23.1. Zonal Breakdowns

A *zonal breakdown* is a partitioning of a product into a set of related zonal elements to support engineering, analysis and other activities that may be performed in relation to the product.
The **Zonal Breakdown Editor** presents a tree table with a zonal breakdown hierarchy, which is available for the selected dataset (see Figure 1):

- **Name** - hierarchy of zonal breakdowns;
- **Id** - zonal breakdown identifier;
- **Type** - zonal breakdown type definition;
- **Role/Position** - associated position.

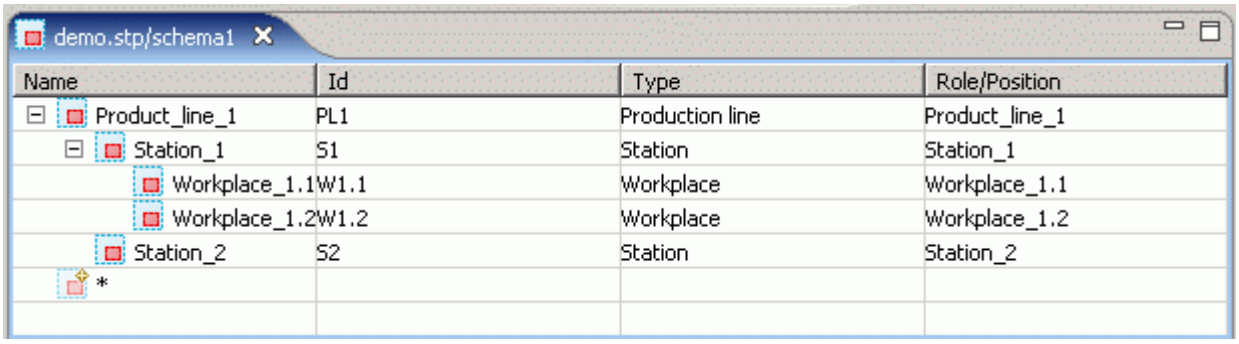

Figure 1. Zonal Breakdown window

To top

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9.23.2. Zonal Breakdown Hierarchy

The **Name** column in the **Zonal Breakdown** window presents a hierarchy of zonal breakdowns (see Figure 1):

- **Root nodes** - zonal breakdowns, that are independent from each other. The order of those zonal breakdowns is not important, but you can change order by dragging an applicable zonal breakdown row and dropping it at a desired place (if editing is available).
- **Second and further level nodes** (branches and leaves) - sub-zonal breakdowns.

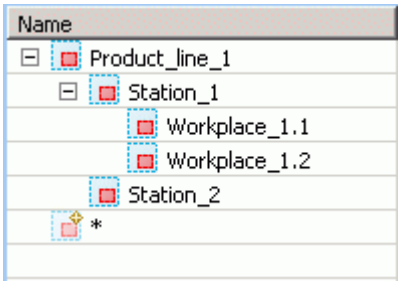

Figure 1. Zonal breakdown hierarchy

To top

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9.23.3. Creating Zonal Breakdown

- Creating Top Level Zonal Breakdown
- Creating Zonal Breakdown Element

There is a possibility to add a new zonal breakdowns to a Zonal Breakdown hierarchy.

Creating Top Level Zonal Breakdown

You can add a new top level zonal breakdown by using the autorow:

- Ensure that the autorow is in the right position - the last row in the tree table, if it is not - drag and drop it at the end of the tree table.
- Enter data into the autorow.
- Press **Enter** key. A new zonal breakdown is created as the root node in the tree.

To top

Creating Zonal Breakdown Element

You can add an element for the already created zonal breakdown:

- Drag and drop the autorow onto the preferable zonal breakdown.

Or

- Right click on a parent zonal breakdown and select **Add element** menu item in the appeared context sensitive menu.
- Enter data into the autorow.
- Press **Enter** key. A new zonal breakdown is added.

Note. A zonal breakdown will be added at the end of the tree branch (the same level node).

To top

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9.23.4. Editing Zonal Breakdown Data

Activate the **Zonal Breakdowns** view to edit data. Select the zonal breakdown you want to edit in a zonal breakdown hierarchy. You can edit the following:

- Enter the text, which defines the zonal breakdown name into the **Name** column.
- Enter the identifier into the **Id** column.

- Select a type of the zonal breakdown in the **Type** column:
  - Double click on the cell to set the editing mode ON. The combo box arrow appears.
  - Click on the arrow to view the drop down list.
  - Select an applicable value.

  Or

  - Enter an applicable value into the text field.
- Assign a position in the **Position** column (see Assigning Position).

To top

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9.23.5. Assigning Position

You can set a position for the selected zonal breakdown in the **Position** column:

- Ensure that **Zonal Breakdowns** and **Persons and Organizations** views are opened (see Figure 1).

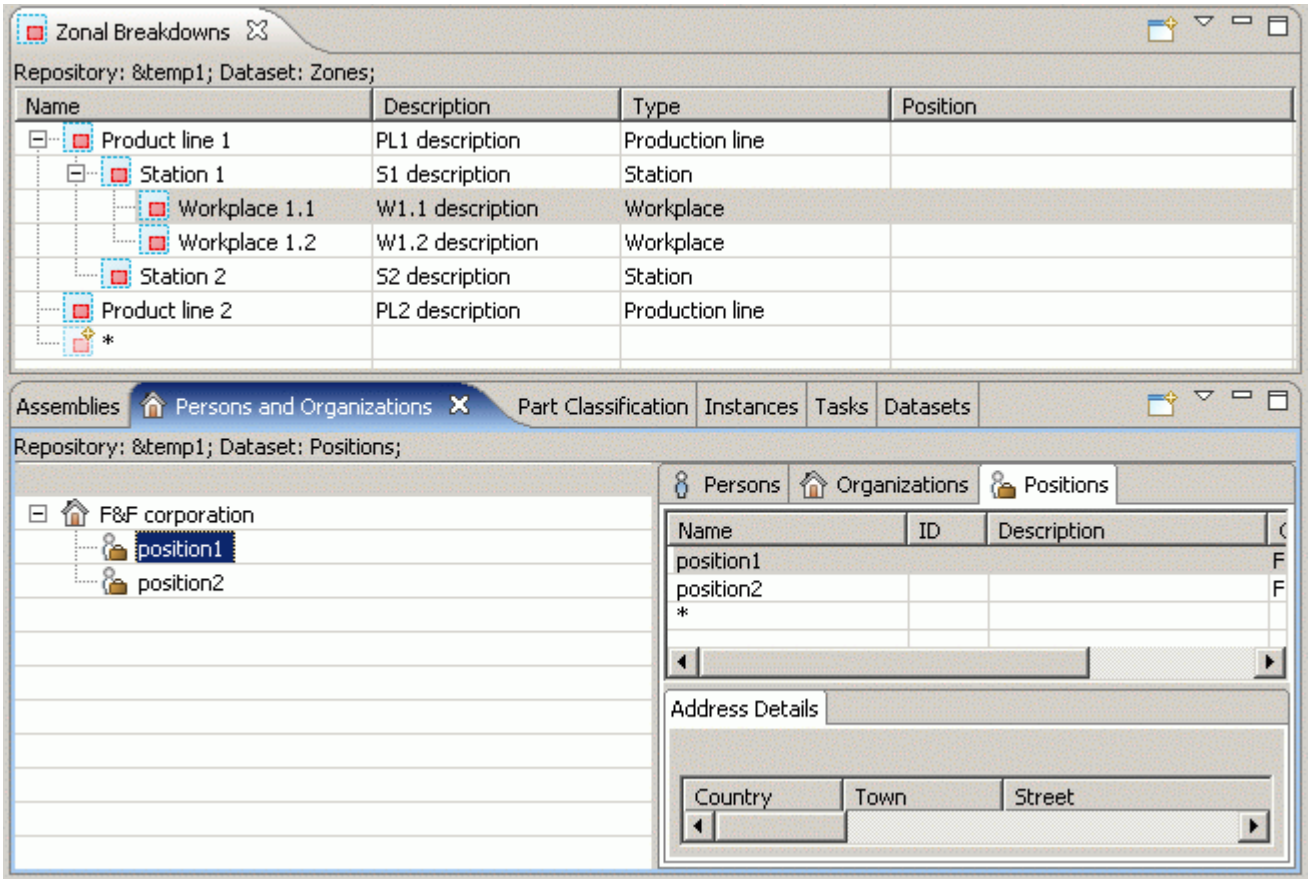

Figure 1. Zonal Breakdowns and Persons and Organizations views in TDE

- Click on an applicable position in the organizations tree (on the left side of the **Persons and Organizations** view) or in a table of the **Positions** tabbed panel (on the right of the **Persons and Organizations** view). The selected object is highlighted in the tree or in the table list (see Figure 1).

- Drag the selected object onto the **Position** column of an applicable zonal breakdown in the **Zonal Breakdowns** view. A transparent icon and name of the selected object appears while dragging.
- Release the mouse button when you have the selected object in the desired location. The object you've selected is added to the **Position** column of the desired zonal breakdown in the **Zonal Breakdowns** view tree table (see Figure 2).

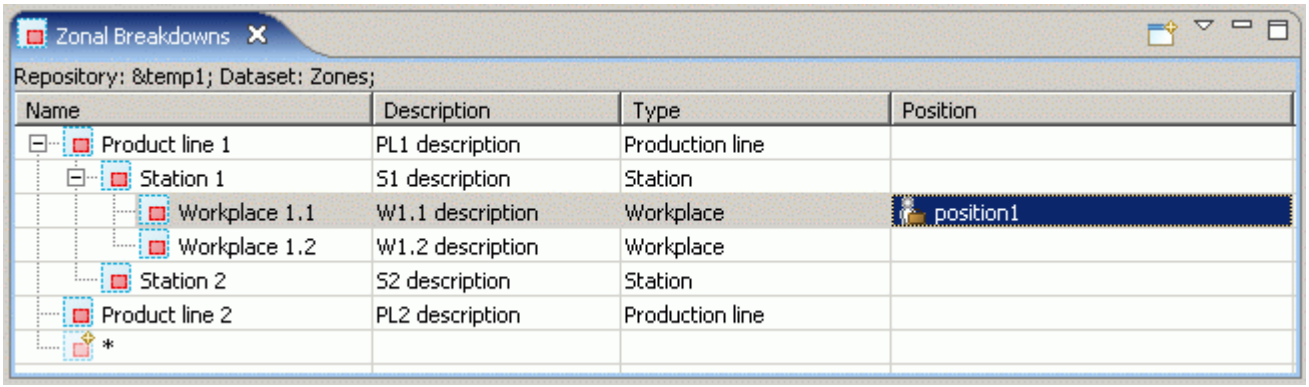

Figure 2. Selected position assigned to zonal breakdown

You can remove an assigned position from the **Position** column in the **Zonal Breakdowns** view:

- Right click on a referenced position name in the **Position** column. The context sensitive menu appears.
- Select **Delete** menu item.

To top

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9.23.6. Moving

You can move an element by dragging and dropping the selected tree table row:

- **Moving zonal breakdown** - to change a zonal breakdown location in the hierarchy. The same result can be achieved by cutting and pasting an element.
- **Moving autorow** - to create a new zonal breakdown in a particular place of a zonal breakdown hierarchy.

To top

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9.23.7. Removing

There is a possibility to remove a redundant or an incorrect zonal breakdown from a zonal breakdown hierarchy (tree table). **Note**, you can remove only those zonal breakdowns, which have no zonal breakdowns beneath (child nodes). Follow the routine to remove a zonal breakdown form the

tree table:

- Right click on a zonal breakdown node. The context sensitive menu appears.
- Select **Erase zone** menu item.

To top

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10.1. Customizing Views

IDA-STEP provides capabilities to customize the views, which display structured data. The **Parts**, **Part Details** and other views in IDA-STEP display data by using tables or tree-tables. The layout of tables is predefined and the data types they display are set. The most right column in the tree table is marked with an asterisk (*). In case the column cell is empty, no additional data is available for the row element. In other case, a user can review a list of available data, which are not currently included into the tree table, by pressing on the asterisk. The **Other data** window appears on the screen with the list of available data together with assigned values. The presenting data can be included into the tree table simply by double clicking on an applicable data type and filling required fields in the appeared **Insert column** wizard dialog window.

The user has full control on the columns in tables - add, remove or configure. When a column is added, the user can define its content by choosing from the predefined data types or use a wizard to specify custom data types using templates.

To top

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10.2. Inserting Column

Follow the routine to insert a new column into the table of the currently active view:

- Right click on the table area - on the element/row.
- Select the **Insert column** menu item in the appeared context sensitive menu.
- The Insert column wizard window appears on the screen. Use the wizard to create an applicable column.
- The created column appears in the table.

Or

- Press with a mouse button on the most right column cell with an asterisk. The **Other data** window appears with available additional data (see Figure 1).
  Note. If you are in a view, which can be linked with other views, the **Other data** window can show up only for a short time. To vanish this inconvenience switch *Off* the **Link with views** button.

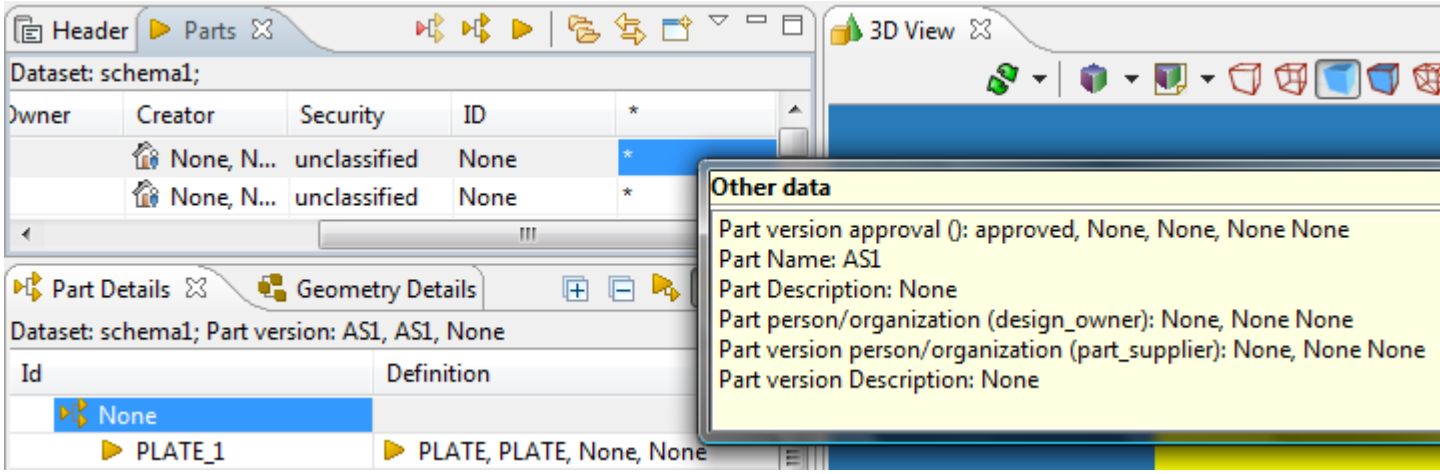

Figure 1. Other data window

- Select an applicable data type in the list and double click on it.
- The Insert column wizard dialog window appears on the screen.

By default the **Column from data** radio button is selected and the previously selected data type is highlighted in the wizard page. You can select another radio button to create new data or press **Finish** button to include already associated data to the tree table.

To top

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10.3. Removing Column

Follow the routine to remove a redundant column from the table:

- Open an applicable view in the workbench.
- Right click on the column you want to remove.
- Select the **Remove column** menu item in the appeared context sensitive menu.
- The selected column disappears from the table.

Note. If the removed column contained some data, the type identification and column value can be found in the most right column (labeled '*') of the tree table. To get a list of available columns, which are not currently located in the table, press on the * and examine the appeared **Other data** window.

To top

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10.4. Configuring Column

You can configure already created dynamic column:

- Right click on the column you want to configure. A context sensitive menu appears.
- Select the **Configure Column** menu item. The **Configure dynamic colum**n wizard dialog window appears (see Figure 1). The current column parameters are presented in the wizard dialog window.

- You can change a role and a template after the **Simple column** radio button is selected (see **Simple column** in Insert column wizard).

Or

- Compose new templates for the column after the **Compose column** radio button is selected (see **Compose column** in Insert column wizard).
- Specify new parameters for the column and click **Finish** button.

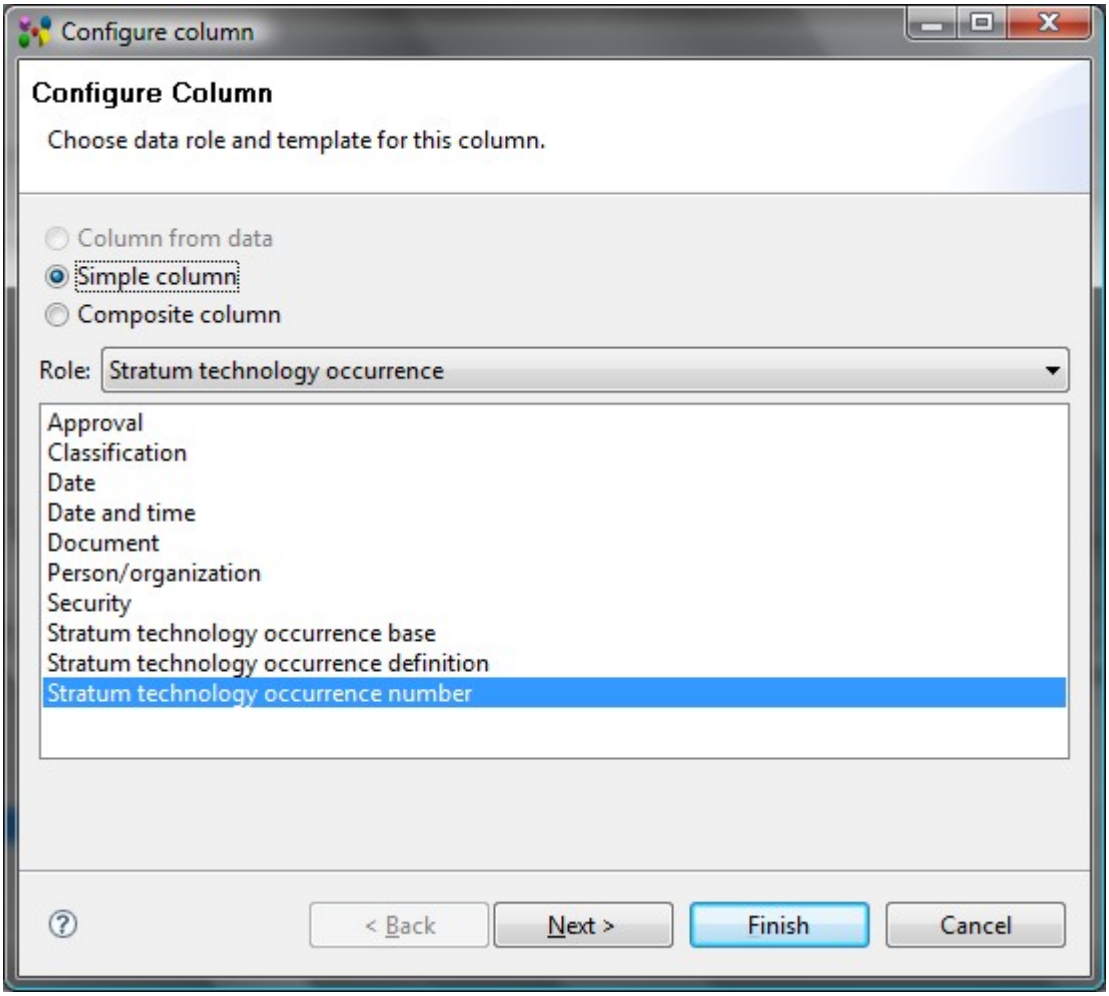

Figure 1. Configure column wizard dialog window

To top

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10.5. Insert Column Wizard

The **Insert Column** wizard helps the user to add an applicable column to the selected view table (Figure 1). This wizard can be invoked by selecting **Insert column** menu item in the context sensitive menu or by clicking on a particular element in **Other data** window list.
A new column may be created in three ways, by selecting a particular radio button:

- **Column from data** - a new column is created with already associated data.
- **Simple column** - a new column is created, by specifying the element object, column type and other criteria.
- **Composite column** - a new column is composed by the user.

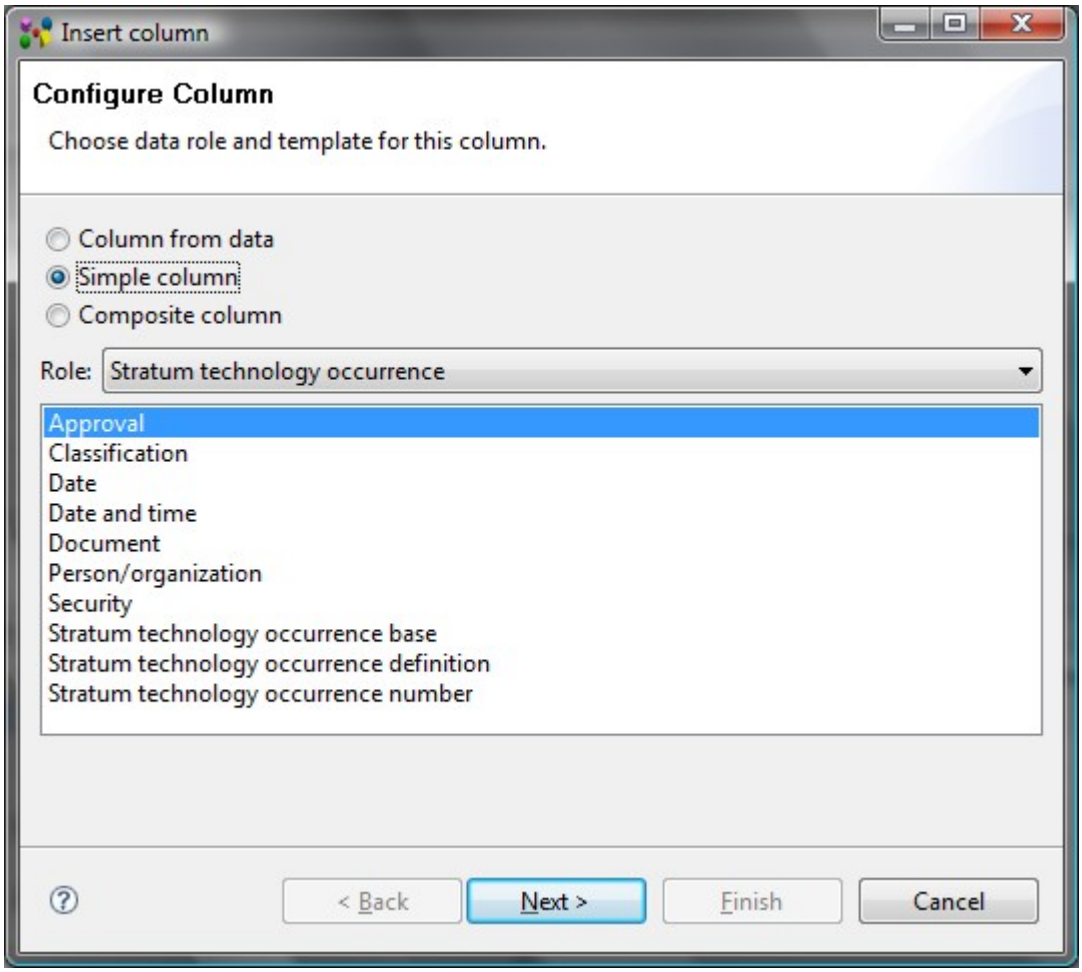

Figure 1. Insert column wizard page

Column from data

After the **Column from data** radio button is selected, a new wizard page appears on the screen (see Figure 2):

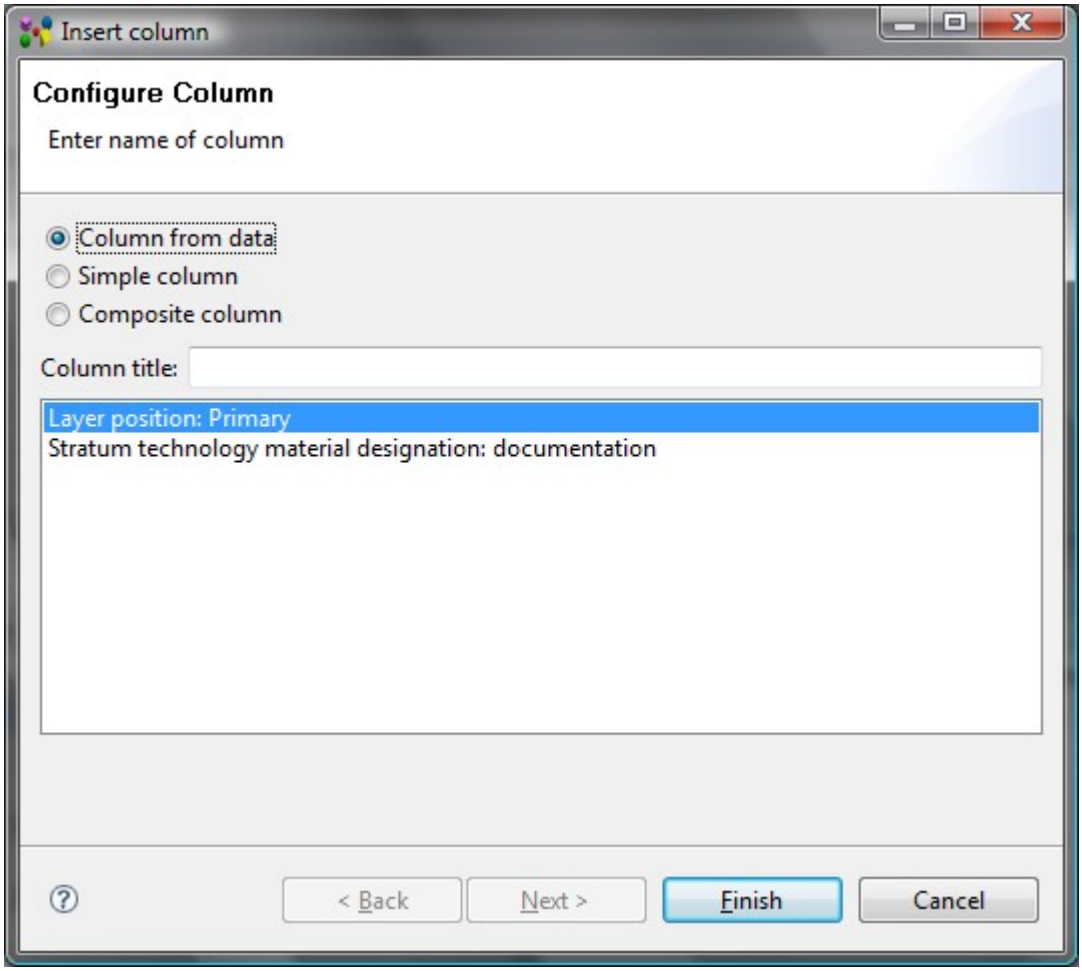

Figure 2. Insert column wizard, Select from data page

- **Column title** - to specify the title for the column to be created.
- Below the **Column title** field the list of data, which are assigned to the selected object, but not displayed in the table, is presented.
  You can see the same list in the table. The asterisk (*) in the column, which title is marked also with an asterisk, identifies that the selected element has additional data. Click on the cell, marked with asterisk to view the list of data (see Figure 3).

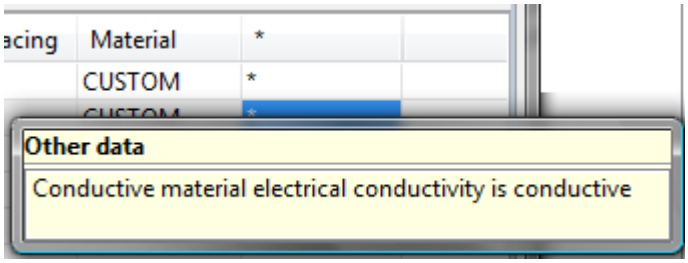

Figure 3. Other data list

- After the column title is entered and the data selected from the list, press the **Finish** button to create a new column. The selected data appears in the newly created column.

To top

Simple column

After the **Simple column** radio button is selected the particular wizard page appears (see Figure 4).

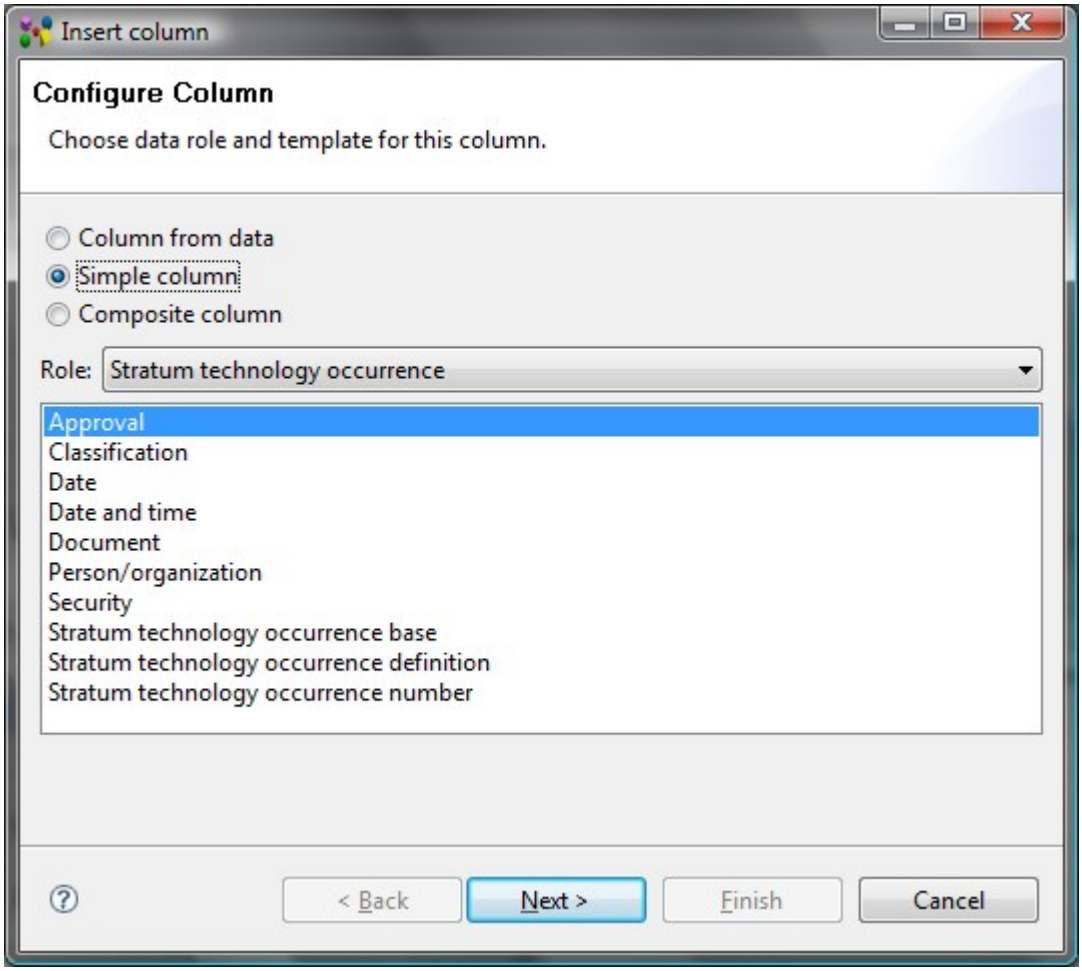

Figure 4. Wizard page to create a new dynamic column

- Select an object, the column to be related to from the **Choose role** drop down list. There are presented available objects of the selected view in the drop down list.
- Choose column type by selecting a particular radio button in the **Choose column** type layout. The list of types is specific for each view.
- Click **Next** button.
- The second page appears, which depends on what selection was made in the previous page.
- Enter needed data and click **Finish** button to create a new dynamic column.

To top

Composite column

After the **Composite column** radio button is selected, a new wizard page appears on the screen (see Figure 5).

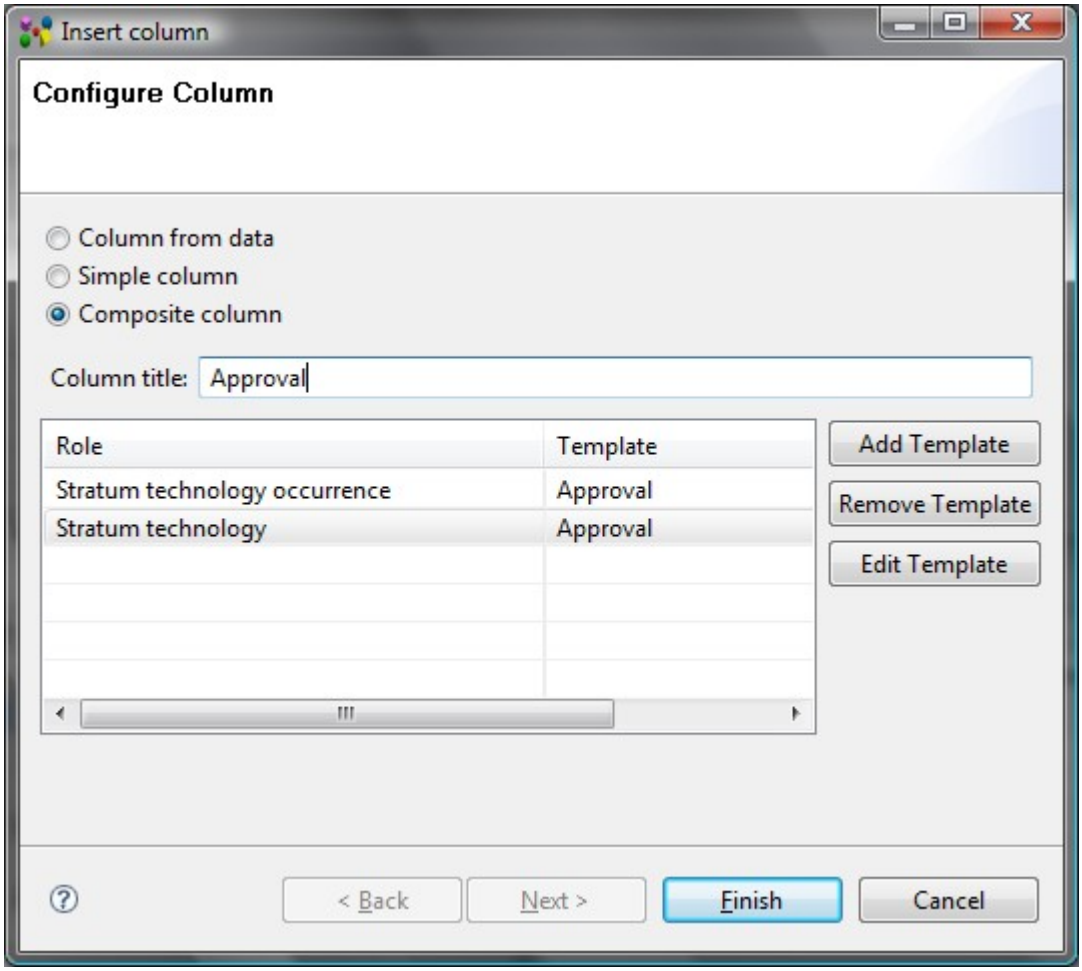

Figure 5. Wizard page to composite new column

- **Column title** - to specify the title for the column to be created.
- The table in the middle section of the page contains a list of column templates. There can be applied several templates for one column.
    - **Role** column specifies a role of the element.
    - **Template** column specifies a type of the column to be created.
- **Add Template** button - create a new template for the column. After this button is pressed a new wizard dialog page appears on the screen (see Figure 6) to choose a role and a template for the column to be created.

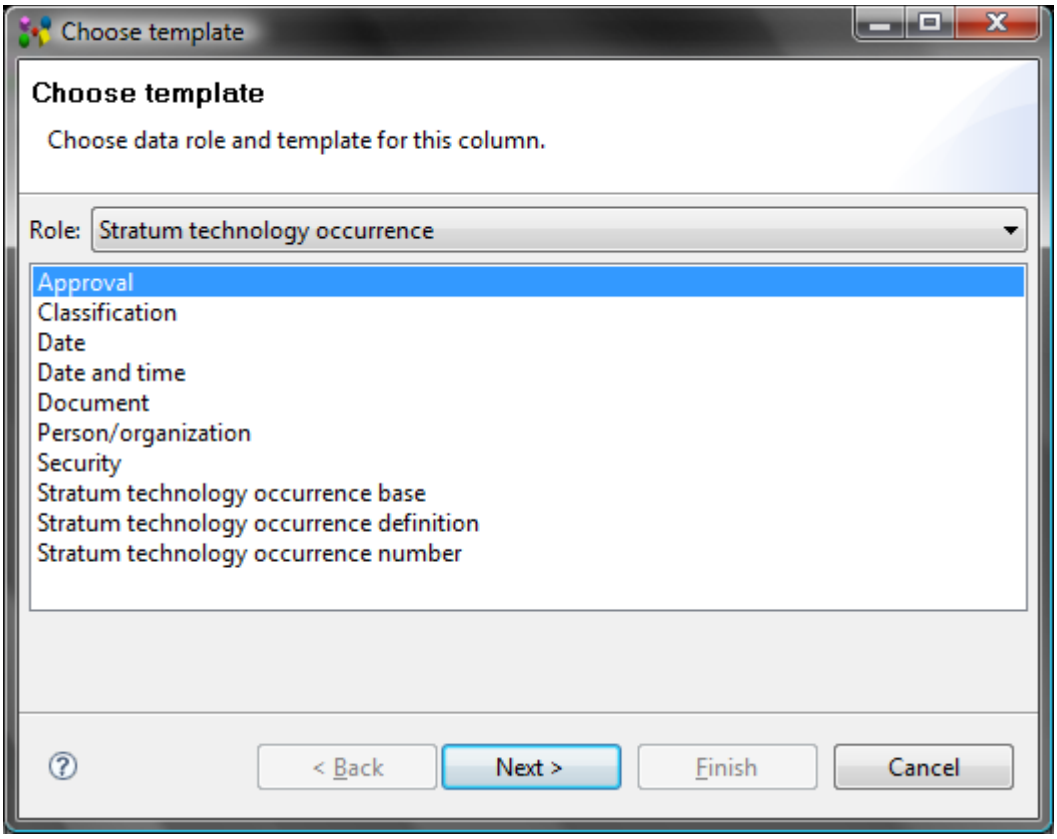

Figure 6. Wizard page to choose data role and template for the column

If there are additional parameters for the selected template, press **Next** button to go to the next wizard page (see Figure 7). After all needed parameters are entered/selected, press **Finish** button.

The created template appears in the table list (wizard page in Figure 5).

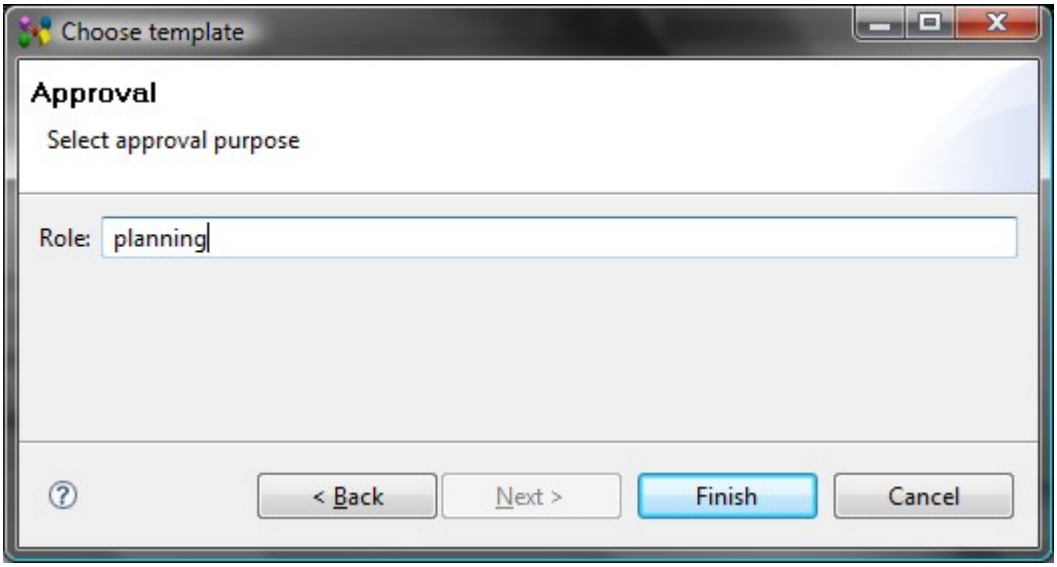

Figure 7. Wizard page to select additional parameters for the column to be created

- **Remove Template** button - remove the selected template from the table.
- **Edit Template** button - edit a particular template after it is selected in the table.

To top

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10.6.1. Activity

An *Activity* is the identification of the occurrence of an action that has taken place, is taking place, or is expected to take place in the future.
There are no additional parameters to enter in the wizard page (see Figure 1) except a column title. Enter a column title into the **Column title** field and click **Finish** button.

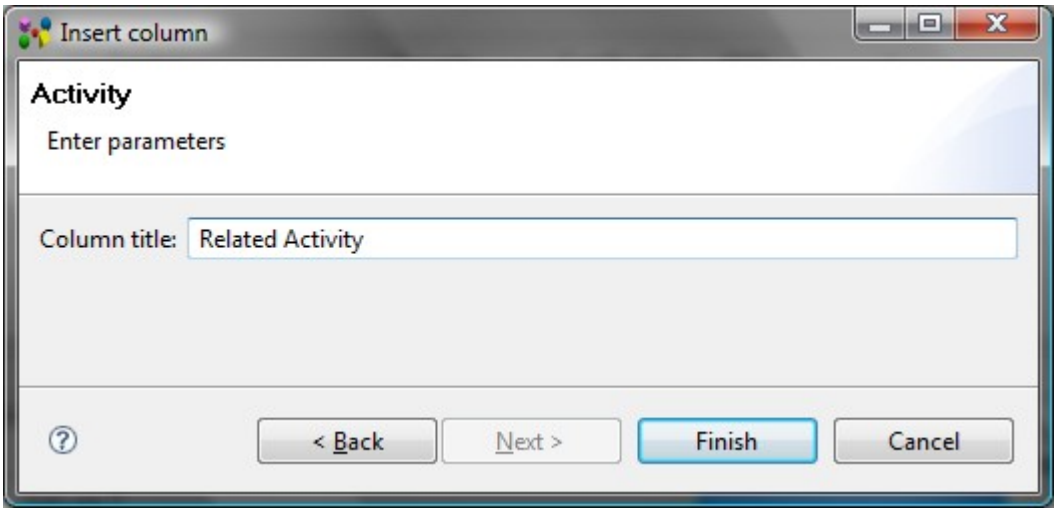

Figure 1. Activity column parameters

The column of type activity with a specified title appears in a table. To fill the column with value:

- Select an activity element in a particular view (for example **Maintenance Activities**).
- Copy and paste it into the activity column (see Figure 2). You can also drag and drop selected activity element.

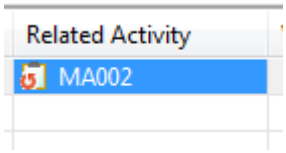

Figure 2. Activity column with value

If there are created a few columns of the same type, all columns for the same object contain the same data.

To top

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10.6.2. Approval

An *Approval* represents a statement made by technical personnel or management personnel whether certain requirements are met.
Enter parameters for the approval column (see Figure 1):

- Enter a title for the column into the **Column title** field.
- Enter a role of the approval into the **Role** filed.

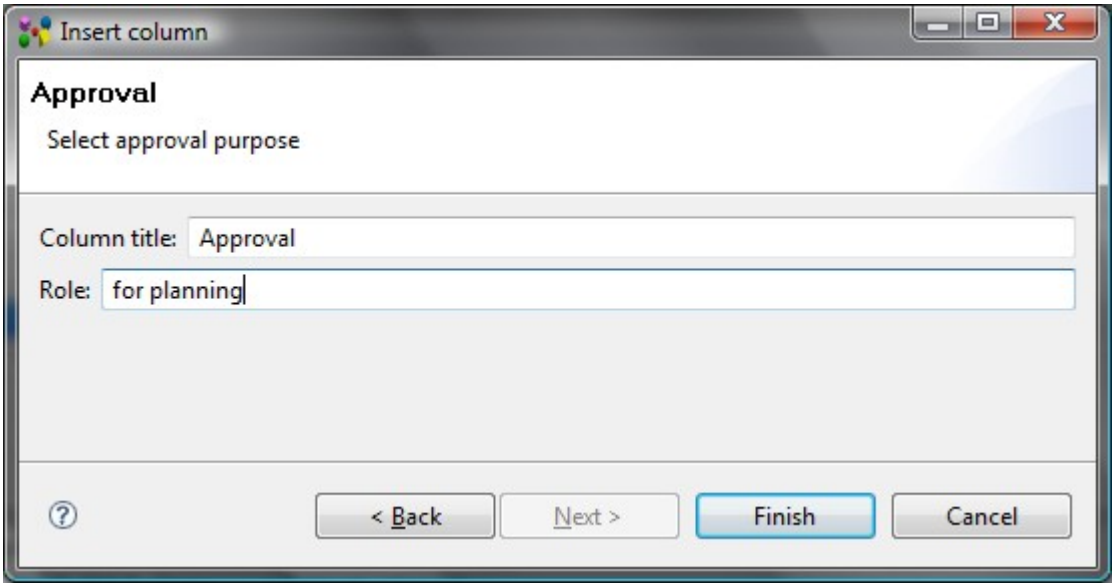

Figure 1. Approval parameters

- After all needed parameters are entered, click **Finish** button to create a column. The column of type approval appears in the table.

Follow the routine to fill an approval column:

- Double click on an approval column cell. The **Approval** dialog window appears on the screen (see Figure 2):

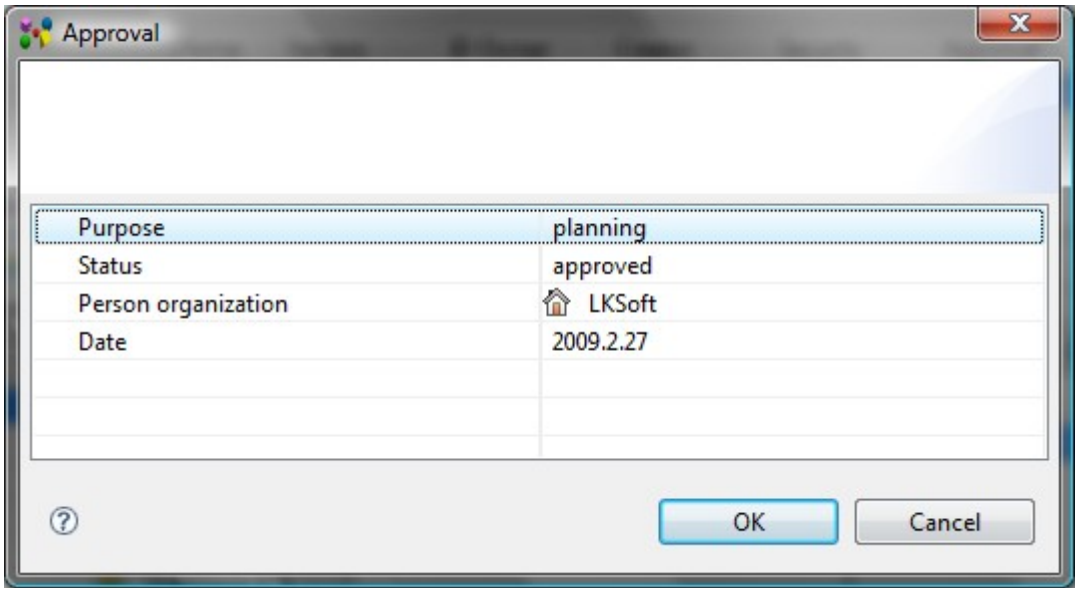

Figure 2. Approval dialog window

  - Enter an applicable value into the **Purpose** field or select the predefined value from the drop down list:
    - **disposition** - the referenced object is approved for series production;
    - **equipment order** - the referenced object has reached a status in which changes are subject to a defined change process and tools and other equipment required for production may be ordered;
    - **planning** - the referenced object is technically complete and has reached a status sufficiently stable so that other designs may be based on it.
  - Enter an applicable value into the **Status** field or select the predefined value from the drop down list:
    - **approved** - the required authorizations have been obtained for a particular role of approval for a piece of product data;

        - **disapproved** - the required authorizations have been denied for a particular role of approval for a piece of product data;
        - **not yet approved** - the required authorizations have not been obtained for a particular role of approval for a piece of product data.
    - Select an organization in the **Person organization** drop down list or use the button next to the field to get the whole list of available organizations.
    - Enter a date of approval or use the button next to the field to get the pop up calendar.

Note. There may be created some columns with the different title and the same role. Data, entered into one of the approval columns, are displayed in all columns with the same role.

To top

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10.6.3. Assembly Component Definition

The *Assembly component definition* is a definition of a component, i.e. the part, which is instantiated as a component in an assembly.
There are no additional parameters to enter in the wizard page (see Figure 1) except a column title. Enter a column title into the **Column title** field and click **Finish** button.

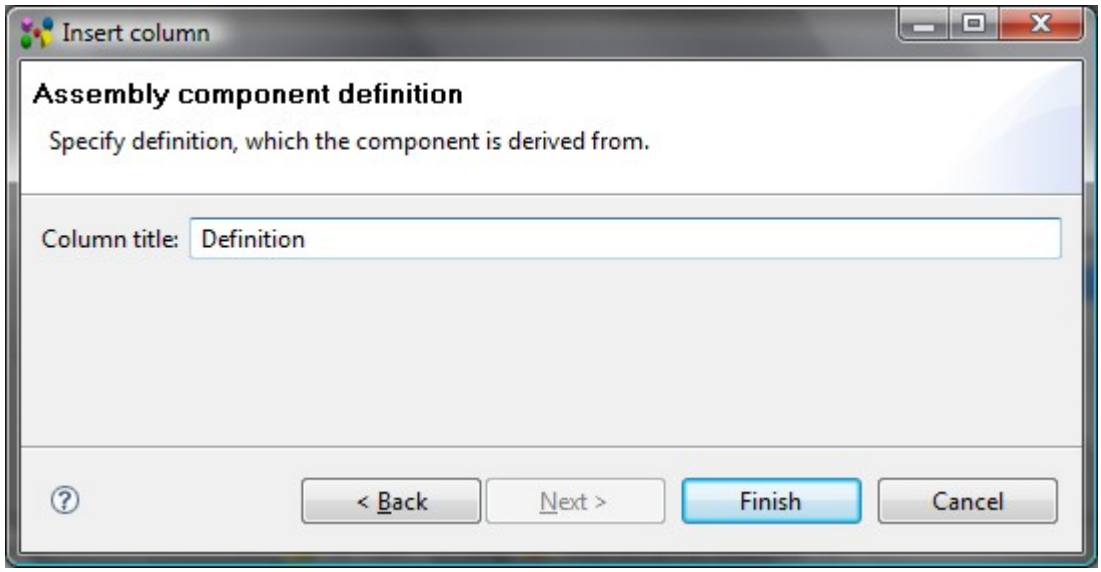

Figure 1. Assembly component definition column parameters

The column of type assembly component definition with a specified title appears in a table. To fill the column with value or edit the existing one:

- Select an applicable value from the drop down list.
  Or
- Double click on the cell.
- The **Select entity** page appears on the screen (see Figure 2) to select an applicable definition to be associated with the selected assembly component.
- Press **OK** button after the selection is made.

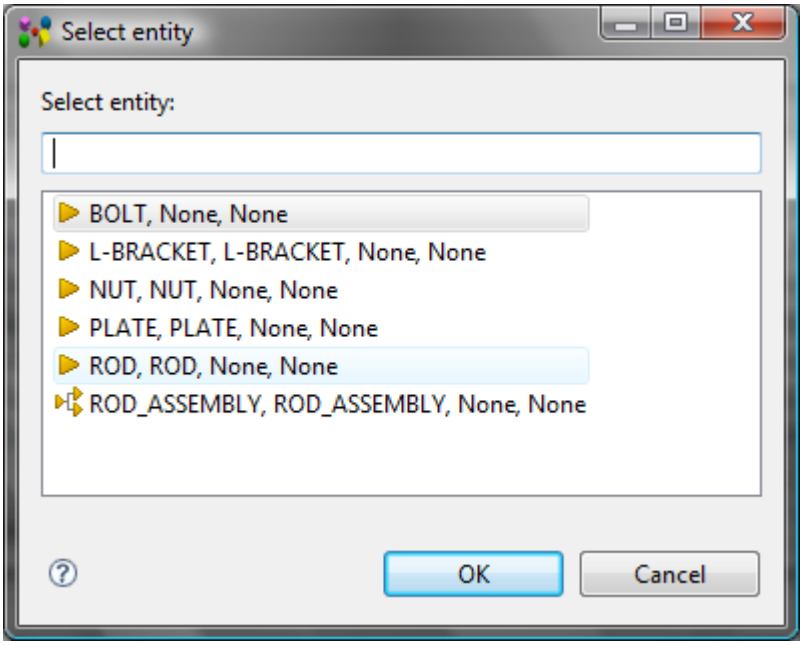

Figure 2. Select entity page with a list of available definitions

The selected definition appears in the selected cell. The value of the cell is composed of some strings, separated by comma: product name, product id,version id,definition id.

To top

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10.6.4. Certification

A *Certification* is a documentation that asserts facts.
Enter parameters for the certification column (see Figure 1):

- Enter a title for the column into the **Column title** field.
- Enter a word or group of words by which the Certification is referred to into the **Name** filed.

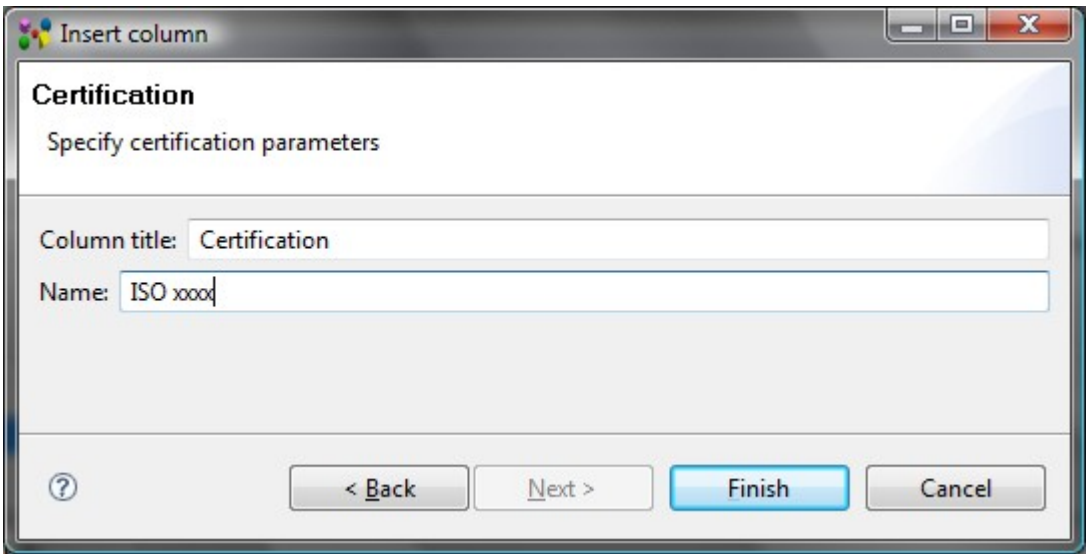

Figure 1. Insert column wizard page to enter certification parameters

After all needed parameters are entered, click **Finish** button to create a column. The column of type approval appears in the table (see Figure 2).

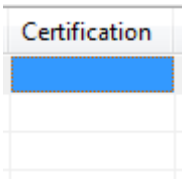

Figure 2. Certification column

A certification column cell displays a name of the certification if one is present, or nothing - if the certification is not present. Follow the routine to fill a certification column with value:

- If a column cell is empty, click on the cell. The name of the certificate (entered while the certificate column creation) appears in the cell.
- The click on the cell with value removes the value from the cell.

Note. There may be created some columns with the different title and the same name. Data, entered into one of the certification columns, are displayed in all columns with the same name.

To top

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10.6.5. Classification

A *Classification association* associates a classification with an object.
Enter the classification association parameters (see Figure 1):

- Enter a title for the column into the **Column title** field.
- Enter the relationship value between classification and associated object.

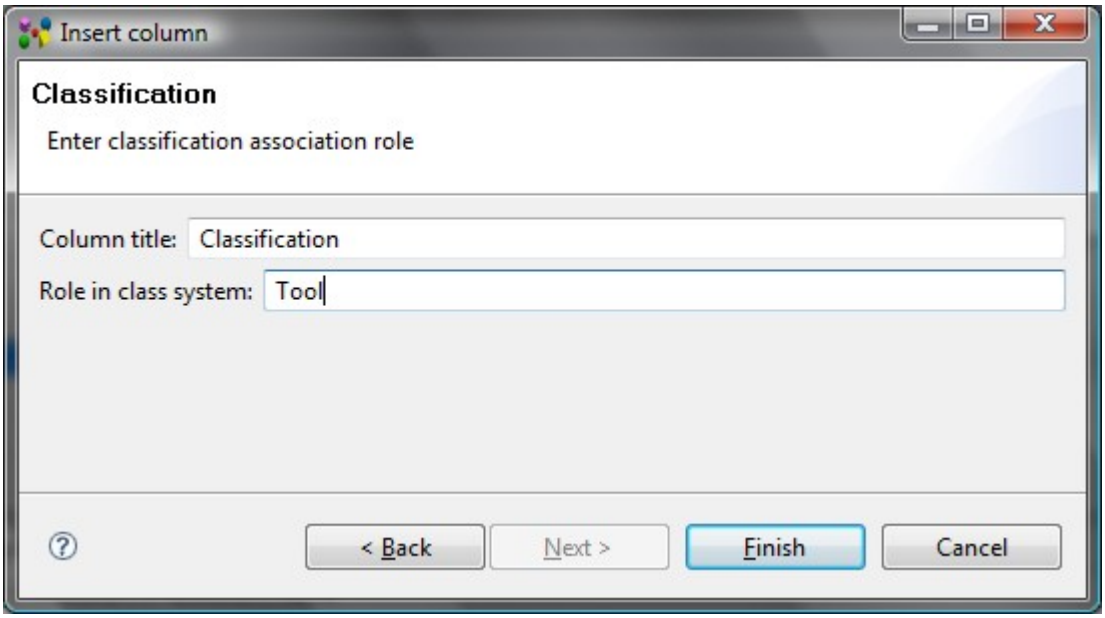

Figure 1. Classification association parameters

- After all needed parameters are entered, click **Finish** button to create a column. The column of type classification is created (see Figure 2).

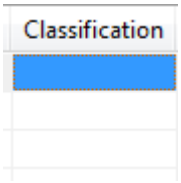

Figure 2. Column of type classification

You can specify an information about the classification or select one of the predefined values form the drop down list (see Figure 3).

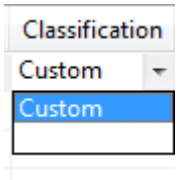

Figure 3. List of predefined values

There can be created several columns with the different column title, but the same role in the classification system. Data entered in one column are displayed in all columns with the same role.

To top

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10.6.6. Chosen Method

The *Chosen method* specifies a task method used to carry out the activity.
There are no additional parameters to enter in the wizard page (see Figure 1) except a column title. Enter a column title into the **Column title** field and click **Finish** button.

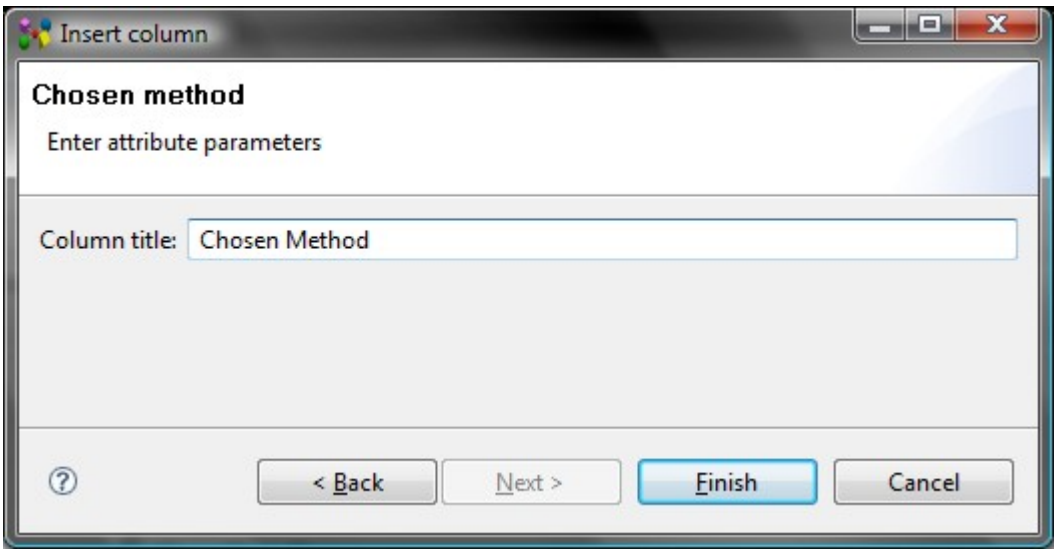

Figure 1. Chosen method column parameters

The column of type chosen method with a specified title appears in a table. To fill the column with value:

- Select an applicable value from the drop down list.
  Or
- Press button next to the cell text field. The **Select entity** page appears on the screen to select an applicable task method to be associated with the referenced object.
- Press **OK** button after the selection is made.
  Or

- Activate **Tasks** view: click on the **Tasks** tab or select **Window -> Show view** and then **PDM -> Tasks** (if the view is not present in the current perspective).
- Select an applicable task method.
- Copy and paste selected task method element into the **Chosen method** column (see Figure 2). You can also use drag and drop to perform the same action.

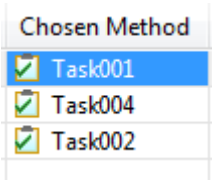

Figure 2. Chosen Method column with values

If there are created a few columns of the same type, all columns for the same object contain the same data.

To top

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10.6.7. Curve Color

The *Curve Color* specifies a colour of the curve/edge of the referenced geometry object.
There are no additional parameters to enter in the wizard page (see Figure 1) except a column title. Enter a column title into the **Column title** field and click **Finish** button.

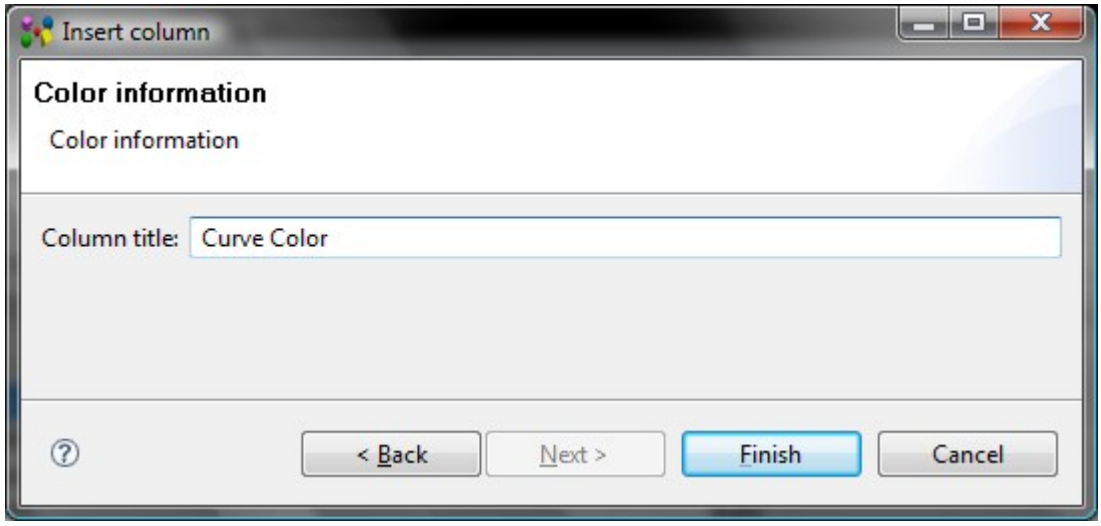

Figure 1. Insert column dialog to enter colour information

The column of type color with a specified title appears in a table. To fill the column with value:

- Double click on the column cell or press **F2** button.
- Select an applicable colour in the appeared **Color** window and press **OK** button (see Figure 2).

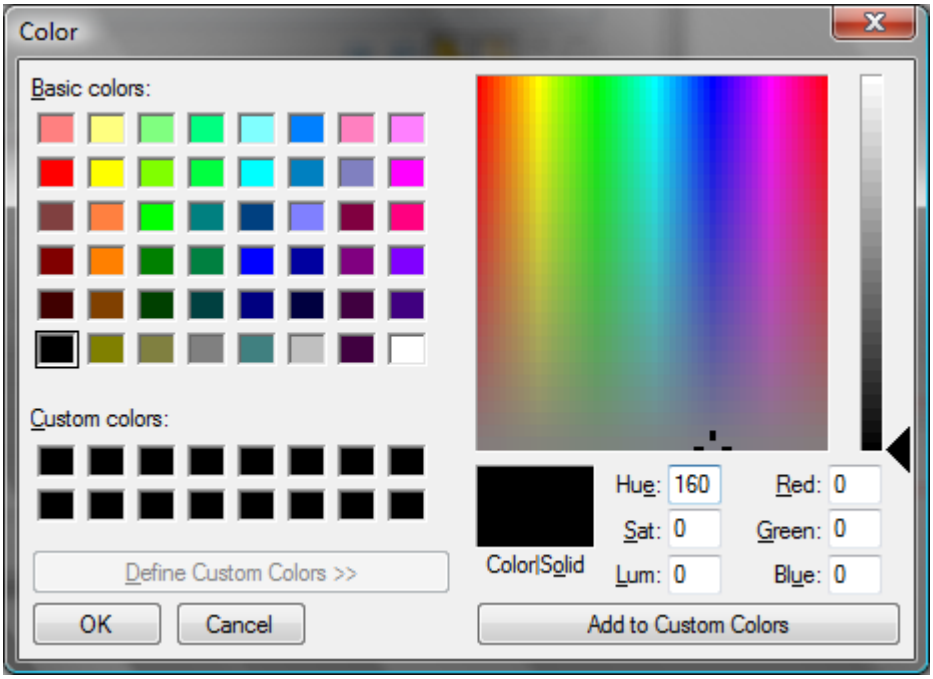

Figure 2. Color window to select colour

- The specified colour square appears in the selected column cell (see Figure 3).

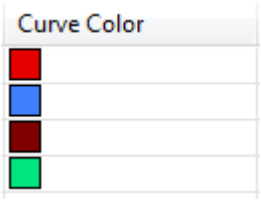

Figure 3. Curve color column with specified colour

To top

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10.6.8. Description

The *Description* specifies additional information about the referenced object.
Enter parameters of the description column (see Figure 1):

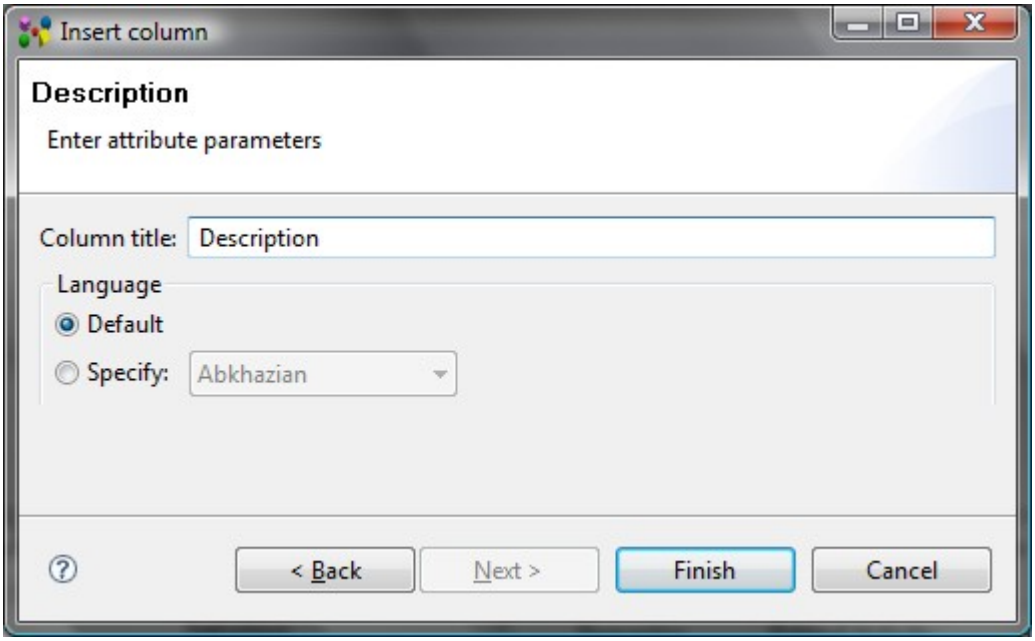

Figure 1. Enter Description column parameters

- Enter a column title into the **Column title** field.
- You can select which language to use for the description:
  - **Default** - the language, which is defined in the IDA-STEP preferences. You can change the default language in the **Language** preferences page (accessible by selecting **Window -> Preferences**, then **General -> Language**).
  - **Specify** - after this radio button is selected, you can select an applicable language in the drop down list.
- Press **Finish** button. The column with a specified title appears in a table.

To fill the column cell with value, simply enter a text into the text field and press ENTER key.
If there are created a few columns of the same type, all columns for the same object contain the same data.

To top

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10.6.9. Date

A *Date* specifies the calendar\ time, defined according to the Georgian calendar, conveying information about the year, the month, and the day in no specific order.
Enter parameters for the date column (see Figure 1):

- Enter a title of the column into the **Column title** field.
- Select an applicable value from the **Role** drop down list:
  - **actual** - the existing or occurring as fact date.
  - **creation** - the referenced object is created at the given date.
  - **end date** - implies a date of rounding off something in a process.
  - **installation** - the referenced object is mounted in a product at the given date.
  - **planned** - the date the referenced object should be in some state.
  - **production** - the referenced object is produced at the given date.
  - **registration** - the referenced object is determined at the given date.
  - **release date** - the referenced object is published at the given date.
  - **request date** - the referenced object is asked for at the given date.

- **signoff** - the referenced object is withdrawn from some product, process and so on, at the given date.
- **start date** - the referenced object was included in some process at the given date.
- **update** - the referenced object was altered at the given date.

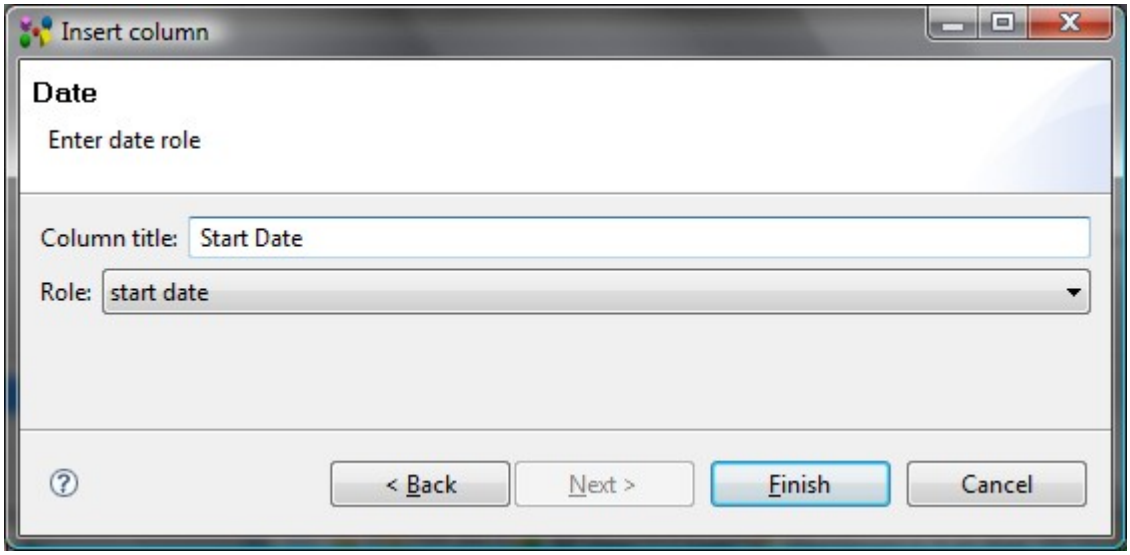

Figure 1. Date parameters

- After all needed parameters are entered, press **Finish** button to create a column. The column of type date appears in the table (see Figure 2).

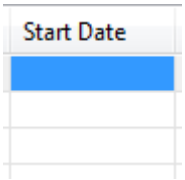

Figure 2. Column of type Date

To fill the column with value:

- Enter an applicable date value into the text box.

Or

- Click on the cell.
- Click on the combo box arrow next to the text box. The calendar sheet appears (see Figure 3).

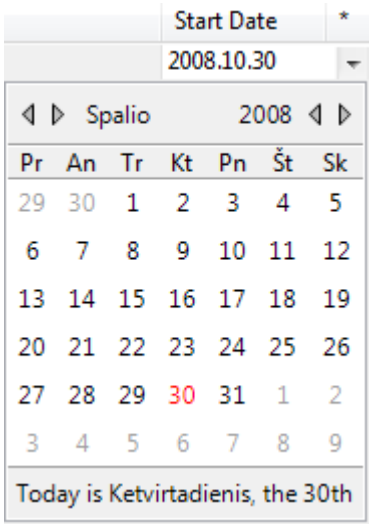

Figure 3. Calendar sheet to select an applicable date

- Select an applicable date from the sheet.
- Press **Enter** key.

If there are created a few columns of the same type, all columns for the same object contain the same data.

To top

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10.6.10. Date and Time

A *Date and Time* represents dates and times based on the Coordinated Universal Time.
Enter parameters for the date column (see Figure 1):

- Enter a title of the column into the **Column title** field.
- Select an applicable value from the **Role** drop down list:
  - **actual** - the existing or occurring as fact date and time.
  - **creation** - the referenced object is created at the given date and time.
  - **end date** - implies a date and time of rounding off something in a process.
  - **installation** - the referenced object is mounted in a product at the given date and time.
  - **planned** - the date and time the referenced object should be in some state.
  - **production** - the referenced object is produced at the given date and time.
  - **registration** - the referenced object is determined at the given date and time.
  - **release date** - the referenced object is published at the given date and time.
  - **request date** - the referenced object is asked for at the given date and time.
  - **signoff** - the referenced object is withdrawn from some product, process and so on, at the given date and time.
  - **start date** - the referenced object was included in some process at the given date and time.
  - **update** - the referenced object was altered at the given date and time.

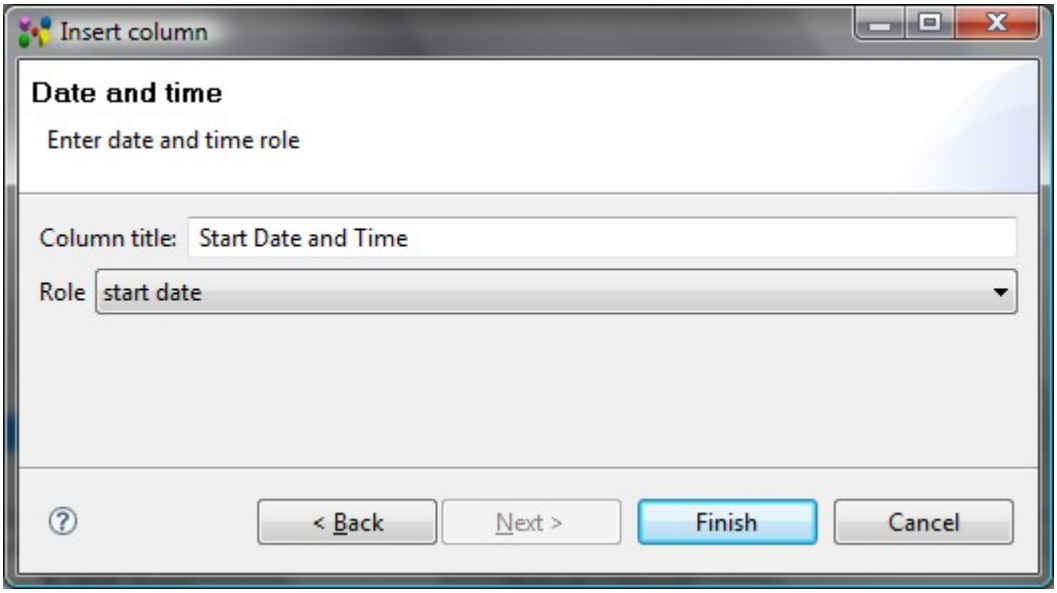

Figure 1. Date and time parameters

- After all needed parameters are entered, click **Finish** button to create a column. The column of type date and time appears in the table.

To fill the column with value:

- Enter an applicable date value into the text box.

Or

- Click on the cell.
- Click on the combo box arrow next to the text box. The date and time sheet appears (see Figure 2).

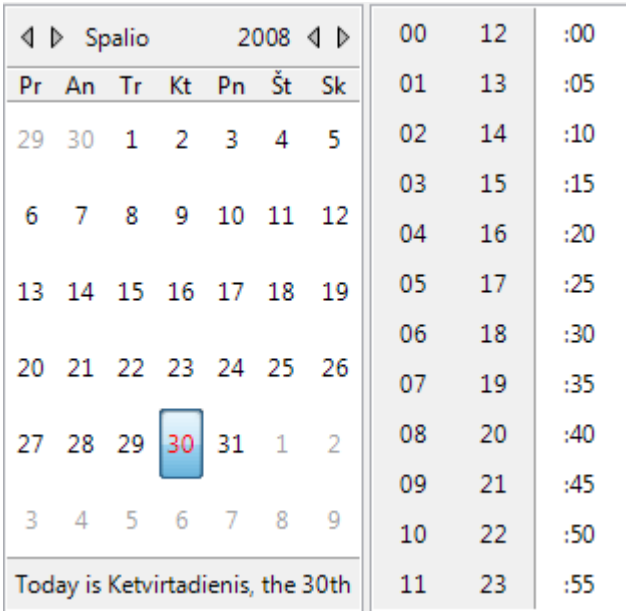

Figure 2. Date and time sheet

- Select an applicable date and time from the sheet.
- Press **Enter** key.

If there are created a few columns of the same type, all columns for the same object contain the same data.

To top

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10.6.11. Design

The *Design* column specifies a part, which is a design for a particular product as individual.
There are no additional parameters to enter in the wizard page (see Figure 1) except a column title. Enter a column title into the **Column title** field and click **Finish** button.

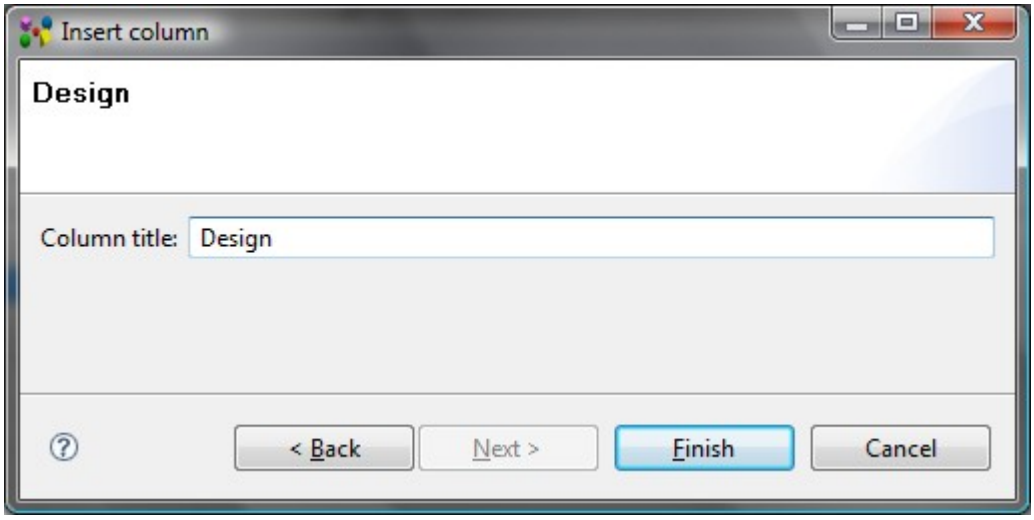

Figure 1. Design column parameters

The column of type design with a specified title appears in a table. To fill the column with value:

- Activate **Part Details** or **Parts** view.
- Select an applicable element in one of the above mentioned views.
- Drag and drop it on the **Design** column cell (see Figure 2).

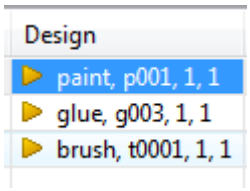

Figure 2. Design column with values

If there are created a few columns of the same type, all columns for the same object contain the same data.

To top

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10.6.12. Document

A *Document* presents an association between a document and an object, where the assigned document provides information about the object it is associated to.
Enter the document parameters (see Figure 1):

- Enter a title for the column into the **Column title** field.
- Specify a role of the document to be associated with the selected object into the **Document role** field.

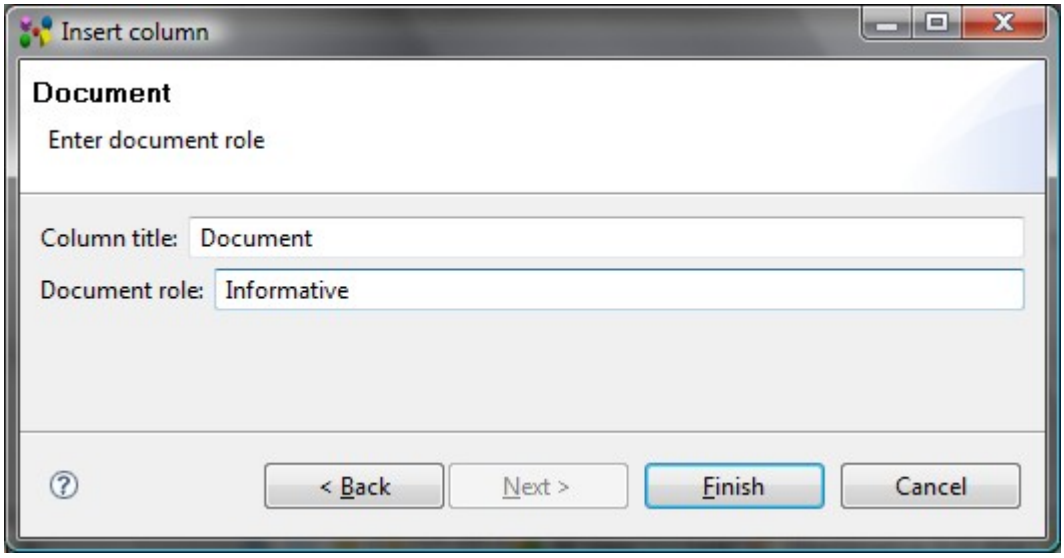

Figure 1. Document parameters

- After all needed parameters are entered, click **Finish** button to create a column. The column of type document appears in the table (see Figure 2).

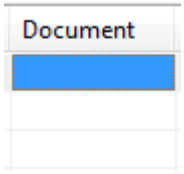

Figure 2. Column of type Document

To fill the column with value:

- Activate **Documents** view: click on the **Documents** tab or select **Window -> Show view** and then **PDM ->** **Documents** (if the view is not present in the current perspective).
- Select an applicable document, document version, file or file reference.
- Copy and paste the selected element into the *Document* column. You can also use drag and drop functionality to perform the same action.

There may be created some columns with the different title and the same document role. Data, entered into one of the document columns, are displayed in all columns with the same document role.

To top

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10.6.13. Element

The *Element* specifies a piece of product data that is under work. The element column content is a particular assembly component, which parent assembly is a design of the *Subject*.
There are no additional parameters to enter in the wizard page (see Figure 1) except a column title. Enter a column title into the **Column title** field and click **Finish** button. The column with a specified title appears in a table.

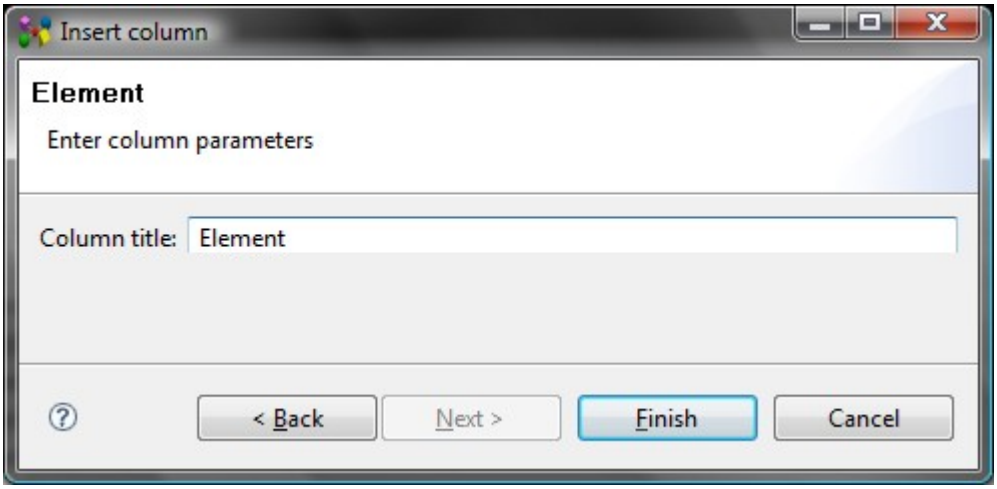

Figure 1. Insert column dialog to enter parameters for element column

To fill the column cell with value:

- Select an element in the drop down list.

Or

- Use the button next to the field to get the whole list of available values. Select an applicable element and press **OK** button.

If there are created a few columns of the same type, all columns for the same object contain the same data.

To top

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10.6.14. External Data Exists

The **External Data Exists** column indicates whether the external data is available for the referenced digital file or not.
There are no additional parameters to enter in the wizard page (see Figure 1) except a column title. Enter a column title into the **Column title** field and click **Finish** button.

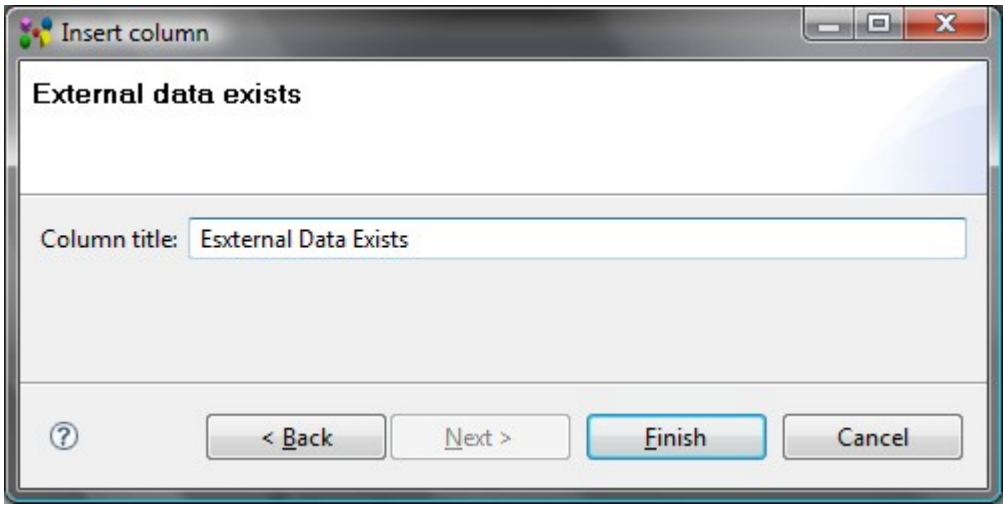

Figure 1. External data exists column parameters

The column cell is read only. The value 'Yes' in this column appears, only if the external data is available for the currently selected digital file.

To top

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10.6.15. External file location

The *External file location* specifies the location of a file in an external storage system.
Enter parameters for the external file location column (see Figure 1):

- Enter a title for the column into the **Column title** field.
- Specify a document location type in the Location type field or select one of the predefined values:
  - **URL** - an absolute path of the external file in your file system or in the Internet,
  - **filename** - a name of the external file,
  - **relative** - a relative path of the external document.
- Press **Finish** button. The column with a specified title appears in the currently selected viewer table.

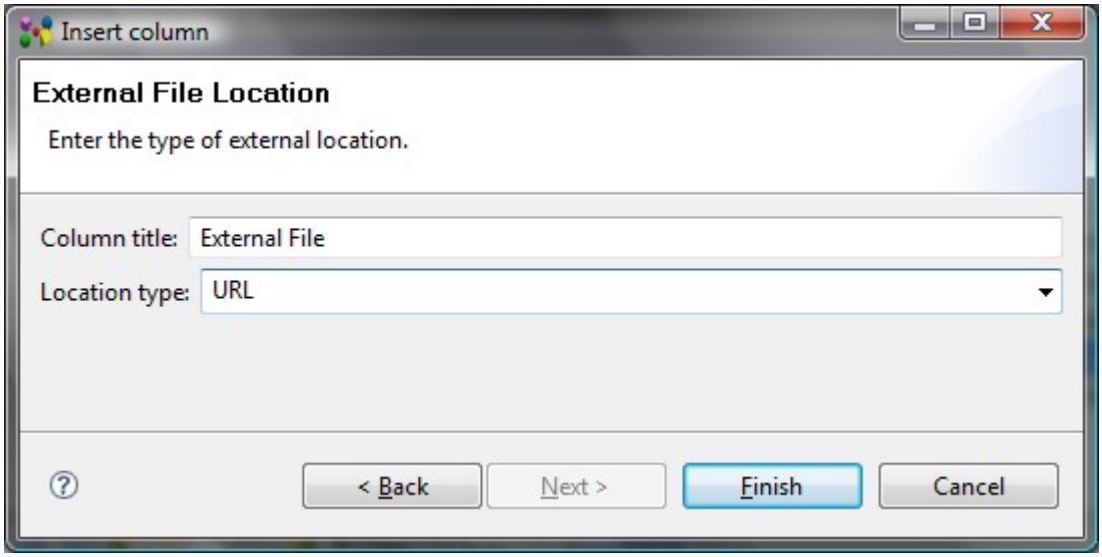

Figure 1. External file parameters

If the type of external location column is URL, the column cell is filled up automatically. **Note:** The value appears in the cell only if the file has an external file associated. You can change the location of the external file:

- Click with mouse on a column cell. The button appears next to the text field.
- Press the button. The dialog appears to choose the target file.
- Select an applicable file and press **Open** button. The path of the recently selected file appears in the cell.

In other cases the value should be entered manually by typing the particular text into the text field of the column cell.

If there are created a few columns of the same type, all columns for the same object contain the same data.

To top

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10.6.16. Id

The *Id* specifies an identifier of the referenced object.
There are no additional parameters to enter in the wizard page (see Figure 1) except a column title. Enter a column title into the **Column title** field and click **Finish** button. The column with a specified title appears in a table.

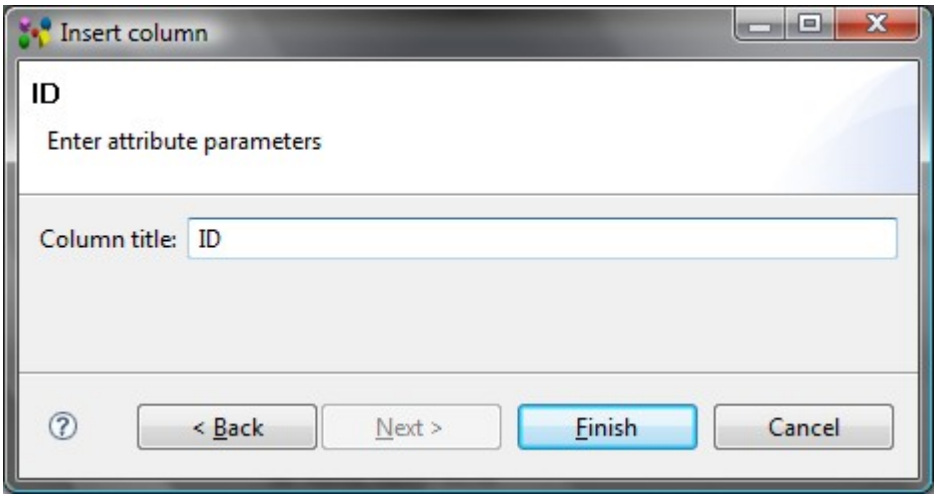

Figure 1. Enter Id column parameters

To fill the column of type ID:

- Enter an identifier of the referenced object into text field.
- Press **Enter** key.

If there are created a few columns of the same type, all columns for the same object contain the same data.

To top

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10.6.17. Magnitude

The *Magnitude* is a numerical quantitative measure associated to a referenced object.
There are no additional parameters to enter in the wizard dialog page (see Figure 1), except a column title. Enter a column title into the **Column title** field and click **Finish** button.

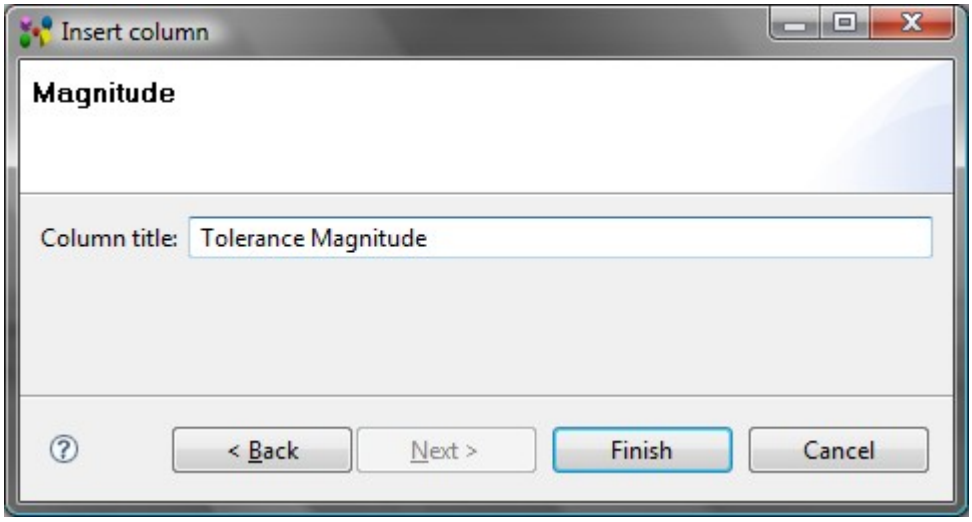

Figure 1. insert column dialog to enter parameters for magnitude

The column cell is read only.
The settings of quantities and units, which appear in the column, are defined in the **Quantity and Units** preferences (**Window -> Preferences -> Numbers -> Quantities and Units**).

To top

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10.6.18. Material property

The *Material property* is a characteristic that depends on material aspects.
Enter parameters for the material property column (see Figure 1):

- Enter a title for the column into the **Column title** field.
- Select a conductivity classification to be associated with a material property from the **Attribute** layout by selecting the particular radio button:
    - **Electric conductivity** - is a measure of a material's ability to conduct an electric current;
    - **Thermal conductivity -** is the property of a material that indicates its ability to conduct heat;
    - **Magnetic permeability** - is the degree of magnetization of a material that responds linearly to an applied magnetic field ;
    - **Optical insertion loss -** is the loss of signal power resulting from the insertion of a device in an optical fiber;
    - **Dielectric permittivity** - is a physical quantity that describes how an electric field affects, and is affected by a dielectric medium, and is determined by the ability of a material to polarize in response to the field, and thereby reduce the total electric field inside the material.

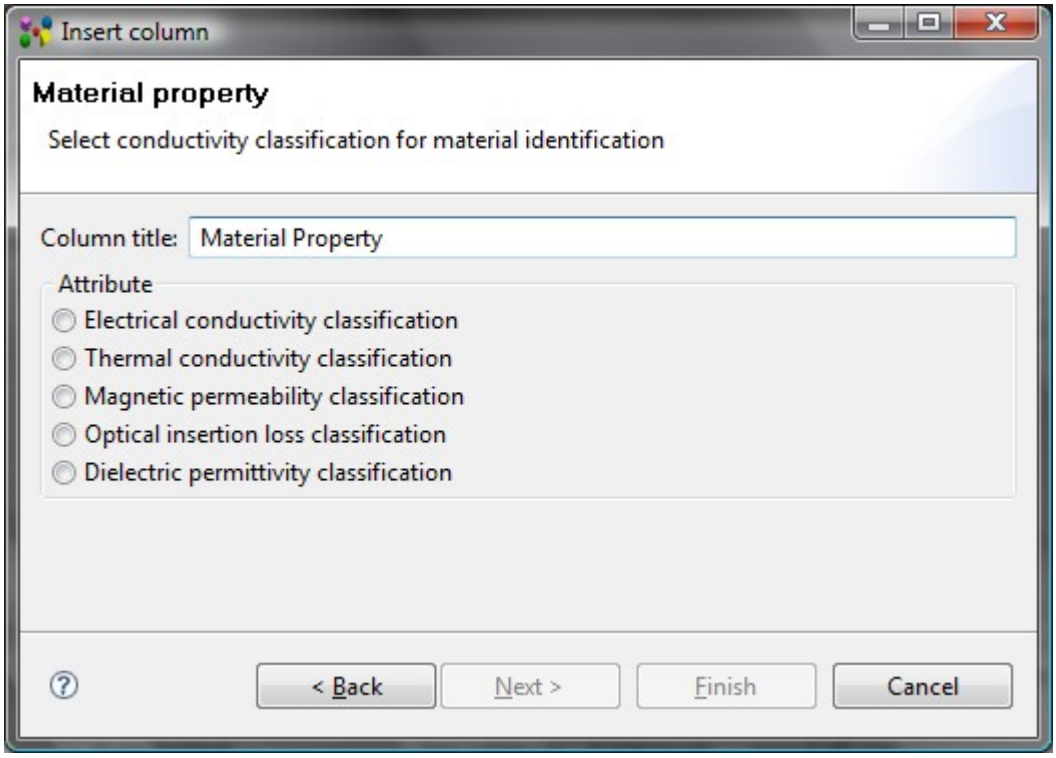

Figure 1. Enter parameters for material property column

- Click **Finish** button to create a column. The column of the specified type appears in the table. (see Figure 2).

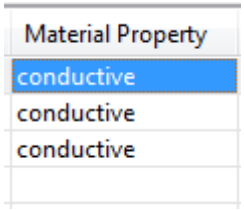

Figure 2. Created column of type material property

The column contains the predefined list of values, which can be selected by the user. The values in the drop down list depend on the conductivity classification, selected in the wizard window:

- *Electrical conductivity classification* column contains the predefined values: conductive, non conductive, resistive, semi conductive and super conductive.
- *Thermal conductivity classification* column contains the predefined values: conductive, non conductive, resistive, semi conductive and super conductive.
- *Magnetic permeability classification* column contains the predefined values: free space permeability, low permeability, medium permeability and highly permeable.
- *Optical insertion loss classification* column contains the predefined values: vacuum, very low loss, low loss, medium loss and high loss.
- *Dielectric permittivity classification* column contains the predefined values: vacuum permittivity, low permittivity, medium permittivity and high permittivity.

There may be created some columns with the different title and the same conductivity classification. Data, entered into one of the material property columns, are displayed in all columns with the same conductivity classification.

To top

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10.6.19. Name

The *Name* specifies the word or group of words by which an object is referred to.
Enter parameters of the name column (see Figure 1):

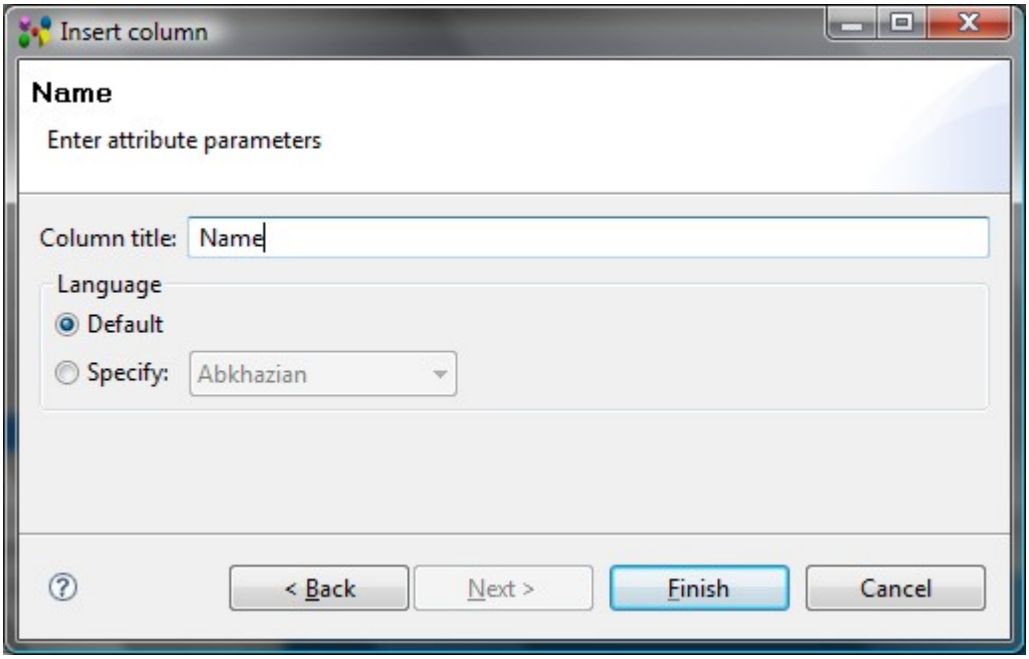

Figure 1. Enter name column parameters

- Enter a column title into the **Column title** field.
- You can select which language to use for the name:
  - **Default** - the language, which is defined in the IDA-STEP preferences. You can change the default language in the **Language** preferences page (accessible by selecting **Window -> Preferences**, then **General -> Language**).
  - **Specify** - after this radio button is selected, you can select an applicable language in the drop down list.
- Press **Finish** button. The column with a specified title appears in a table.

To fill the column cell with value, simply enter a text into the text field and press ENTER key.
If there are created a few columns of the same type, all columns for the same object contain the same data.

To top

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10.6.20. Network Assignment

The *Network assignment* associates a network with a referenced terminal.
There are no additional parameters to enter in the wizard page (see Figure 1) except a column title. Enter a column title into the **Column title** field and click **Finish** button.

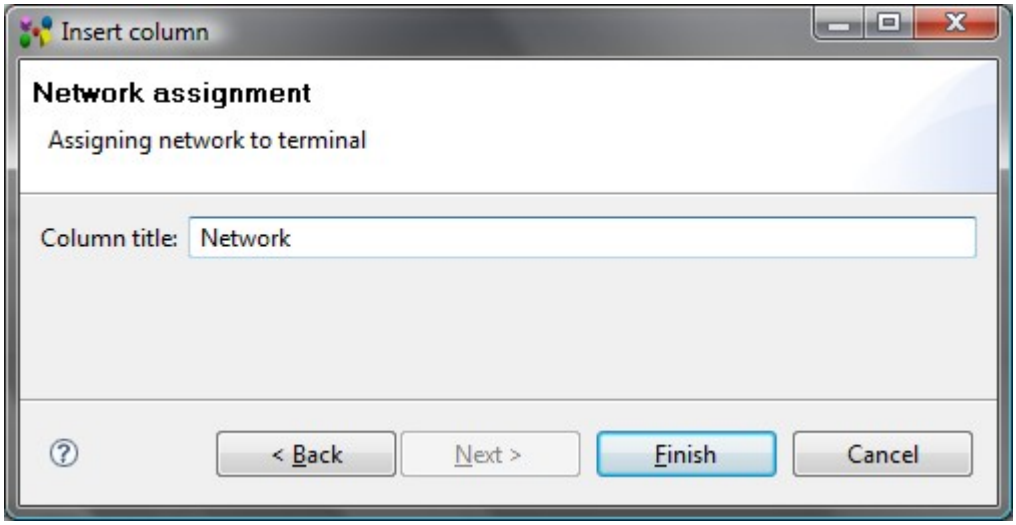

Figure 1. Network assignment column parameters

The column of type network assignment with a specified title appears in a table. To fill the column with value or edit the existing one:

- Select an applicable value from the drop down list.
  Or
- Double click on the cell.
- The **Select entity** page appears on the screen (see Figure 2) to select an applicable network to be associated with the selected terminal.
- Press **OK** button after the selection is made.

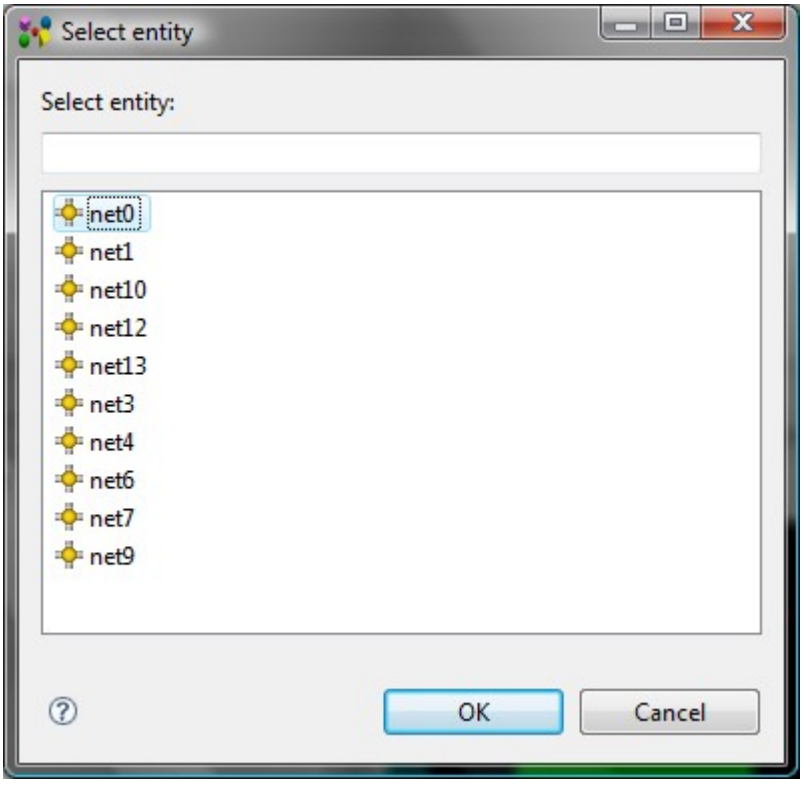

Figure 2. Select entity with the list of available networks

If there are created a few columns of the same type, all columns for the same object contain the same data.

To top

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10.6.21. Person/Organization

The *Person or organization* associates a person or organization with the referenced object.
Enter the person or organization assignment parameters (see Figure 1):

- Enter a title for the column into the **Column title** field.
- Select a role of a person or organization from the **Assignment role** drop down list:
    - **alias scope** - the alias of the referenced object is valid in the assigned person or organization;
    - **author** - the referenced object has been created by the assigned person or organization. The author holds the copyright;
    - **classification officer** - the assigned person or organization is formally responsible for the classification of the referenced object;
    - **concerned organization** - the associated person or organization is interested in or affected by the referenced object;
    - **creator** - the referenced object has been created by the assigned person or organization;
    - **custodian** - the assigned person or organization is responsible for the existence and integrity of the referenced object;
    - **customer** - the assigned person or organization acts as a purchaser or consumer of the referenced object;
    - **design supplier** - the assigned person or organization is the one who delivers the data describing the referenced object;
    - **editor** - the assigned person or organization is responsible for making any changes to any attribute of the referenced object;
    - **id owner** - the assigned person or organization is the one responsible for the designation of an identifier;
    - **location** - the assigned organization is the place where the referenced object can be found or where it takes place;
    - **manufacturer** - the assigned person or organization is the one who produces the actual (physical) object;
    - **notified person or organization** - the person or organization that shall be informed about the referenced object;
    - **organization in contract** - the organization that participates in the contract, where the referenced object is the subject of the contract;
    - **owner** - the assigned person or organization owns the referenced object, and has final say over its disposition and any changes to it;
    - **publisher** - the referenced object has been published by the assigned person or organization;
    - **read access** - the person or organization that has the read access to the referenced object;
    - **request** - the person or organization who issued the referenced object;
    - **scope** - the person or organization is the subject to the referenced object;
    - **supplier** - the assigned person or organization is the one who delivers the actual (physical) object (e.g., a dealer);
    - **supplying organization** - the assigned organization carries out the work;
    - **wholesaler** - the assigned person or organization is the one who is in the sales chain between the manufacturer and the supplier;
    - **write access** - the person or organization that has the write access to the referenced object;

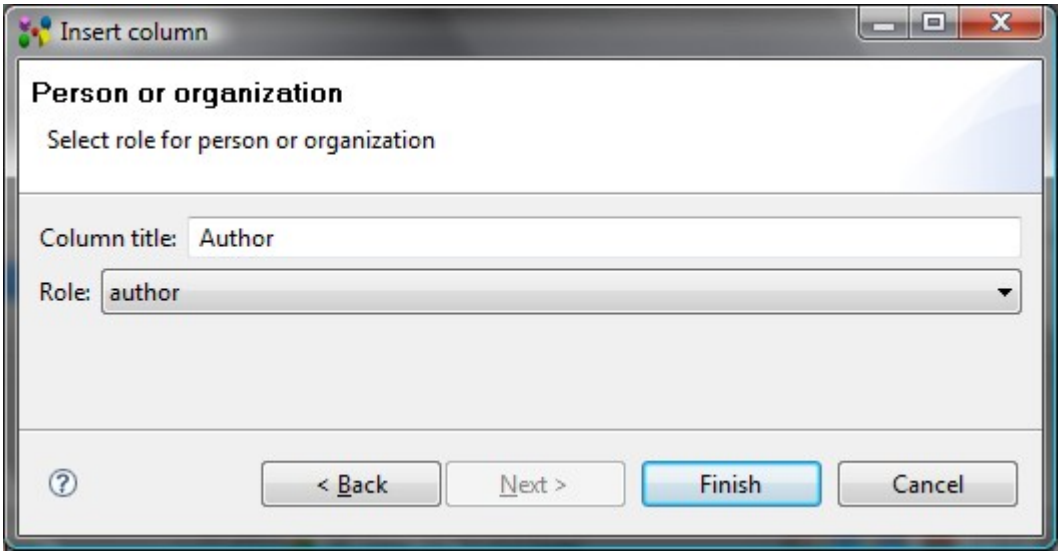

Figure 1. Enter person or organization parameters

- After all needed parameters are entered, click **Finish** button to create a column. The column of type approval appears in the table (see Figure 2).

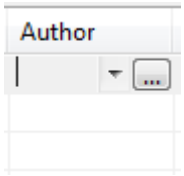

Figure 2. Person or organization column

Follow the routine to fill a person or organization column with value:

- Select an organization in the drop down list (see Figure 3).

Or

- Use the button next to the field to get the whole list of available values (see Figure 4).

Or

- Copy and paste an applicable organization from the **Persons and Organizations** view. You can also use drag and drop functionality to perform the same action.

Figure 3. Drop down list of person or organization column

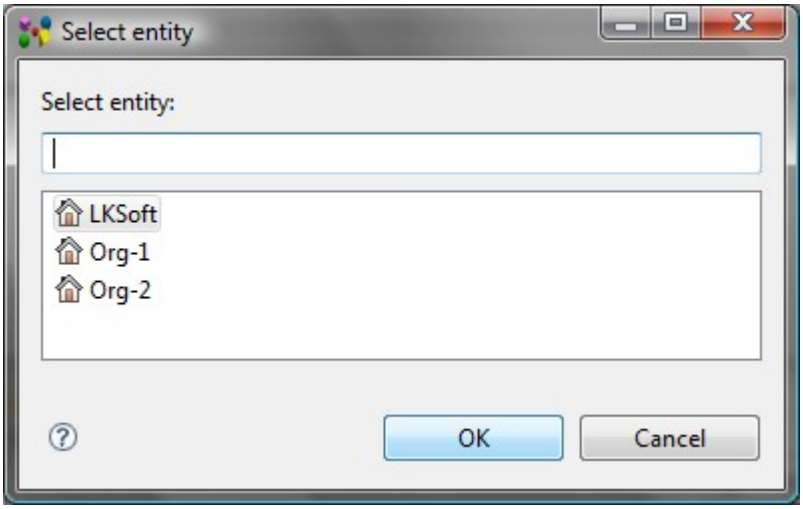

Figure 4. Dialog window with whole list of available organizations

Note. There may be created some columns with the different title and the same role. Data, entered into one of the person or organization columns, are displayed in all columns with the same role.

To top

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10.6.22. Position

The *Position* associates a position in an organization to the referenced object.
Enter a position parameters (see Figure 1):

- Enter a title for the column into the **Column title** field.
- Enter the responsibility of the assigned position with respect to the object that it is applied to into the **Position role** field.
- After all needed parameters are entered, click **Finish** button to create a column. The column of type approval appears in the table.

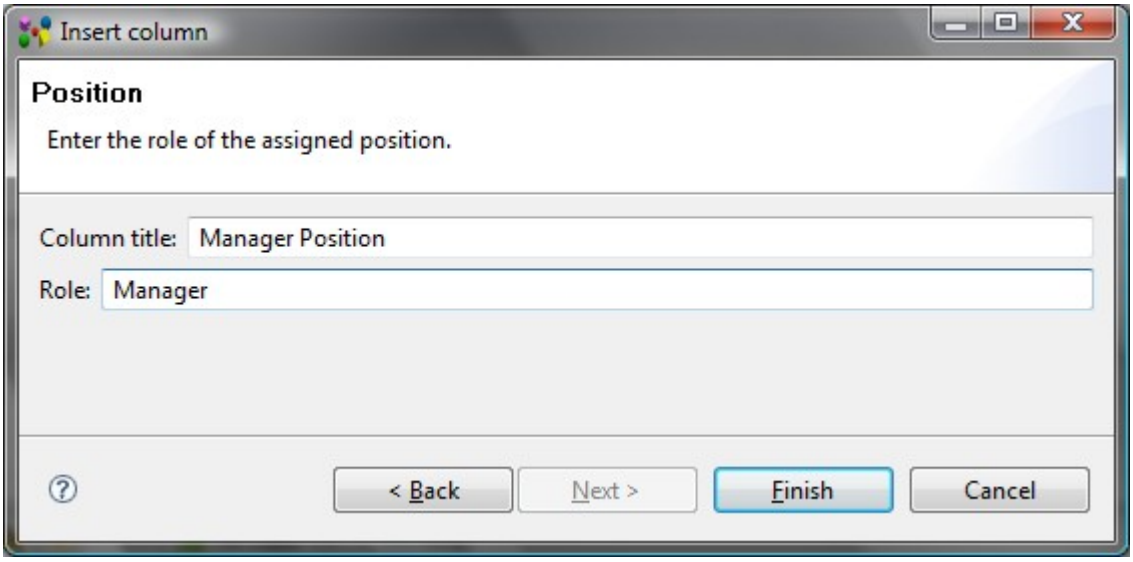

Figure 1. Enter position parameters

To fill the column with value:

- Activate **Persons and Organizations** view: click on the **Persons and Organizations** tab or select **Window -> Show view** and then **PDM -> Persons and Organizations** (if the view is not present in the current perspective).
- Select an applicable position in the **Positions** tabbed panel.
- Copy and paste the selected element into the *Position* column. You can also use drag and drop functionality to perform the same action.

There may be created some columns with the different title and the same role. Data, entered into one of the position columns, are displayed in all columns with the same role.

To top

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10.6.23. Security assignment

The *Security assignment* is the level of confidentiality that is required in order to protect product data against unauthorized usage.
There are no additional parameters to enter in the wizard page (see Figure 1) except a column title. Enter a column title into the **Column title** field and click **Finish** button. The column with a specified title appears in a table (see Figure 2).

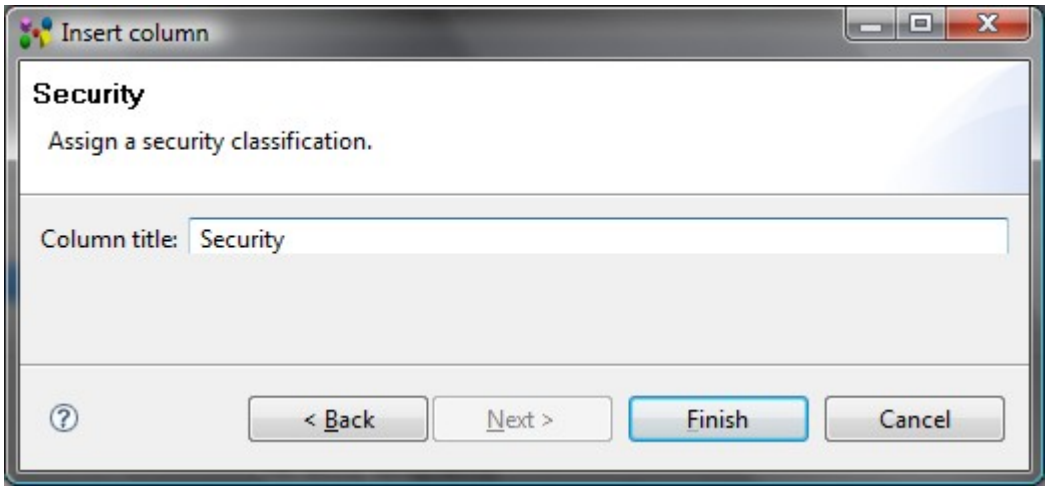

Figure 1. Enter parameters of security assignment

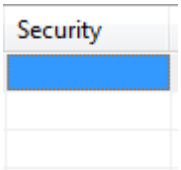

Figure 2. Security column

Follow the routine to fill an approval column with data:

- Double click on a security column cell. The **Security classification** dialog window appears on the screen (see Figure 3):

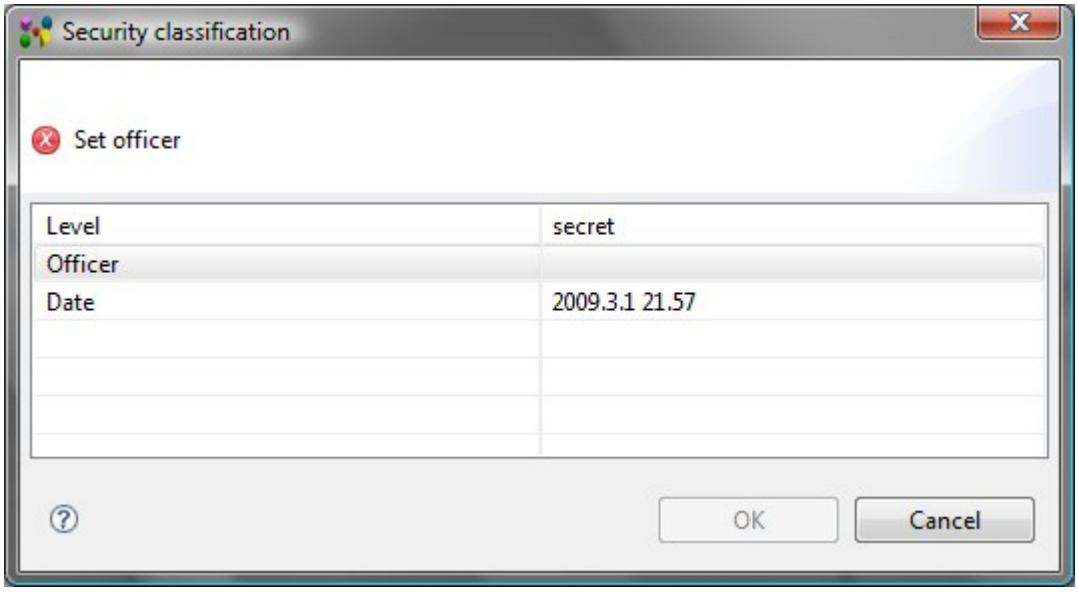

Figure 3. Security classification dialog window

- Select one of the predefined values from the **Level** drop down list:
  - confidential,
  - proprietary,
  - secret,
  - top secret,
  - unclassified.
- Select an officer (a person, which is associated with some organization) in the **Officer** drop down list or use the button next to the field to get the whole list of available officers.
- Enter a date of approval or use the button next to the field to get the pop up calendar.

Note. If there are created a few columns of the same type, all columns for the same object contain the same data.

To top

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10.6.24. Subject

The *Subject* specifies a the product as individual, which element is under work.
There are no additional parameters to enter in the wizard page (see Figure 1) except a column title. Enter a column title into the **Column title** field and click **Finish** button. The column with a specified title appears in a table.

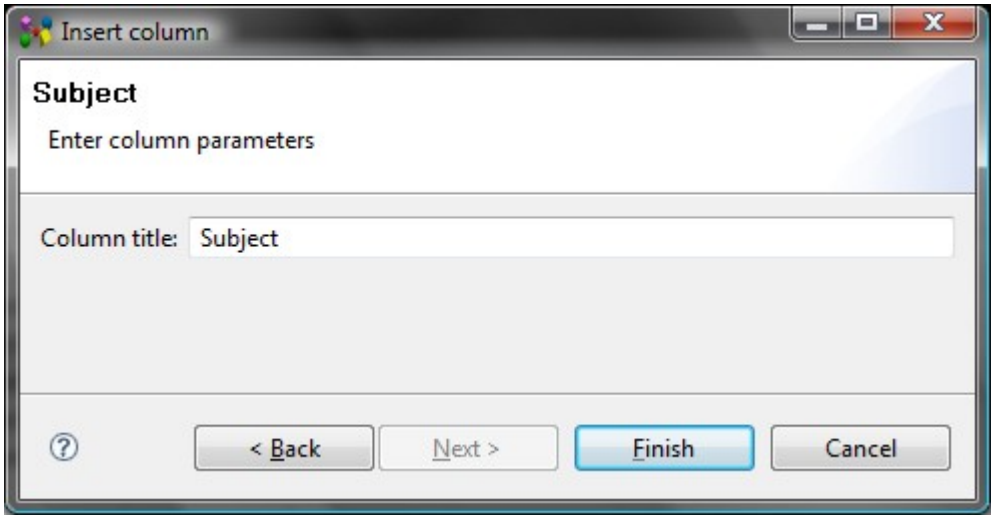

Figure 1. Insert column dialog to enter parameters for subject column

To fill the cell with value:

- Select an applicable product from the drop down list.

Or

- Use the button next to the field to get the whole list of available values. Select an applicable product and press **OK** button.

Or

- Copy and paste an applicable product from the **Products** view. You can also use drag and drop functionality to perform the same action.

If there are created a few columns of the same type, all columns for the same object contain the same data.

To top

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10.6.25. Surface Color

The *Surface Color* specifies a colour of the surface of the referenced geometry object.
There are no additional parameters to enter in the wizard page (see Figure 1) except a column title. Enter a column title into the **Column title** field and click **Finish** button.

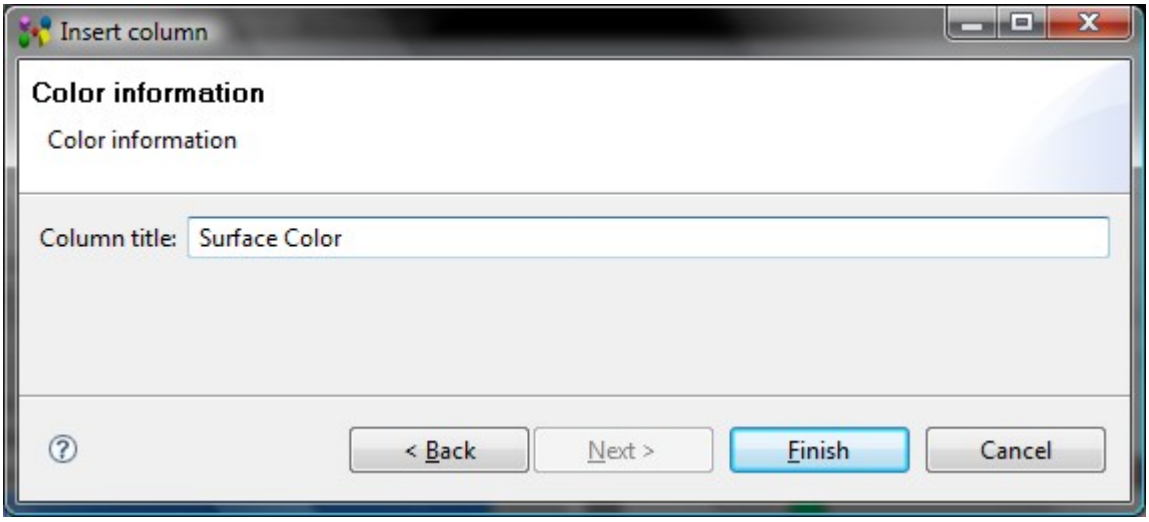

Figure 1. Insert column dialog to enter colour information

The column of type color with a specified title appears in a table. To fill the column with value:

- Double click on the column cell or press **F2** button.
- Select an applicable colour in the appeared **Color** window and press **OK** button (see Figure 2).

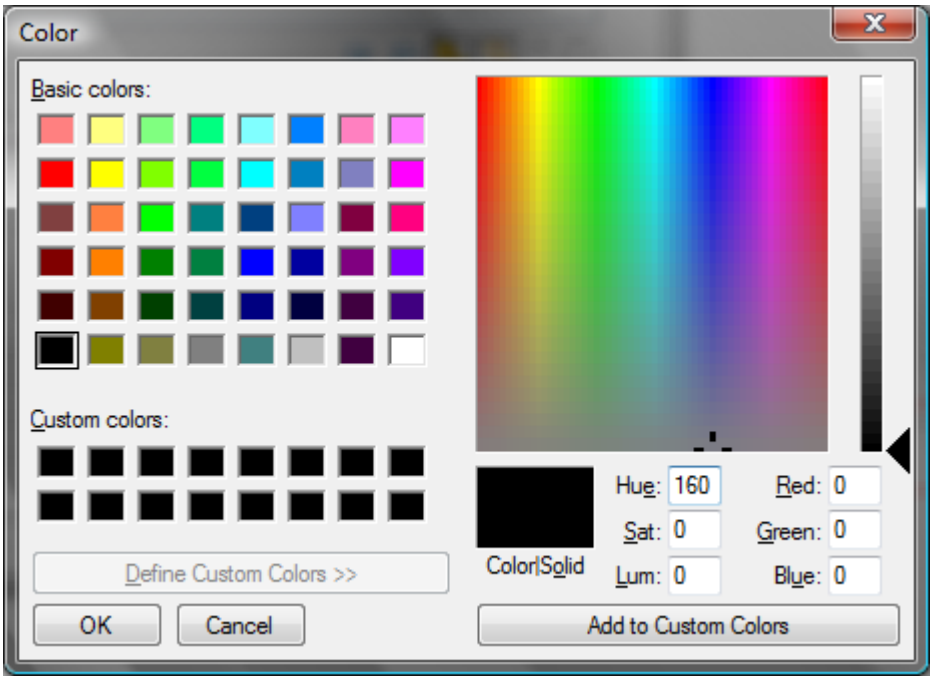

Figure 2. Color window to select colour

- The specified colour square appears in the selected column cell (see Figure 3).

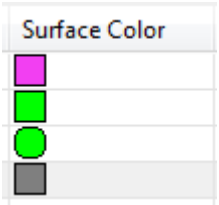

Figure 3. Surface color column with specified colour

To top

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10.6.26. Text property

The *Text property* associates a text with a referenced object.
Enter the text property parameters (see Figure 1):

- Enter a title for the column into the **Column title** field.
- Enter an identifier of a property into the **Property id** field or select an applicable identifier from the drop down list.
- Enter the text, by which the text property is known, into the **Property name** field.
- You can select which language to use for the text property:
    - **None (not translatable)** - the content of the column is not translatable.
    - **Default** - the language, which is defined in the IDA-STEP preferences. You can change the default language in the **Language** preferences page (accessible by selecting **Window -> Preferences**, then **General -> Language**).
    - **Specify** - after this radio button is selected, you can select an applicable language in the drop down list.
- After all needed parameters are entered, click **Finish** button to create a column. The column with the specified column title appears in the table

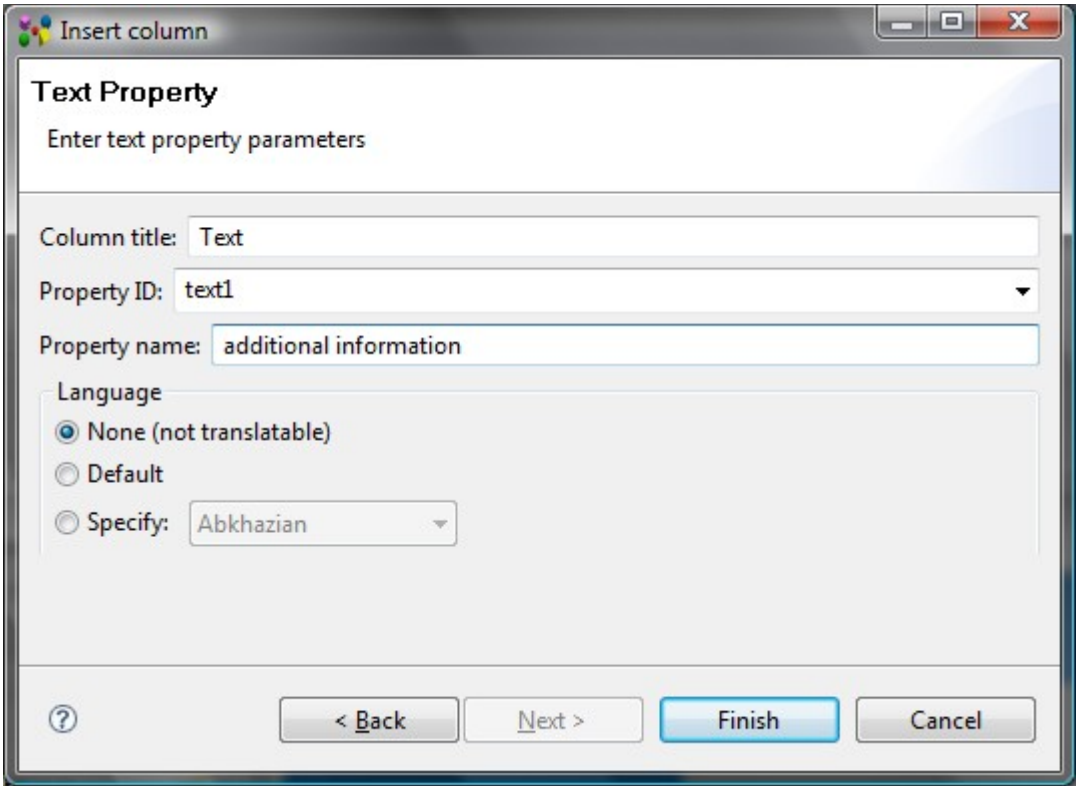

Figure 1. Enter text property parameters

To fill the column cell with value, simply enter a text into the text field and press ENTER key.
There may be created some columns with the different title, property name and the same property identifier. Data, entered into one of the text property columns, are displayed in all columns with the same property identifier.

To top

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10.6.27. Transformation

A *Transformation* is a geometric transformation composed of translation and rotation.
There are no additional parameters to enter in the wizard page (see Figure 1) except a column title. Enter a column title into the **Column title** field and click **Finish** button.

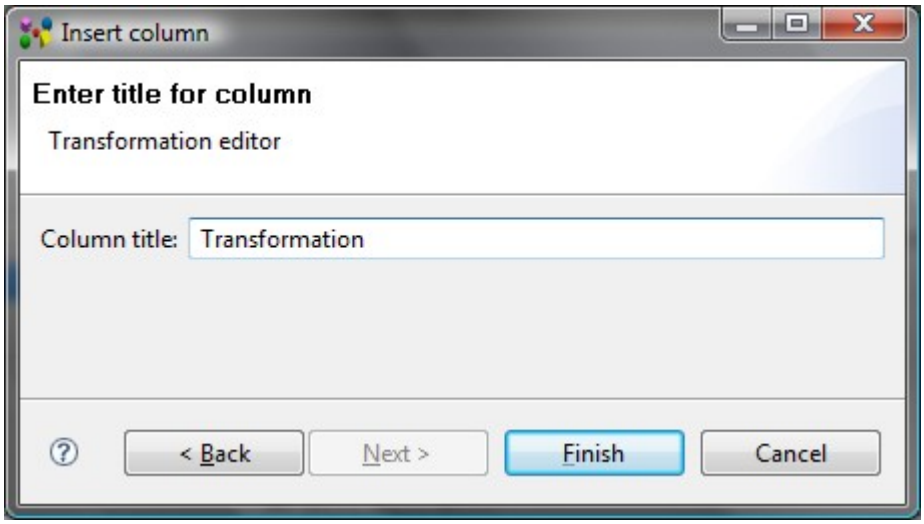

Figure 1. Transformation column parameters

The column of type transformation with a specified title appears in a table. The column is filled with default values (all parameters are set 0). The units, which appear in the column may be changed in the Quantities and Units preferences page (accessible by selecting in the main menu **Window --> Preferences**, then **Numbers -> Quantities and Units**).
Follow the routine to change transformation data:

- Double click on an applicable cell in the column of type transformation.
- The **3D transformation editor** page appears on the screen (see Figure 2):

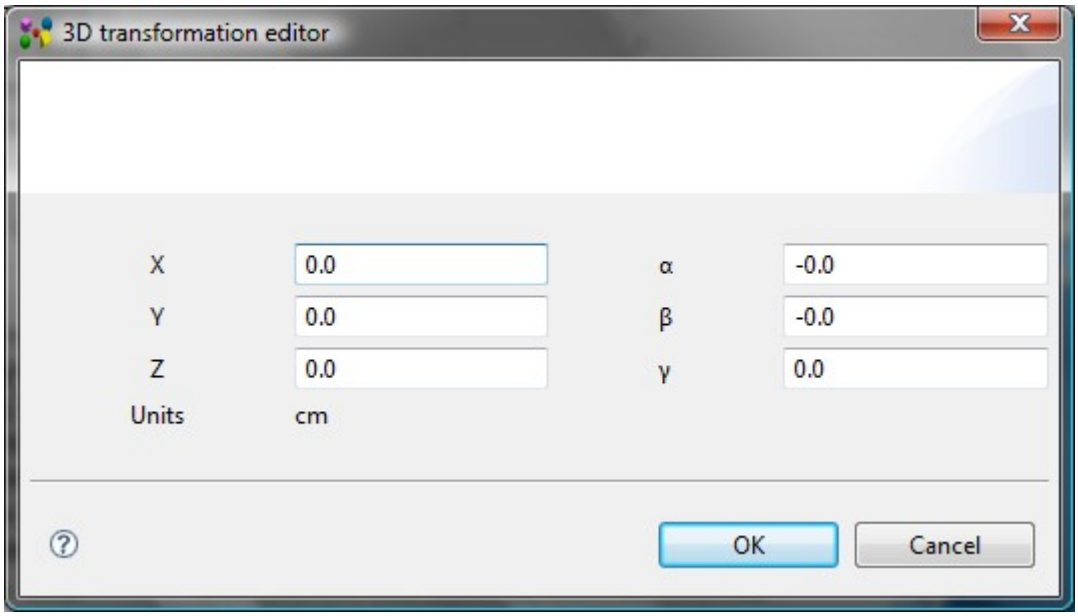

Figure 2. 3D transformation editor page

- **X** field to change the translation along X axis;
- **Y** field to change the translation along Y axis;
- **Z** field to change the translation along Z axis;
- **Units** read only field presents the measurement units, used for the translation.
- **α** field to change the rotation around X axis;

- **β** field to change the rotation around Y axis;
- **γ** field to change the rotation around Z axis.

- Press **OK** button to confirm your changes.
  Note. Units in the column are converted to the units, defined in the **Quantity and Units** preferences.

To top

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10.6.28. Transparency Information

The *Transparency information* specifies the transparency level for the referenced object.
There are no additional parameters to enter in the wizard page (see Figure 1) except a column title. Enter a column title into the **Column title** field and click **Finish** button.

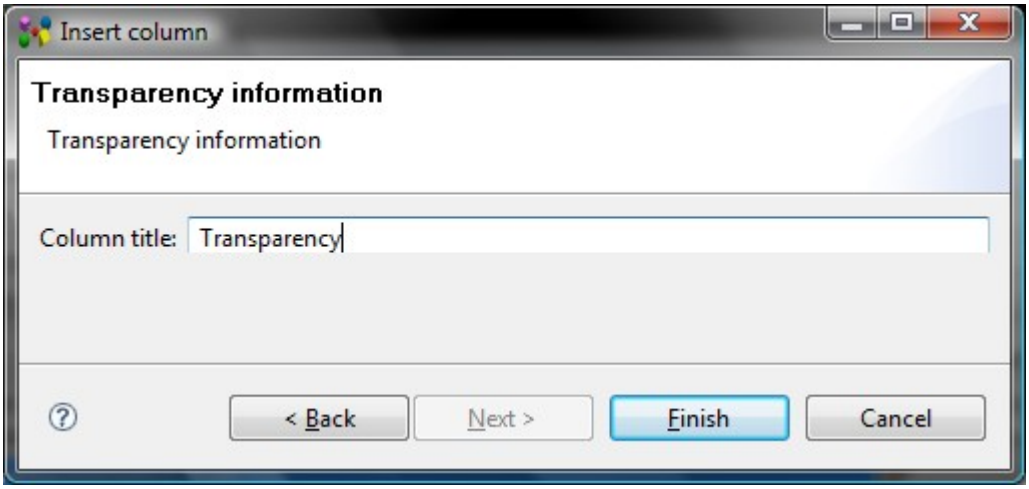

Figure 1. Insert column dialog to enter transparency information parameters

The column of type transparency information with a specified title appears in a table. To fill the column with value:

- Select an applicable predefined value in the **Transparency** column drop down list. The predefined values are: opaque, 25 %, 50 % and 75%.
  Or
- Enter an applicable number value into the **Transparency** column edit box.

To top

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10.6.29. Quantity

The *Quantity* presents the quantity of the referenced object (value with unit).
Enter parameters for the value with unit column (see Figure 1):

- Enter a title for the column into the **Column title** field.

- Enter an identifier of the quantity into the **Property id** field.
- Enter a name of the quantity into the **Property name** field.
- Select a quantity type from the **Type** drop down list. The list values are defined in the **Quantities and Units** page (use main menu **Windows - > Preferences -> Numbers**).

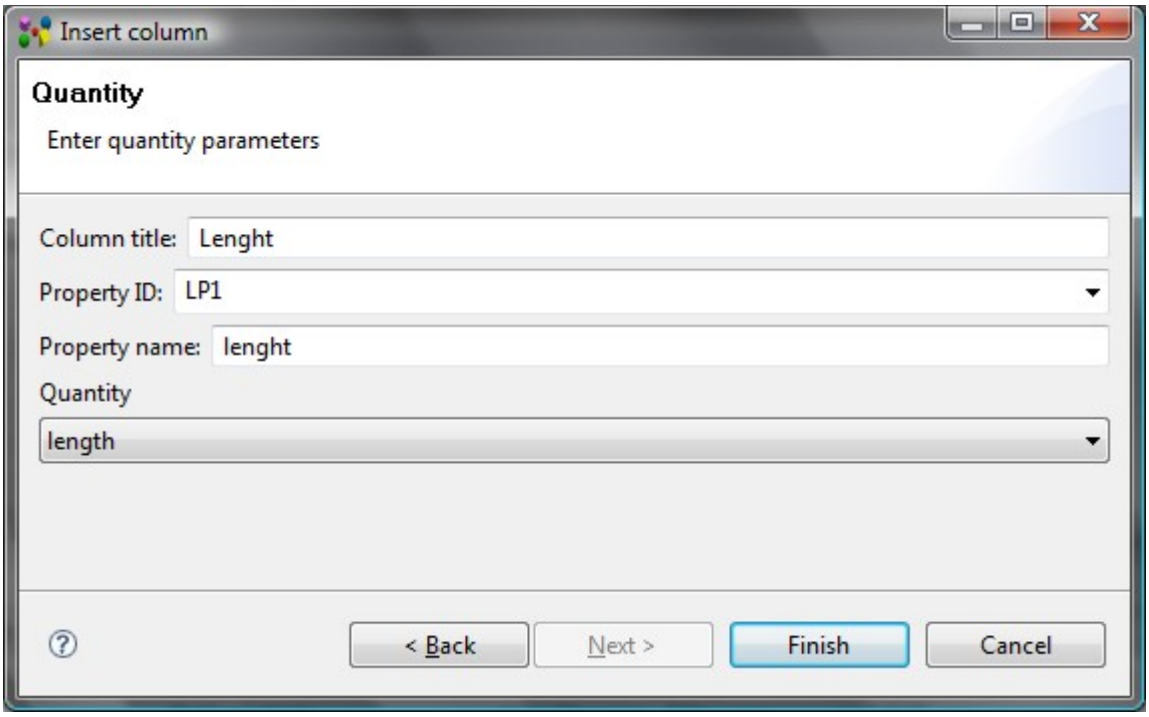

Figure 1. Enter quantity parameters

- After all needed parameters are entered, click **Finish** button to create a column. The column of the specified type appears in the table.

To fill a column cell with value:

- Enter a number value into the text field.
- Select an applicable unit from the drop down list.
- Press ENTER key.

The units, and other parameters of the quantity, which appear in the column, are defined in the **Quantities and Units** page (use main menu **Windows - > Preferences -> Numbers**).
There may be created some columns with the different title and the same property identifier. Data, entered into one of the columns, are displayed in all columns with the same identifier (**Property id** field value).

To top

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10.6.30.1. Enumeration Property

The *Enumeration Property* specifies the worst possible consequence of a failure classified by the degree of injury, property damage, system damage and mission loss that could occur.
Enter parameters for the enumeration property column (see Figure 1):

- Specify a title of the column to be created by entering a word or group of words into **Column title** field.
- Select an identifier of the property from the **Property ID** drop down list or enter a new value.

- Specify a name of the property by entering a word or group of words into **Property name** field.
- Specify the enumeration in the Enums layout by selecting one of the radio buttons:
    - TextEnumerationOneToTen,
    - FailureModeClasses,
    - FailureCauseClasses.

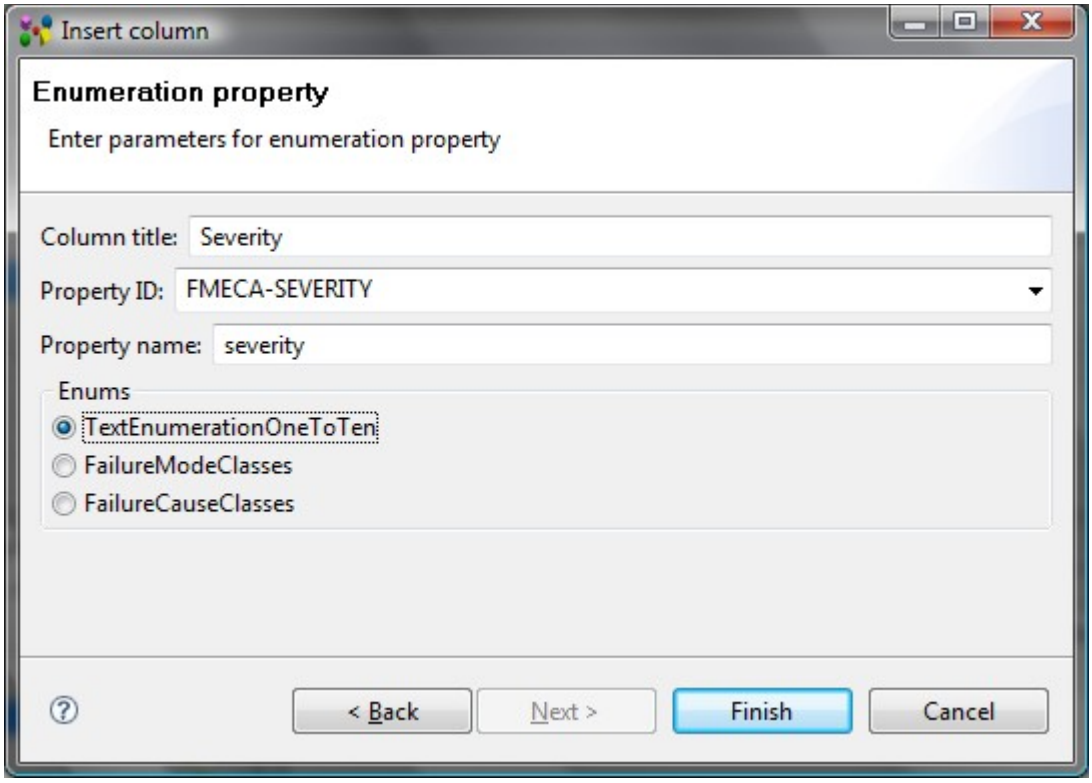

Figure 1. Enumeration property column parameters

The column of type enumeration property with a specified title appears in a table. To fill the column cell with value, select an applicable value from the drop down list.

To top

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10.6.30.2. Failure Detection Method

The *Failure Detection Method* specifies the method by which a failure can be discovered by the system operator under normal system operation or by a maintenance crew carrying out a specific diagnostic action.
There are no additional parameters to enter in the wizard page (see Figure 1) except a column title. Enter a column title into the **Column title** field and click **Finish** button.

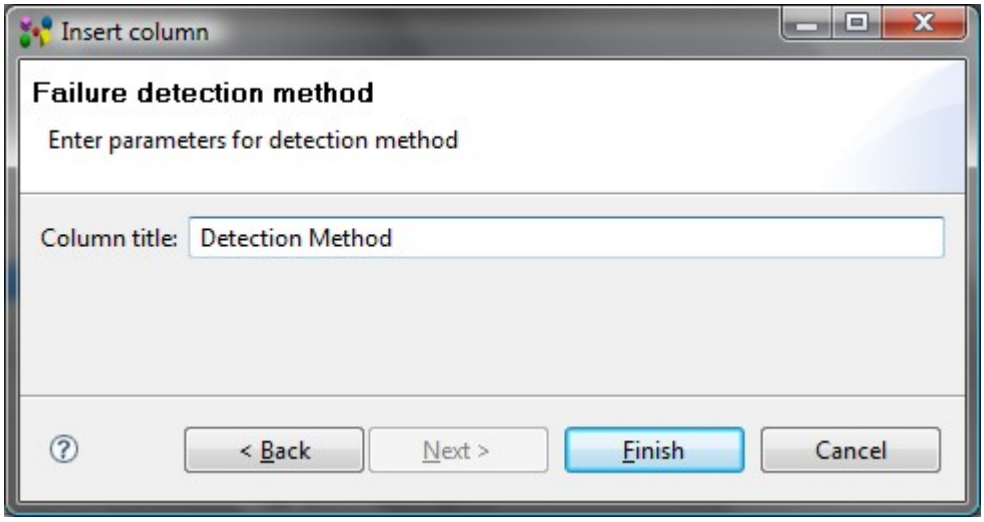

Figure 1. Failure detection method column parameters

The column of type failure detection method with a specified title appears in a table. To fill the column with value or edit the existing one:

- Select an applicable predefined value from the drop down list. The predefined values are: 'Alarm', 'Built in test', 'Non destructive examination', 'Vibration analysis', 'Particle analysis', 'Chemical analysis', 'Spectographic analysis', 'Visual', 'Audible' and 'Olfactory'.
  Or
- Enter a word or group of words, which describe the failure detection method.

To top

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10.6.30.3. Failure Effect

The *Failure effect* is immediate consequences of a failure on operation, function or functionality, or status of some item.
Enter parameters for the failure effect column (see Figure 1):

- Specify a title of the column to be created by entering a word or group of words into **Column title** field.
- Enter an identifier of the failure effect into the **Id_x** field.
- Specify a name of the failure effect by entering a word or group of words into **Name** field.
- Press **Finish** button.

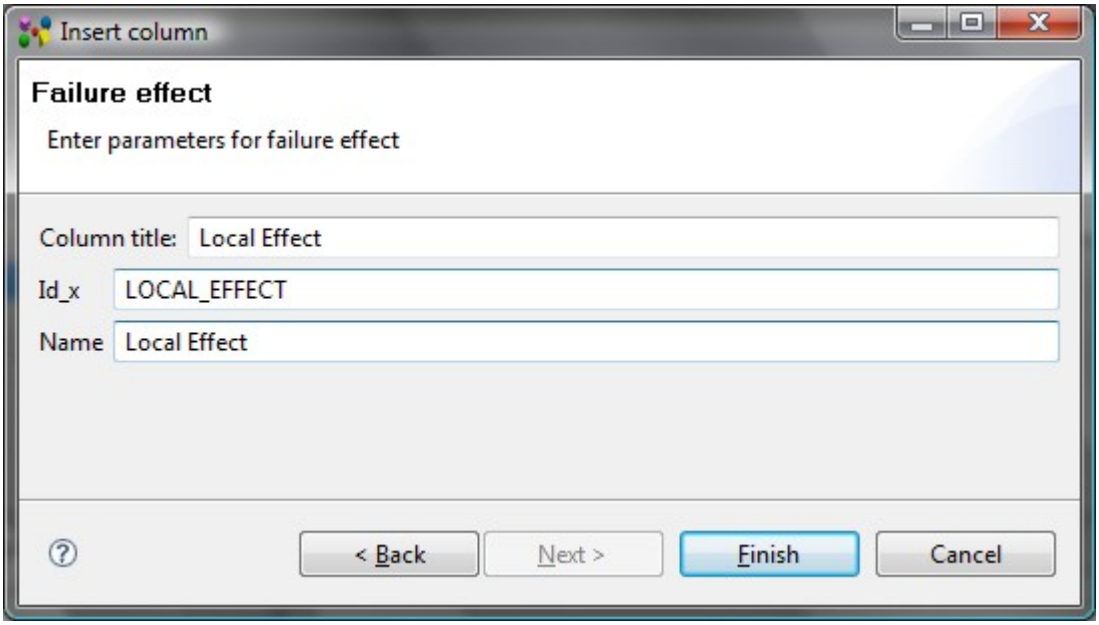

Figure 1. Failure effect column parameters

The column of type failure effect with a specified title appears in a table. To fill the cell with value - type a text into the text field.

To top

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10.6.31.1. Allocated Technology

The *Allocated technology* column presents a definition of allocated passage technology.
There are no additional parameters to enter in the wizard dialog page (see Figure 1), except for a column title. Enter a column title into the **Column title** field and click **Finish** button.

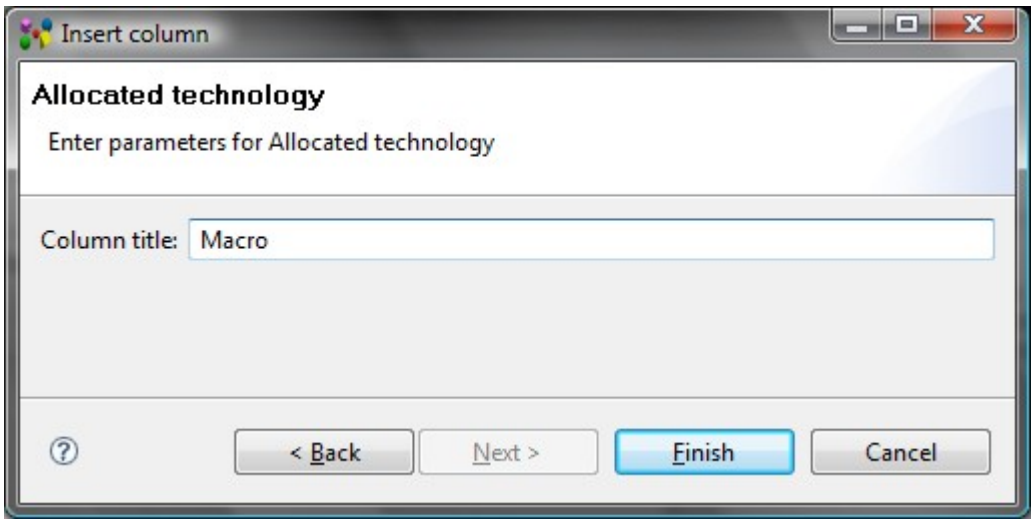

Figure 1. Insert column dialog to enter parameters for allocated technology column

The column with a specified title appears in a table. To fill the column with value:

- Click with a mouse or press **F2** key on the column of type *allocated technology* to enter into edit mode.
- Select a passage technology definition from the drop down list, if present, or create a new one by entering a definition into the field.

- Press **Enter** key.

If there are created a few columns of the same type, all columns for the same object contain the same data.

To top

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10.6.31.2. Allowed Component Terminal Extent

The *Allowed component terminal extent* property is available only for passages, which technology type is *component termination*.
There are no additional parameters to enter in the wizard dialog window (see Figure 1), except for a column title. Enter a column title into the **Column title** field and click **Finish** button.

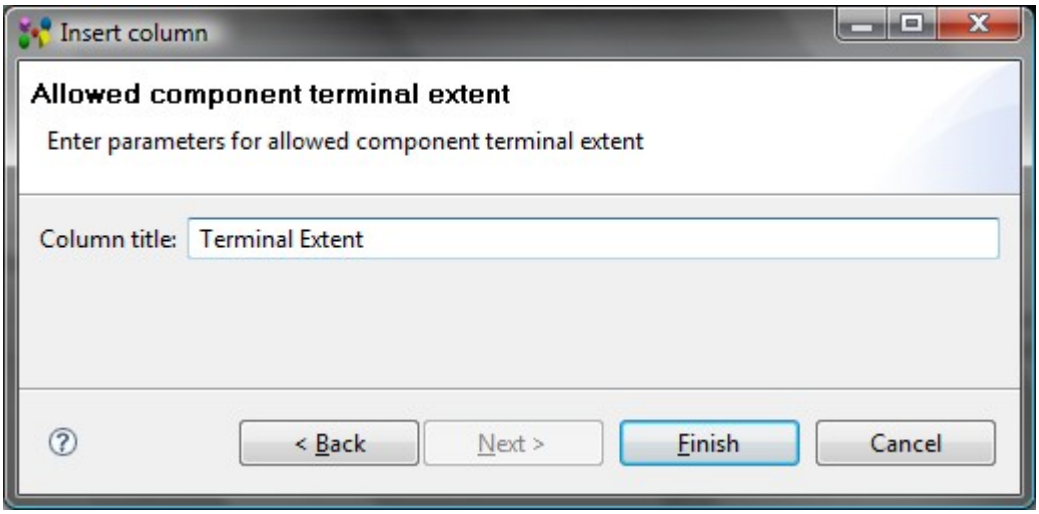

Figure 1. Insert column to enter parameters for allowed component terminal extent column

The column with a specified title appears in a table. To fill the column with value:

- Click with a mouse or press **F2** key on the column of type *allowed component terminal extent* to enter into edit mode.
- You can change parameters of tolerance and units by pressing the button next to the edit field. The Tolerances wizard window appears, where you can select an applicable tolerance and create new units.
- Enter a number into the edit field.
- Press **Enter** key. If current units are not the same as default units, the entered value is recalculated and displayed in default units.

If there are created a few columns of the same type, all columns for the same object contain the same data.
To top

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10.6.31.3. Conductive Layers

The *Conductive layers* column specifies a summation of the set of conductive layers in a referenced

object.
There are no additional parameters to enter in the wizard dialog page (see Figure 1), except for a column title. Enter a column title into the **Column title** field and click **Finish** button.

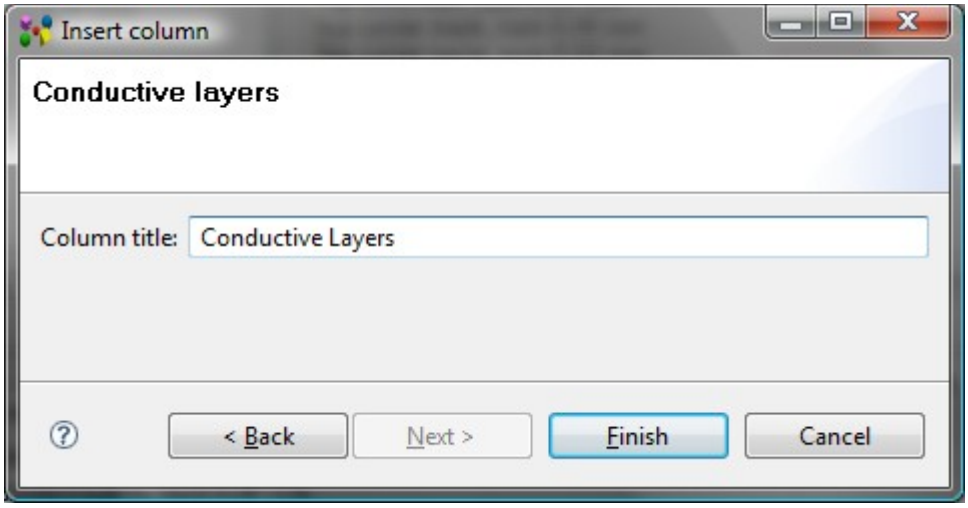

Figure 1. Insert column dialog to enter parameters for conductive layers column

The column with a specified title appears in a table. You can not enter or edit a value in the column. The value is derivable.
If there are created a few columns of the same type, all columns for the same object contain the same data.

To top

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10.6.31.4. Connected Minimum Annular Ring

The *Connected minimum annular ring* column specifies the size of the connected minimum pad area that remains after a hole is drilled through the pad.
There are no additional parameters to enter in the wizard dialog page (see Figure 1), except for a column title. Enter a column title into the **Column title** field and click **Finish** button.

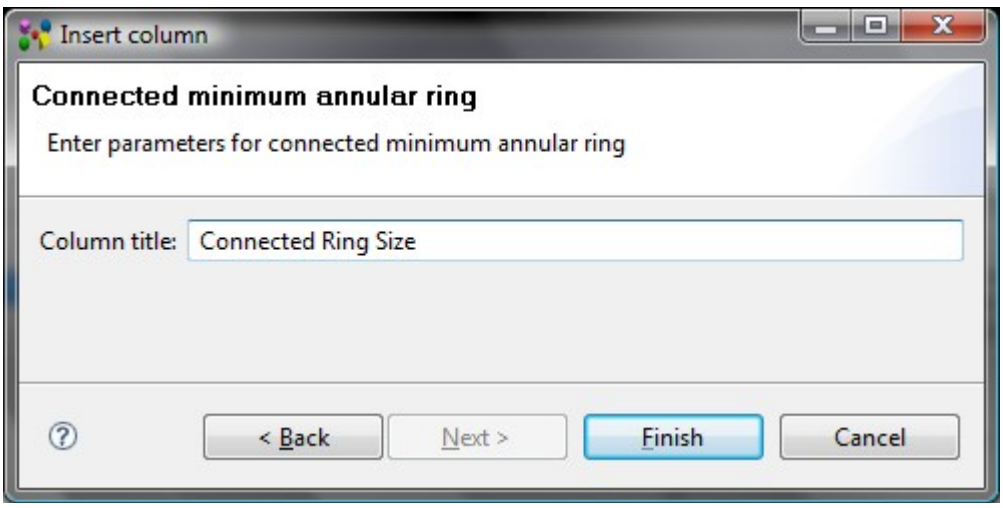

Figure 1. Insert column dialog to add parameters for connected minimum annular ring column

The column with a specified title appears in a table. To fill the column with value:

- Click with a mouse or press **F2** key on the column of type *connected minimum annular ring* to

enter into edit mode.
- You can select an applicable unit from the list next to the text field. The units, and other parameters of the quantity, which appear in the column, are defined in the **Quantities and Units** preferences page (use main menu **Windows - > Preferences -> Numbers**).
- Enter a number into the edit field.
- Press **Enter** key. If current units are not the same as default units, the entered value is recalculated and displayed in default units.

If there are created a few columns of the same type, all columns for the same object contain the same data.

To top

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10.6.31.5. Finished Deposition Thickness

The *Finished deposition thickness* column specifies the finished thickness of a particular deposition material for a passage.
There are no additional parameters to enter in the wizard dialog window (see Figure 1), except for a column title. Enter a column title into the **Column title** field and click **Finish** button.

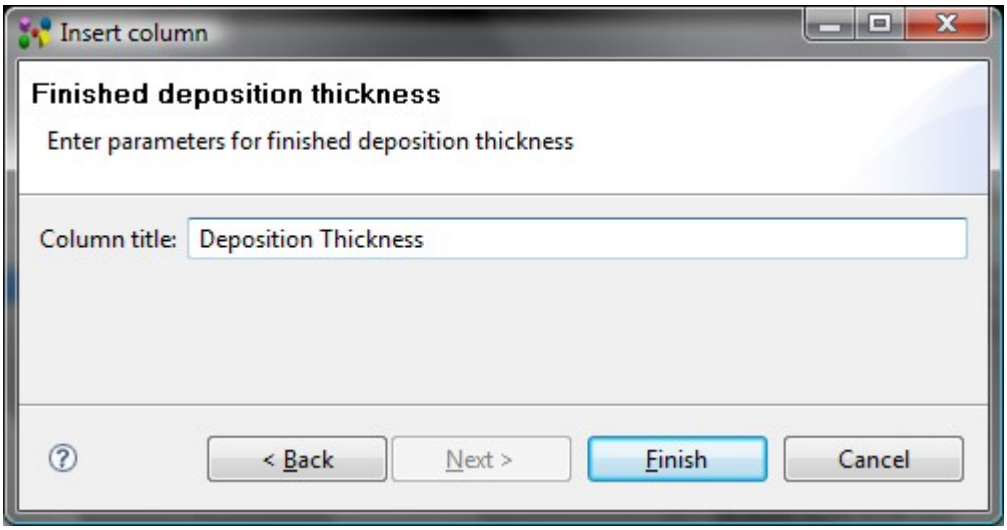

Figure 1. Insert column dialog to enter parameters for finished deposition thickness column

The column with a specified title appears in a table. To fill the column with value:

- Click with a mouse or press **F2** key on the column of type *finished deposition thickness* to enter into edit mode.
- You can change parameters of tolerance and units by pressing the button next to the edit field. The Tolerances wizard window appears, where you can select an applicable tolerance and create new units.
- Enter a number into the edit field.
- Press **Enter** key. If current units are not the same as default units, the entered value is recalculated and displayed in default units.

If there are created a few columns of the same type, all columns for the same object contain the same data.

To top

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10.6.31.6. Finished Passage Extent

The *Finished passage extent* column specifies a minimal size of the finished passage.
There are no additional parameters to enter in the wizard dialog window (see Figure 1), except for a column title. Enter a column title into the **Column title** field and click **Finish** button.

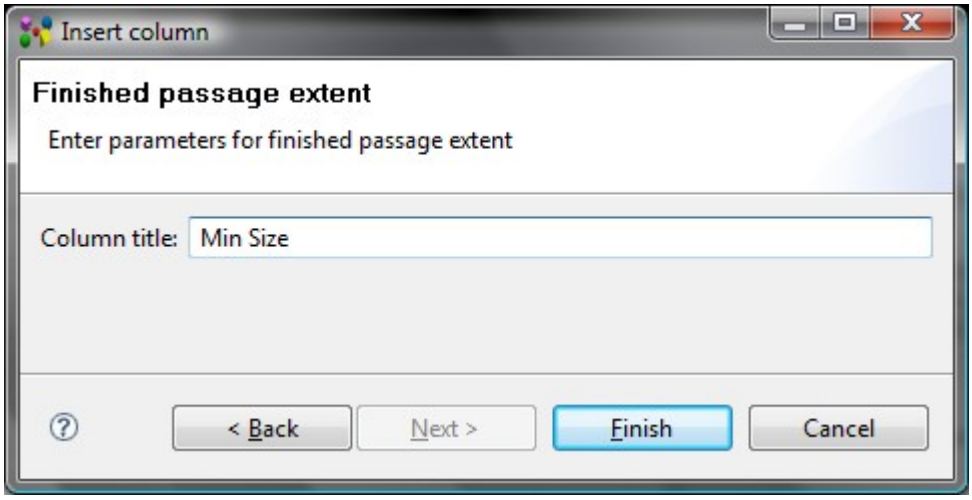

Figure 1. Insert column dialog to enter parameters for finished passage extent column

The column with a specified title appears in a table. To fill the column with value:

- Click with a mouse or press **F2** key on the column of type *finished passage extent* to enter into edit mode.
- You can change parameters of tolerance and units by pressing the button next to the edit field. The Tolerances wizard window appears, where you can select an applicable tolerance and create new units.
- Enter a number into the edit field.
- Press **Enter** key. If current units are not the same as default units, the entered value is recalculated and displayed in default units.

If there are created a few columns of the same type, all columns for the same object contain the same data.
To top

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10.6.31.7. From

The *From* column presents the current passage technology allocation to the stack model - the top most stratum the passage is passing through, if the passage orientation is top-bottom, and the bottom most stratum otherwise.
There are no additional parameters to enter in the wizard dialog page (see Figure 1), except for a column title. Enter a column title into the **Column title** field and click **Finish** button.

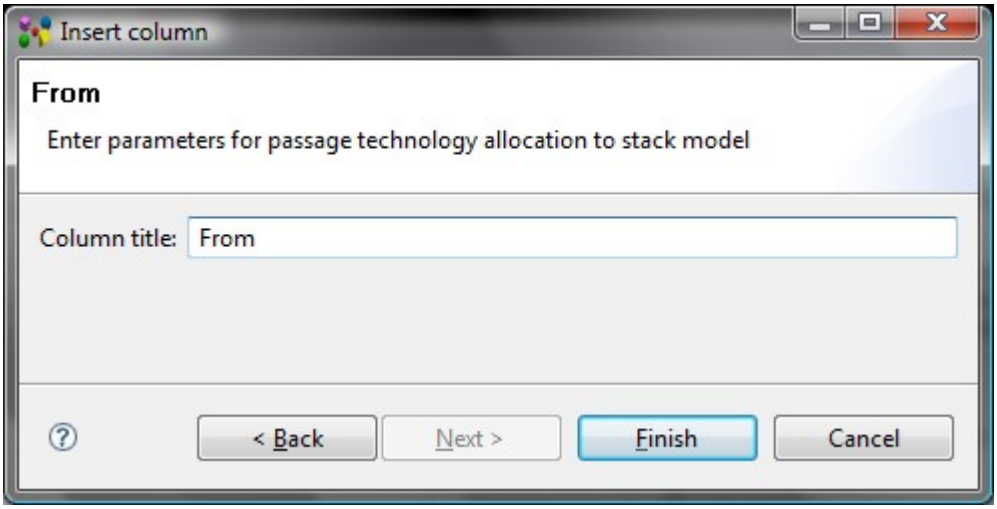

Figure 1. Insert column to enter parameters for passage technology allocation to stack model column

The column with a specified title appears in a table. To fill the column with value:

- Select an applicable layer in the **Stratum** view tree table.
- Drag the selected layer and drop on the destination cell (recently created).

If there are created a few columns of the same type, all columns for the same object contain the same data.

To top

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10.6.31.8. Laminate Stiffness Class

The *Laminate Stiffness Class* associates the classification of the laminate stiffness to the referenced object.
There are no additional parameters to enter in the wizard dialog page (see Figure 1), except a column title. Enter a column title into the **Column title** field and click **Finish** button.

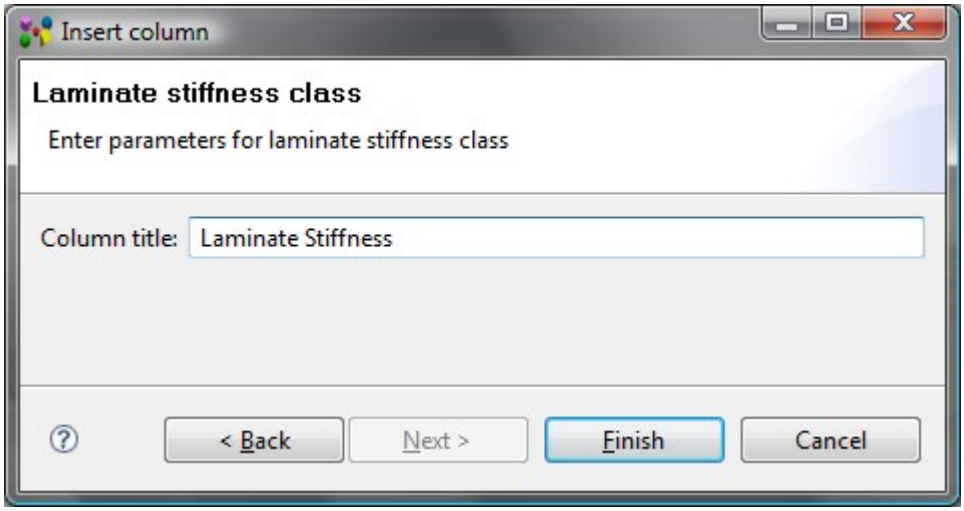

Figure 1. Insert column dialog to enter parameters for laminate stiffness class column

The column with a specified title appears in a table. To fill the column with value:

- Click with a mouse or press **F2** key on the column of type *laminate stiffness class* to enter into edit mode.

- Select an applicable value in the r/o combo box. The following values can be selected:
  - **Fluid like with constant thickness -** a laminate is a fluid like and the thickness of this laminate is constant;
  - **Fluid like with varying thickness** - a laminate is a fluid like and the thickness of this laminate is varying;
  - **Stiff laminate -** a laminate is stiff.

If there are created a few columns of the same type, all columns for the same object contain the same data.

To top

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10.6.31.9. Layer Position

The *Layer position* associates a position with a references object.
There are no additional parameters to enter in the wizard dialog page (see Figure 1), except a column title. Enter a column title into the **Column title** field and click **Finish** button.

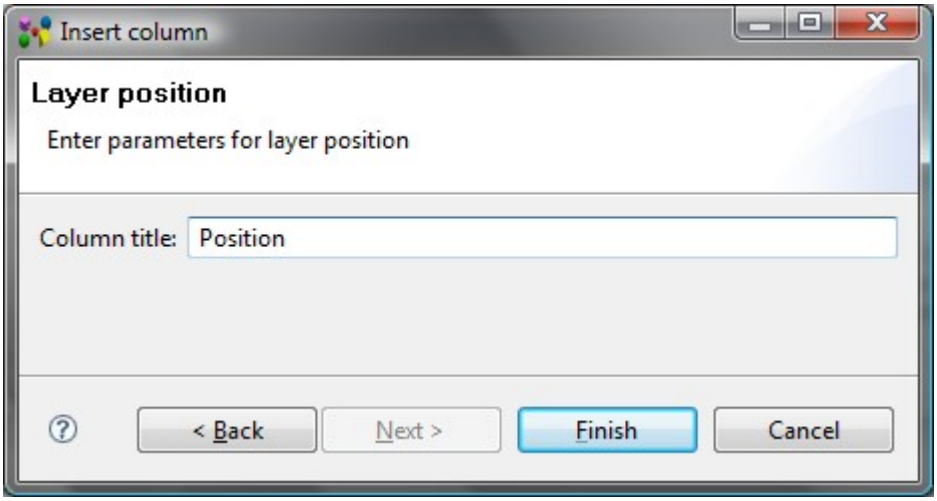

Figure 1. Insert column dialog to enter parameters for layer position column

The column with a specified title appears in a table. To fill the column with value:

- Click with a mouse or press **F2** key on the column of type *layer possition* to enter into edit mode.
- Select an applicable value in the r/o combo box. The following values can be selected:
  - **All**,
  - **Internal**,
  - **External**,
  - **Primary**,
  - **Secondary**.

If there are created a few columns of the same type, all columns for the same object contain the same data.
To top

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10.6.31.10. Maximum Aspect Ratio

The *Maximum aspect ratio* column specifies a maximum value of the aspect ratio for a particular passage.

There are no additional parameters to enter in the wizard dialog window (see Figure 1), except for a column title. Enter a column title into the **Column title** field and click **Finish** button.

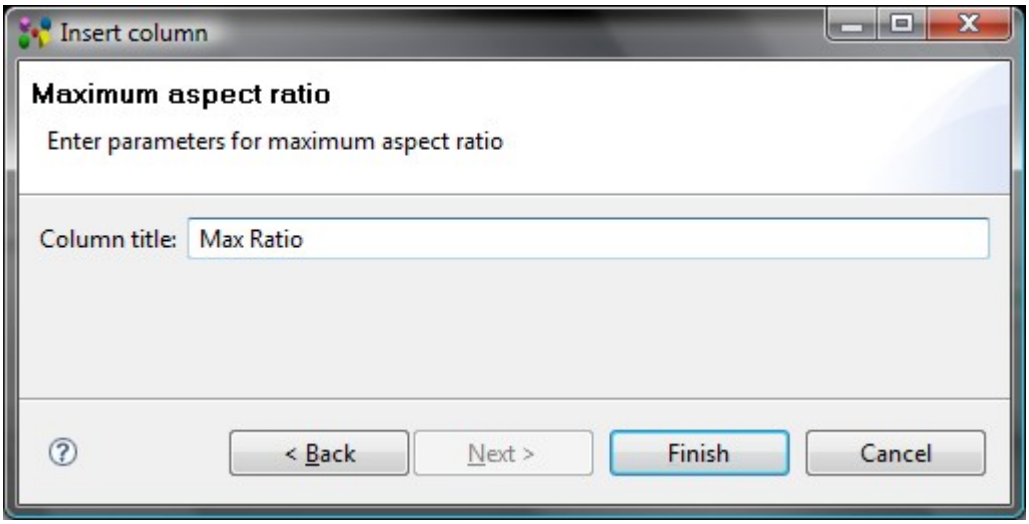

Figure 1. Insert column dialog to enter parameters for maximum aspect ratio column

The column with a specified title appears in a table. To fill the column with value:

- Click with a mouse or press **F2** key on the column of type *maximum aspect ratio* to enter into edit mode.
- Enter a number into the edit field.
- Press **Enter** key.

Parameters of the quantity, which appear in the column, are defined in the **Quantities and Units** preferences page (use main menu **Windows - > Preferences -> Numbers**).
If there are created a few columns of the same type, all columns for the same object contain the same data.

To top

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10.6.31.11. Maximum Feature Size Requirement

The *Maximum feature size requirement* associates a maximum feature size requirement with a referenced object.
There are no additional parameters to enter in the wizard dialog page (see Figure 1), except a column title. Enter a column title into the **Column title** field and click **Finish** button.

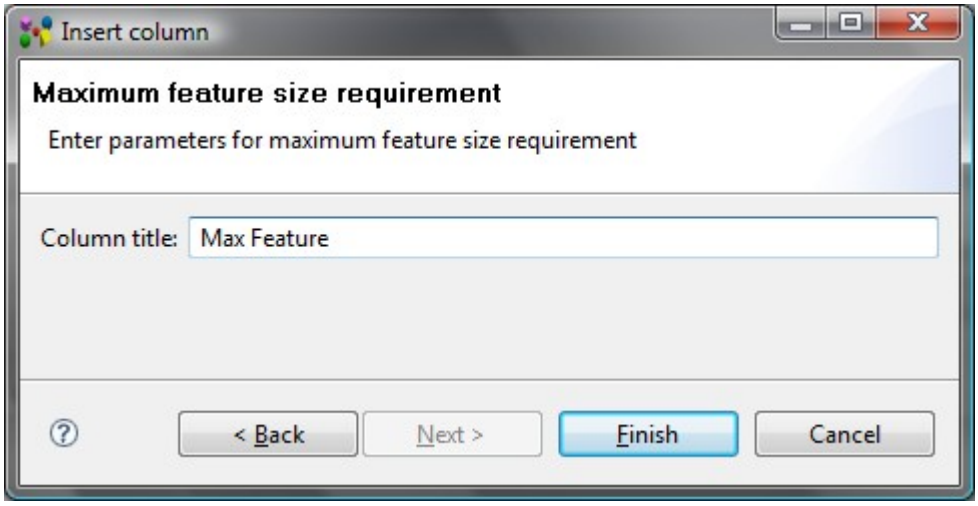

Figure 1. Insert column dialog to enter parameters for maximum feature size requirement column

The column with a specified title appears in a table. To fill the column with value:

- Click with a mouse or press **F2** key on the column of type *maximum feature size requirement* to enter into edit mode.
- You can select an applicable unit in the list next to the text field. The units, and other parameters of the quantity, which appear in the column, are defined in the **Quantities and Units** page (use main menu **Windows - > Preferences -> Numbers**).
- Enter a number into the edit field.
- Press **Enter** key. If current units are not the same as default units, the entered value is recalculated and displayed in default units.

If there are created a few columns of the same type, all columns for the same object contain the same data.

To top

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10.6.31.12. Minimum Aspect Ratio

The *Minimum aspect ratio* column specifies a minimum value of the aspect ratio of the referenced object. There are no additional parameters to enter in the wizard page (see Figure 1), except a column title. Enter a column title into the **Column title** field and click **Finish** button.

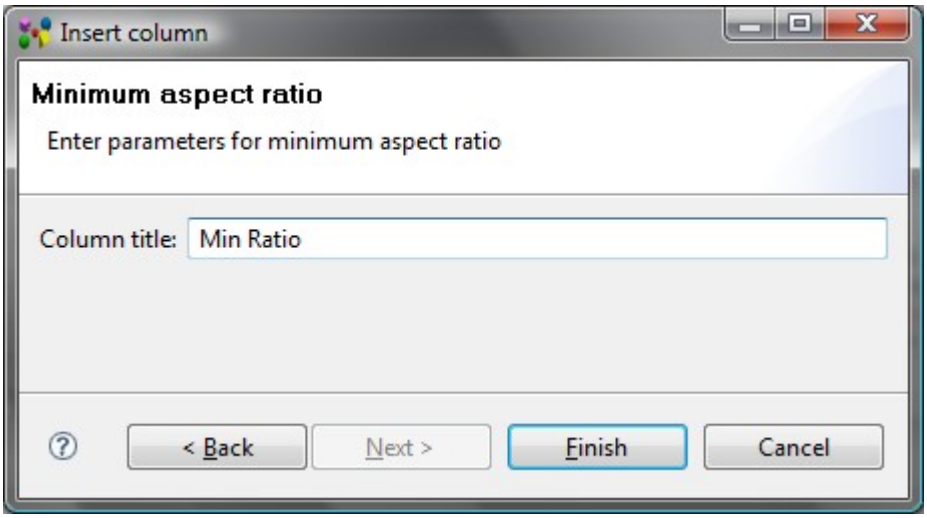

Figure 1. Insert column dialog to enter parameters for minimum aspect ratio column

The column with a specified title appears in a table. To fill the column with value:

- Click with a mouse or press **F2** key on the column of type *minimum aspect ratio* to enter into edit mode.
- Enter a numeric value into the field.
- Press **Enter** key.

Parameters of the quantity, which appear in the column, are defined in the **Quantities and Units** preferences page (use main menu **Windows - > Preferences -> Numbers**).
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To top

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10.6.31.13. Minimum Finished Feature Size

The *Minimum finished feature size* identifies a minimum finished feature size for the referenced object. There are no additional parameters to enter in the wizard dialog page (see Figure 1), except for a column title. Enter a column title into the **Column title** field and click **Finish** button.

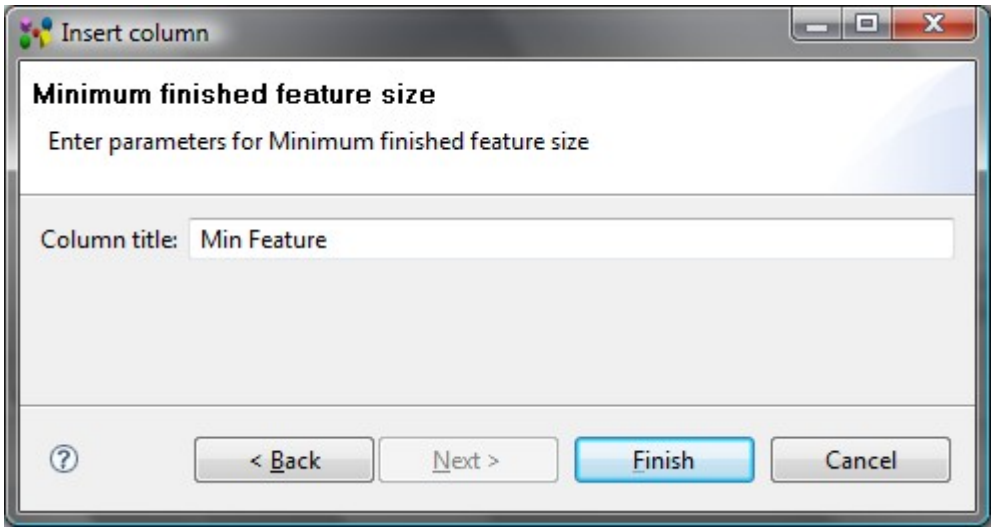

Figure 1. Insert column dialog to enter parameters for minimum finished feature size column

The column with a specified title appears in a table. To fill the column with value:

- Click with a mouse or press **F2** key on the column of type *minimum finished feature size* to enter into edit mode.
- You can select an applicable unit in the list next to the text field. The units, and other parameters of the quantity, which appear in the column, are defined in the **Quantities and Units** preferences page (use main menu **Windows - > Preferences -> Numbers**).
- Enter a number into the edit field.
- Press **Enter** key. If current units are not the same as default units, the entered value is recalculated and displayed in default units.

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To top

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10.6.31.14. Minimum Finished Feature Spacing

The *Minimum finished feature spacing* identifies a minimum finished feature spacing for the referenced object.
There are no additional parameters to enter in the wizard dialog page (see Figure 1), except a column title. Enter a column title into the **Column title** field and click **Finish** button.

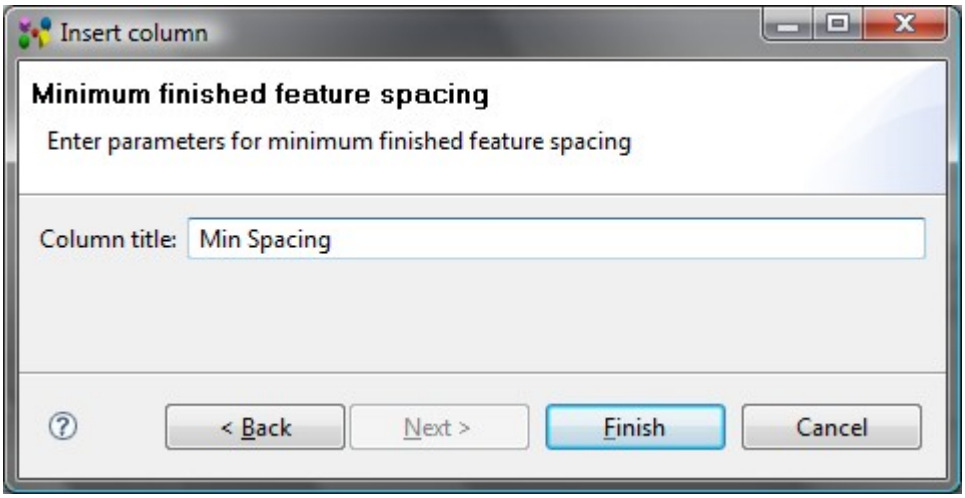

Figure 1. Insert column dialog to enter parameters for minimum finished feature spacing column

The column with a specified title appears in a table. To fill the column with value:

- Click with a mouse or press **F2** key on the column of type *minimum finished feature spacing* to enter into edit mode.
- You can select an applicable unit from the list next to the text field. The units, and other parameters of the quantity, which appear in the column, are defined in the **Quantities and Units** preferences page (use main menu **Windows - > Preferences -> Numbers**).
- Enter a number into the edit field.
- Press **Enter** key. If current units are not the same as default units, the entered value is recalculated and displayed in default units.

If there are created a few columns of the same type, all columns for the same object contain the same data.

To top

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10.6.31.15. Minimum Finished Size

The *Minimum finished size* column specifies a minimum size of a passage after all adjacent components are built in.
There are no additional parameters to enter in the wizard dialog page (see Figure 1), except for a column title. Enter a column title into the **Column title** field and click **Finish** button.

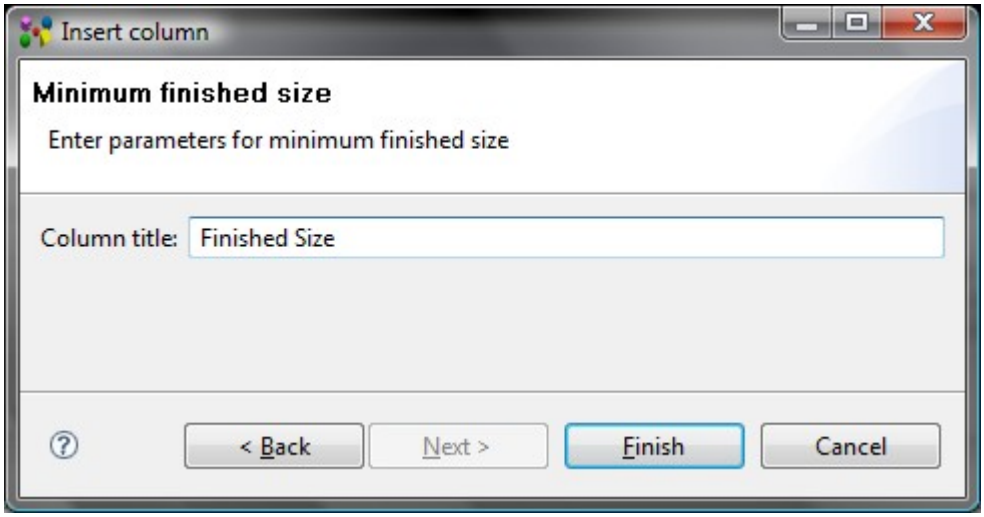

Figure 1. Insert column dialog to enter parameters for minimum finished size column

The column with a specified title appears in a table. To fill the column with value:

- Click with a mouse or press **F2** key on the column of type *minimum finished size* to enter into edit mode.
- You can select an applicable unit from the list next to the text field. The units, and other parameters of the quantity, which appear in the column, are defined in the **Quantities and Units** preferences page (use main menu **Windows - > Preferences -> Numbers**).
- Enter a number into the edit field.
- Press **Enter** key. If current units are not the same as default units, the entered value is recalculated and displayed in default units.

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To top

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10.6.31.16. Minimum Interior Extent

The *Minimum interior extent* column specifies the minimal size of the interior for a passage.
This characteristic is available only for a passage, which technology type is *via*. There are no additional parameters to enter in the wizard dialog page (see Figure 1), except for a column title. Enter a column title into the **Column title** field and click **Finish** button.

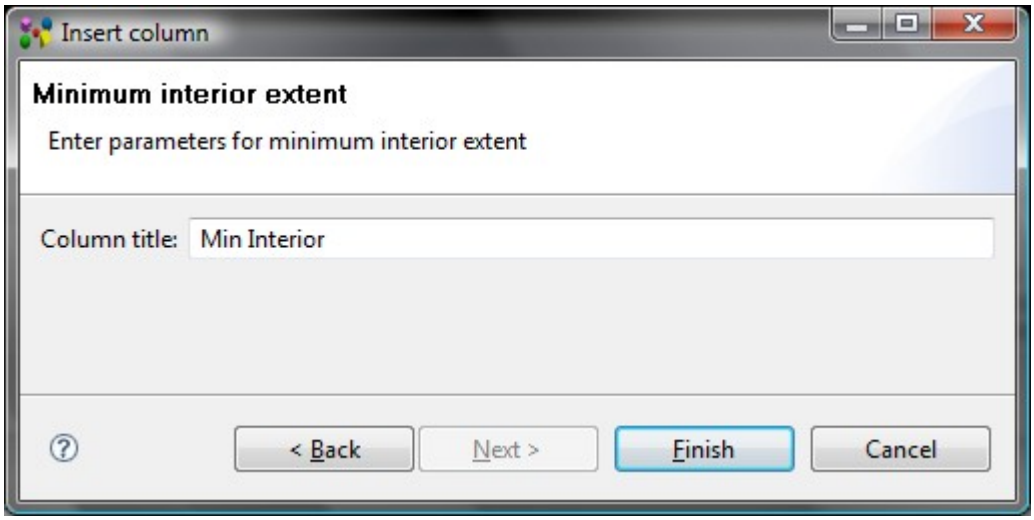

Figure 1. Insert column dialog to enter parameters for minimum interior extent column

The column with a specified title appears in a table. To fill the column with value:

- Click with a mouse or press **F2** key on the column of type *minimum interior extent* to enter into edit mode.
- You can change parameters of tolerance and units by pressing the button next to the edit field. The Tolerances wizard window appears, where you can select an applicable tolerance and create new units.
- Enter a number into the edit field.
- Press **Enter** key. If current units are not the same as default units, the entered value is recalculated and displayed in default units.

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To top

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10.6.31.17. Passage Deposition Material Attribute

The *Passage deposition material attribute* is a characteristic that depends on the deposition material aspects.
Enter parameters for a *passage deposition material attribute* column (see Figure 1):

- Enter a title for the column into the **Column title** field.
- Select a conductivity classification to be associated with a deposition material from the **Attribute** layout by selecting the particular radio button:
    - **Electric conductivity -** is a measure of a material's ability to conduct an electric current;
    - **Thermal conductivity -** is the property of a material that indicates its ability to conduct heat;
    - **Magnetic permeability** - is the degree of magnetization of a material that responds linearly to an applied magnetic field;
    - **Optical insertion loss -** is the loss of signal power resulting from the insertion of a device in an optical fiber;
    - **Dielectric permittivity -** is a physical quantity that describes how an electric field affects, and is affected by a dielectric medium, and is determined by the ability of a material to polarize in response to the field, and thereby reduce the total electric field inside the material.
- Press **Finish** button to create a column.

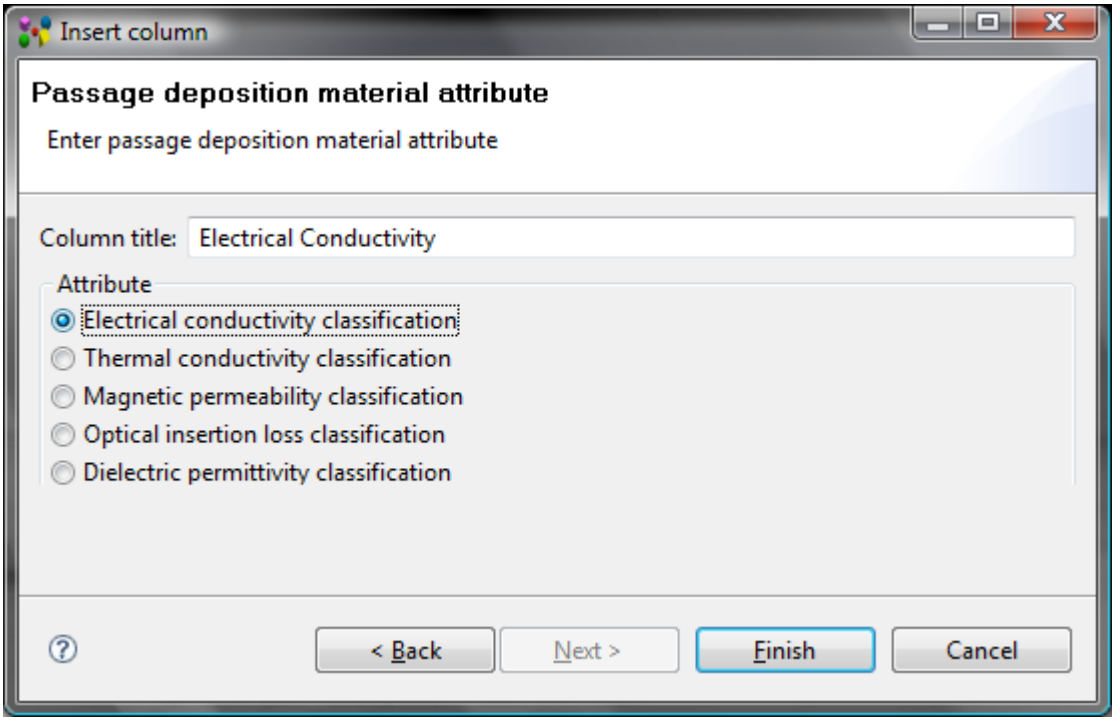

Figure 1. Insert column dialog to enter parameters for passage deposition material attribute column

The column with a specified title appears in a table. To fill the column with value:

- Click with a mouse or press **F2** key on the column of type *passage deposition material attribute* to enter into edit mode.
- Select an applicable value from the combo box. The values in the list depend on the conductivity classification, selected in the wizard window:
    - *Electrical conductivity classification* column contains the predefined values: conductive, non conductive, resistive, semi conductive and super conductive.
    - *Thermal conductivity classification* column contains the predefined values: conductive, non conductive, resistive, semi conductive and super conductive.
    - *Magnetic permeability classification* column contains the predefined values: free space permeability, low permeability, medium permeability and highly permeable.
    - *Optical insertion loss classification* column contains the predefined values: vacuum, very low loss, low loss, medium loss and high loss.
    - *Dielectric permittivity classification* column contains the predefined values: vacuum permittivity, low permittivity, medium permittivity and high permittivity.
- Press **Enter** key.

If there are created a few columns of the same type, all columns for the same object contain the same data.

To top

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10.6.31.18. Passage Deposition Material Identification

The *Passage deposition material identification* column specifies a deposition material of a passage. There are no additional parameters to enter in the wizard dialog page (see Figure 1), except for a column title. Enter a column title into the **Column title** field and click **Finish** button.

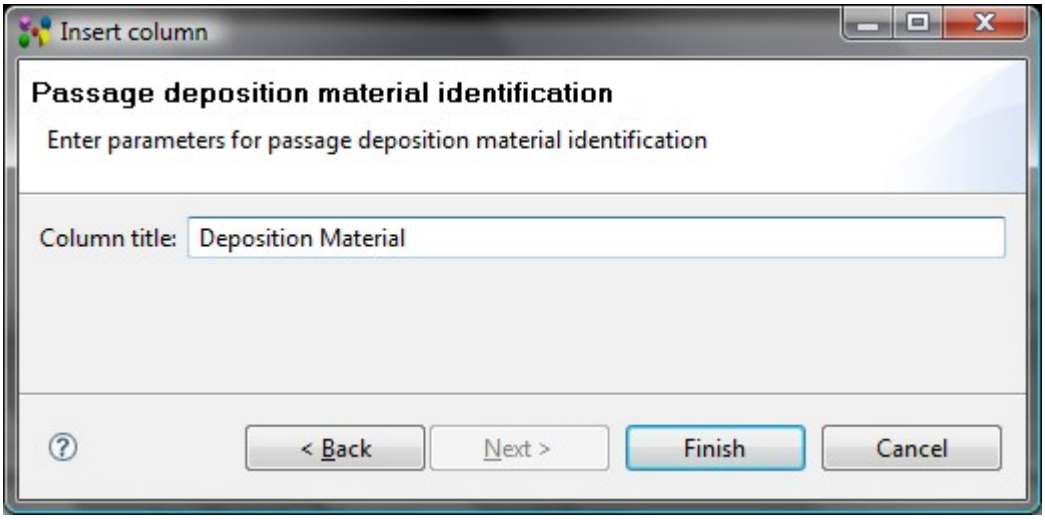

Figure 1. Insert column dialog to enter parameters for passage deposition material identification column

The column with a specified title appears in a table. To fill the column with value:

- Click with a mouse or press **F2** key on the column of type *passage deposition material identification* to enter into edit mode.
- The 'CUSTOM' material identification appears in the editable combo box. You can select another material identification in the list, if present, or create a new one by entering an identification into the field.
- Press **Enter** key.

If there are created a few columns of the same type, all columns for the same object contain the same data.

To top

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10.6.31.19. Passage Filling Material Attribute

The *Passage filling material attribute* is a characteristic that depends on the filling material aspects. Enter parameters for a *passage filling material attribute* column (see Figure 1):

- Enter a title for the column into the **Column title** field.
- Select a conductivity classification to be associated with a filling material from the **Attribute** layout by selecting the particular radio button:
  - **Electric conductivity -** is a measure of a material's ability to conduct an electric current;
  - **Thermal conductivity -** is the property of a material that indicates its ability to conduct heat;
  - **Magnetic permeability** - is the degree of magnetization of a material that responds linearly to an applied magnetic field;
  - **Optical insertion loss -** is the loss of signal power resulting from the insertion of a device in an optical fiber;
  - **Dielectric permittivity -** is a physical quantity that describes how an electric field affects, and is affected by a dielectric medium, and is determined by the ability of a material to polarize in response to the field, and thereby reduce the total electric field inside the material.
- Press **Finish** button to create a column.

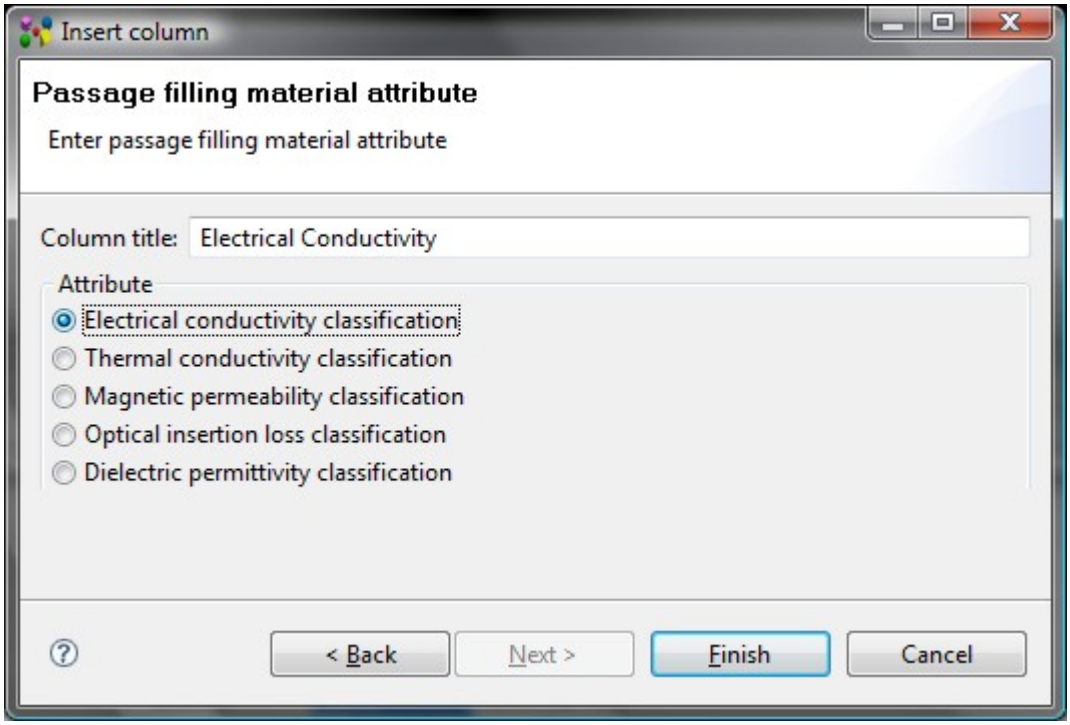

Figure 1. Insert column dialog to enter parameters for passage filling material attribute column

The column with a specified title appears in a table. To fill the column with value:

- Click with a mouse or press **F2** key on the column of type *passage filling material attribute* to enter into edit mode.
- Select an applicable value from the combo box. The values in the list depend on the conductivity classification, selected in the wizard window:
    - *Electrical conductivity classification* column contains the predefined values: conductive, non conductive, resistive, semi conductive and super conductive.
    - *Thermal conductivity classification* column contains the predefined values: conductive, non conductive, resistive, semi conductive and super conductive.
    - *Magnetic permeability classification* column contains the predefined values: free space permeability, low permeability, medium permeability and highly permeable.
    - *Optical insertion loss classification* column contains the predefined values: vacuum, very low loss, low loss, medium loss and high loss.
    - *Dielectric permittivity classification* column contains the predefined values: vacuum permittivity, low permittivity, medium permittivity and high permittivity.
- Press **Enter** key.

If there are created a few columns of the same type, all columns for the same object contain the same data.
To top

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10.6.31.20. Passage Filling Material Identification

The *Passage filling material identification* column specifies a deposition material of a passage.
There are no additional parameters to enter in the wizard dialog page (see Figure 1), except for a column title. Enter a column title into the **Column title** field and click **Finish** button.

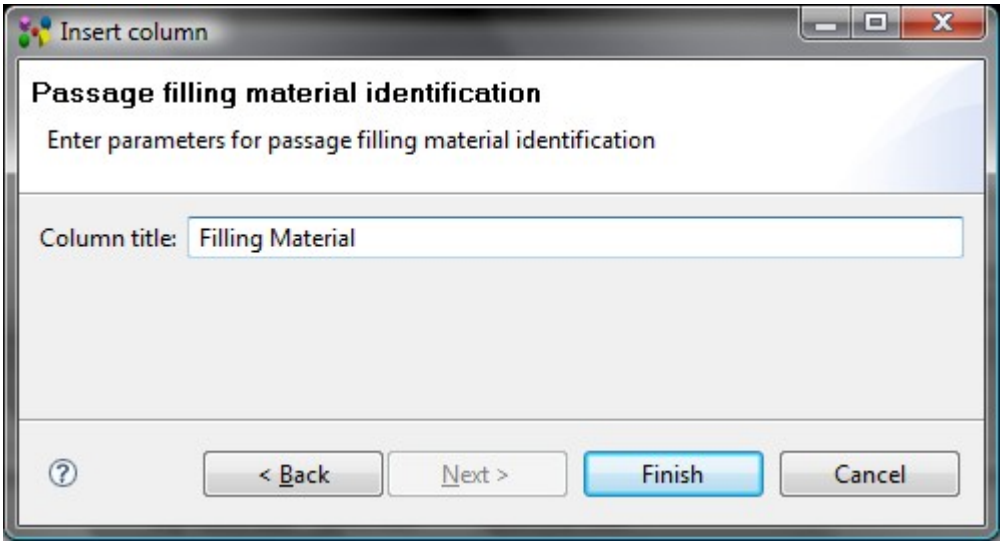

Figure 1. Insert column dialog to enter parameters for passage filling material identification column

The column with a specified title appears in a table. To fill the column with value:

- Click with a mouse or press **F2** key on the column of type *passage filling material identification* to enter into edit mode.
- The 'CUSTOM' material identification appears in the editable combo box. You can select another material identification in the list, if present, or create a new one by entering an identification into the field.
- Press **Enter** key.

If there are created a few columns of the same type, all columns for the same object contain the same data.

To top

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10.6.31.21. Passage Terminus Condition

The *Passage terminus condition* column specifies a terminus condition of a passage.
There are no additional parameters to enter in the wizard dialog page (see Figure 1), except for a column title. Enter a column title into the **Column title** field and click **Finish** button.

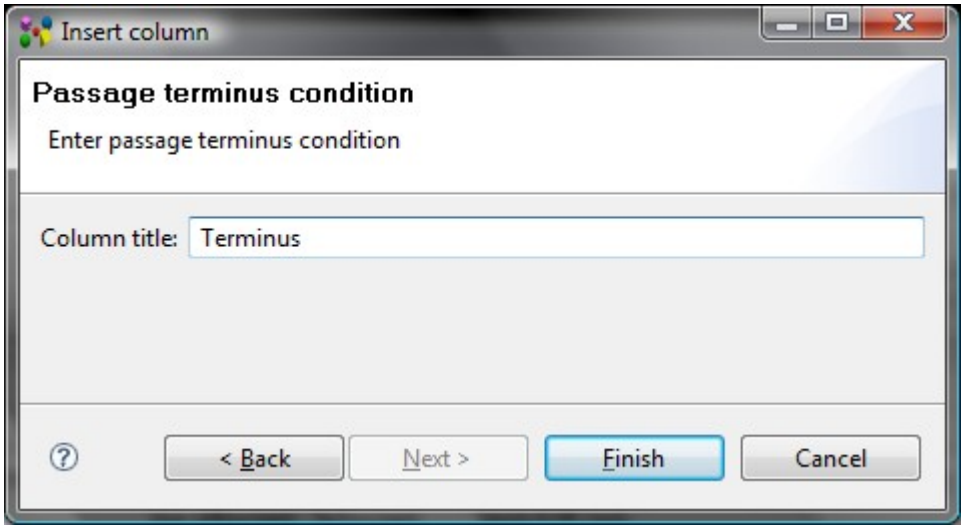

Figure 1. Insert column dialog to enter parameters for passage terminus condition column

The column with a specified title appears in a table. To fill the column with value:

- Click with a mouse or press **F2** key on the column of type *passage terminus condition* to enter into edit mode.
- Select an applicable value in the r/o combo box. The following values can be selected:
  - **Bilateral bond**;
  - **Bilateral complete removal**;
  - **Unilateral bond**.
- Press **Enter** key.

If there are created a few columns of the same type, all columns for the same object contain the same data.

To top

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10.6.31.22. Plated Passage

The *Plated passage* column specifies the plating feature of a passage (whether passage is plated or not). There are no additional parameters to enter in the wizard dialog page (see Figure 1), except for a column title. Enter a column title into the **Column title** field and click **Finish** button.

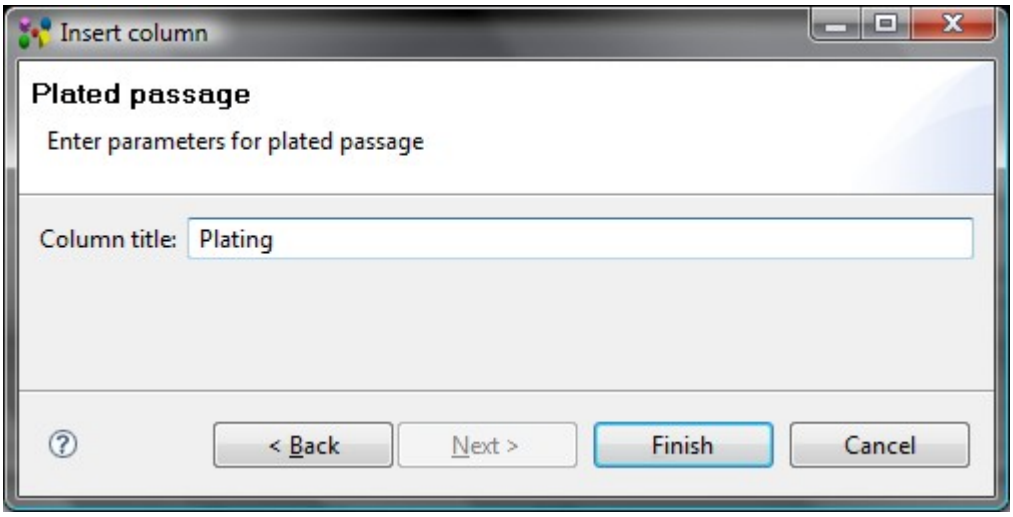

Figure 1. Insert column dialog to enter parameters for plated passage column

The column with a specified title appears in a table. To fill the column with value:

- Click with a mouse or press **F2** key on the column of type *plated passage* to set one of the available values:
  - **Plated**,
  - **Unplated**.
- Press **Enter** key.

If there are created a few columns of the same type, all columns for the same object contain the same data.

To top

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10.6.31.23. Stratum Purpose

The *Stratum purpose* identifies the purpose of the stratum for the references object.
There are no additional parameters to enter in the wizard dialog page (see Figure 1), except a column title.
Enter a column title into the **Column title** field and click **Finish** button.

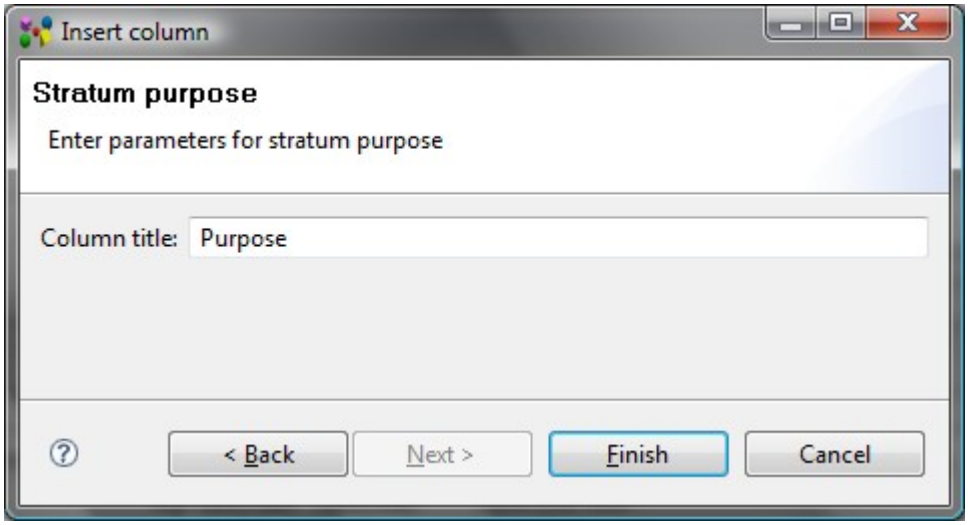

Figure 1. Insert column to enter parameters for stratum purpose column

The column with a specified title appears in a table. To fill the column with value:

- Click with a mouse or press **F2** key on the column of type *stratum purpose* to enter into edit mode.
- Select one of the available values in the read only combo box:
  - **Build-up material**,
  - **Coating**,
  - **Core material**,
  - **Embedded passive capacitor dielectric**,
  - **Embedded passive resistor**,
  - **Generic layer**,
  - **Glue**,
  - **Gluemask**,
  - **Lands only**,
  - **Pastemask**,
  - **Power or ground**,
  - **Signal**,
  - **Silkscreen**,
  - **Soldermask**,
  - **Solderpaste**.
- Press **Enter** key.

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To top

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10.6.31.24. Stratum Technology Material Designation

The *Stratum technology material designation* column specifies a text, that is used in a stratum technology to label a material.
There are no additional parameters to enter in the wizard dialog page (see Figure 1), except a column title. Enter a column title into the **Column title** field and click **Finish** button.

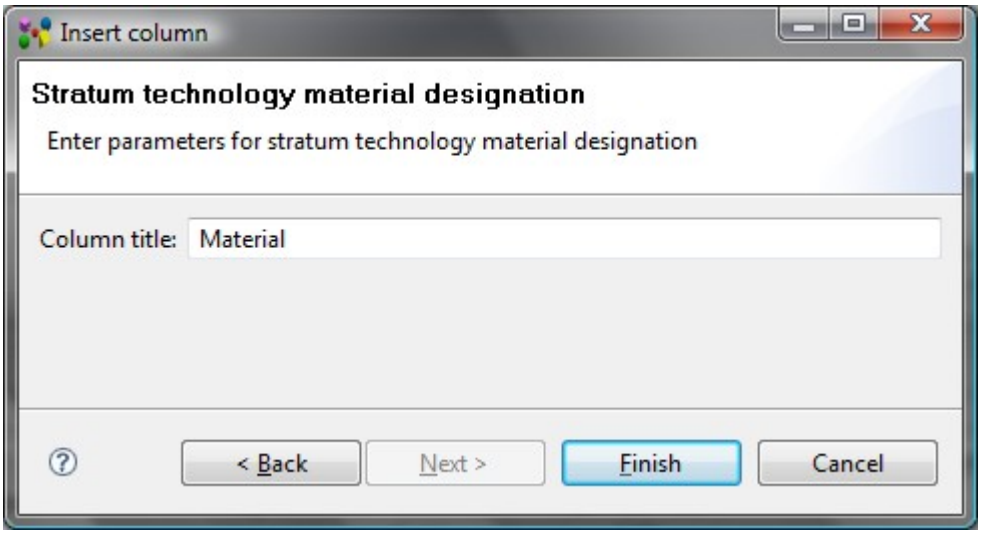

Figure 1. Insert column dialog to enter parameters for stratum technology material designation column

The column with a specified title appears in a table. To fill the column with value:

- Click with a mouse or press **F2** key on the column of type *stratum technology material designation* to enter into edit mode.
- Enter an applicable material designation text into the edit field.
- Press **Enter** key.

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To top

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10.6.31.25. Stratum Technology Occurrence Base

The *Stratum Technology Occurrence Base* column presents the base value (down, independent or up) of a stratum technology occurrence. One stratum can be based on another stratum. For example stratum A can be based on stratum B, but stratum B can not be based on stratum A at the same time.
There are no additional parameters to enter in the wizard dialog page (see Figure 1), except for a column title. Enter a column title into the **Column title** field and click **Finish** button.

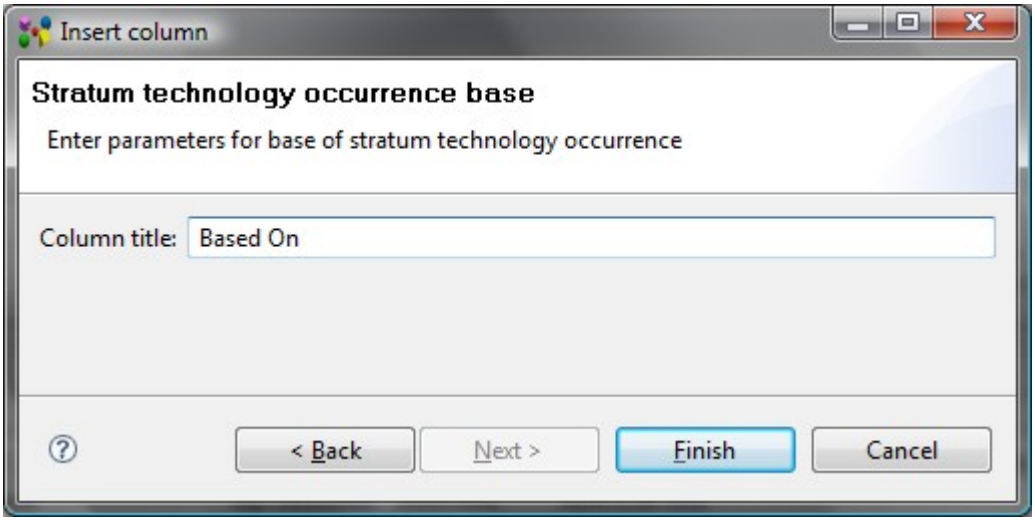

Figure 1. Insert column dialog to enter parameters for base of stratum technology occurrence column

The column with a specified title appears in a table. To fill the column with value:

- Click with a mouse or press **F2** key on the column of type *stratum technology occurrence base* to enter into edit mode.
- Select an applicable value in the r/o combo box. The following values can be selected:
  - **Down**;
  - **Independent**;
  - **Up**.
- Press **Enter** key.

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To top

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10.6.31.26. Stratum Technology Occurrence Definition

The *Startum technology occurrence definition* presents an explanation of the stratum technology occurrence.
There are no additional parameters to enter in the wizard dialog page (see Figure 1), except for a column title. Enter a column title into the **Column title** field and click **Finish** button.

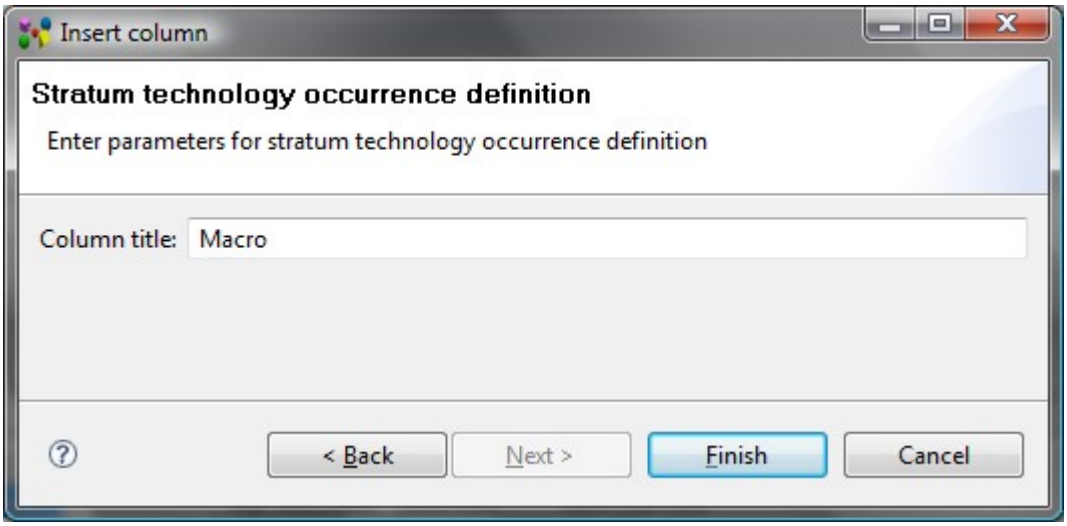

Figure 1. Insert column dialog to enter parameters for stratum technology occurrence column

The column with a specified title appears in a table. To fill the column with value:

- Click with a mouse or press **F2** key on the column of type *stratum technology occurrence definition* to enter into edit mode.
- The 'CUSTOM' value appears in the editable combo box. You can select another stratum technology occurrence definition in the list, if present, or create a new one by entering a definition into the field.
- Press **Enter** key.

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To top

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10.6.31.27. Stratum Technology Occurrence Number

The *Stratum technology occurrence number* presents the number of conductive layers.
There are no additional parameters to enter in the wizard dialog page (see Figure 1), except for a column title. Enter a column title into the **Column title** field and click **Finish** button.

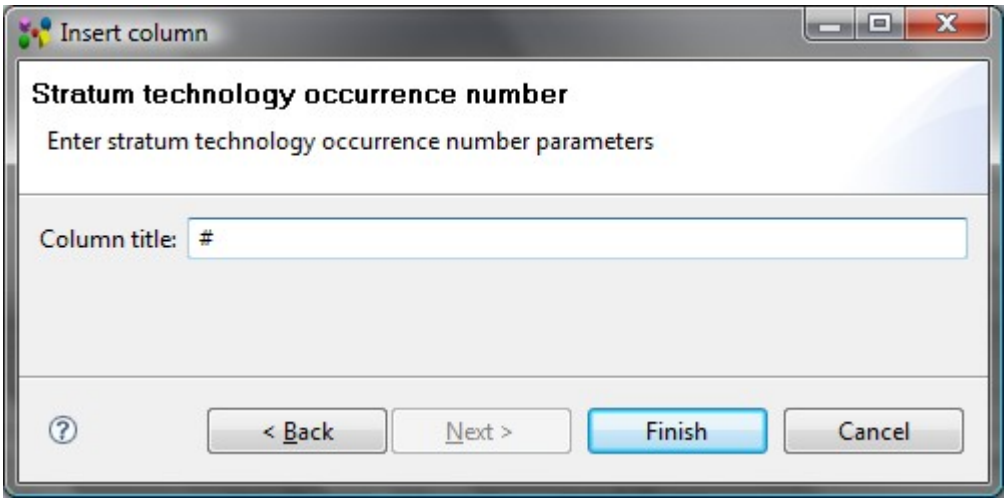

Figure 1. Insert column dialog to enter parameters for stratum technology occurrence number column

The column with a specified title appears in a table. You can not enter or edit a value in the column. The value is derivable.
If there are created a few columns of the same type, all columns for the same object contain the same data.

To top

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10.6.31.28. Stratum Thickness

The *Stratum thickness* column specifies a thickness of a stratum in a stratum technology.

There are no additional parameters to enter in the wizard dialog page (see Figure 1), except for a column title. Enter a column title into the **Column title** field and click **Finish** button.

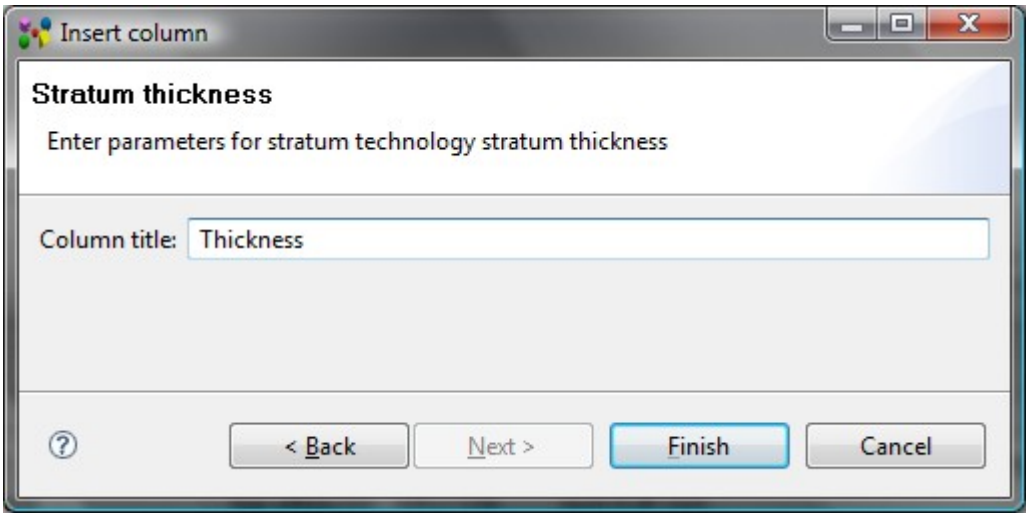

Figure 1. Insert column dialog to enter parameters for stratum technology stratum thickness column

The column with a specified title appears in a table. To fill the column with value:

- Click with a mouse or press **F2** key on the column of type *stratum thickness* to enter into edit mode.
- You can change parameters of tolerance and units by pressing the button next to the edit field. The Tolerances wizard window appears, where you can select an applicable tolerance and create new units.
- Enter a number into the edit field.
- Press **Enter** key. If current units are not the same as default units, the entered value is recalculated and displayed in default units.

If there are created a few columns of the same type, all columns for the same object contain the same data.

To top

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10.6.31.29. Technology Type

The *Technology type* column specifies a type of technology for a referenced object.
There are no additional parameters to enter in the wizard dialog page (see Figure 1), except for a column title. Enter a column title into the **Column title** field and click **Finish** button.

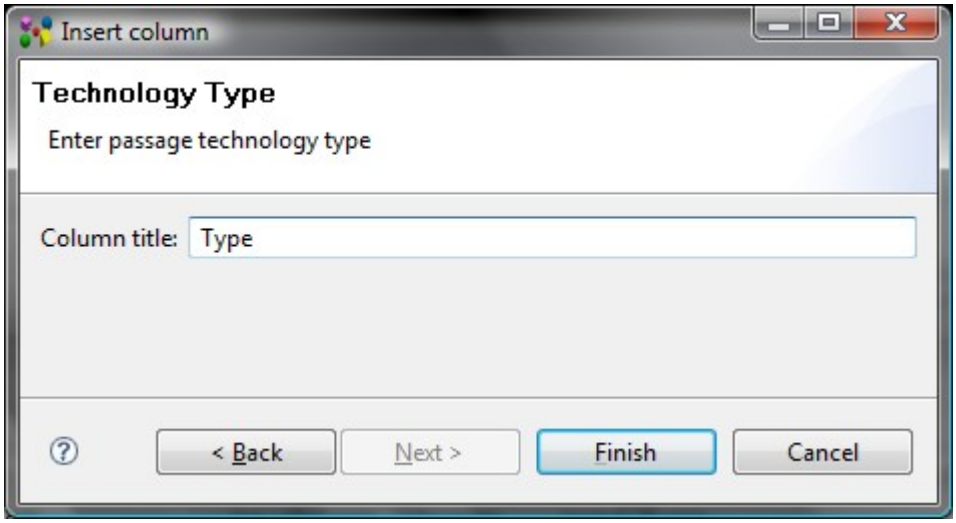

Figure 1. Insert column dialog to enter parameters for technology type column

The column with a specified title appears in a table and automatically filled with a particular value, which is associated to the referenced object. You can edit the column value:

- Click with a mouse or press **F2** key on the column of type *technology type* to set one of the available values:
  - **Border**,
  - **Component termination**,
  - **Unsupported**,
  - **Via**.
- Press **Enter** key.

If there are created a few columns of the same type, all columns for the same object contain the same data.

To top

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10.6.31.30. To

The *To* column presents the current passage technology allocation to the stack model - the top most stratum the passage is passing through, if the passage orientation is bottom-top, and the bottom most stratum otherwise.
There are no additional parameters to enter in the wizard dialog page (see Figure 1), except for a column title. Enter a column title into the **Column title** field and click **Finish** button.

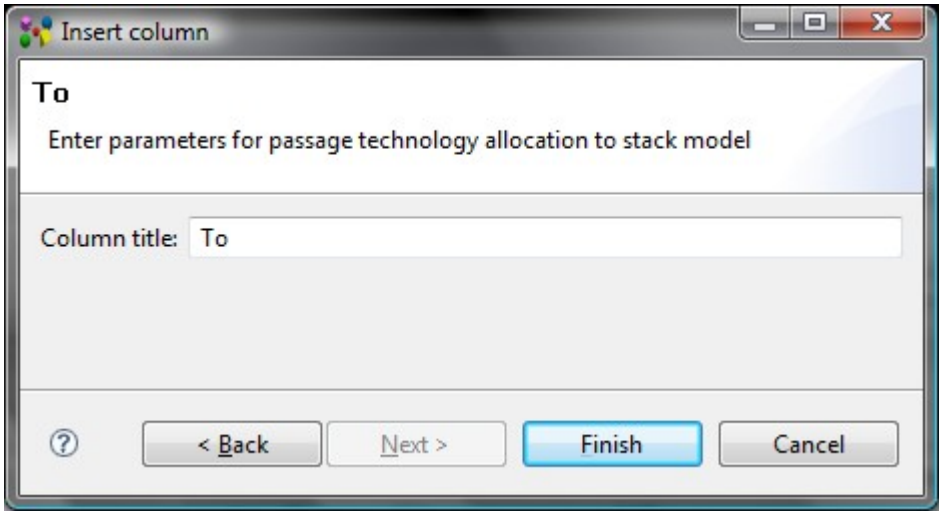

Figure 1. Insert column dialog to enter parameters for passage technology allocation to stack model column

The column with a specified title appears in a table. To fill the column with value:

- Select an applicable layer in the **Stratum** view tree table.
- Drag the selected layer and drop on the destination cell (recently created).

If there are created a few columns of the same type, all columns for the same object contain the same data.

To top

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10.6.31.31. Total Thickness

The *Total thickness* presents the summation of the set of layer thicknesses in the referenced object. There are no additional parameters to enter in the wizard dialog page (see Figure 1), except for a column title. Enter a column title into the **Column title** field and click **Finish** button.

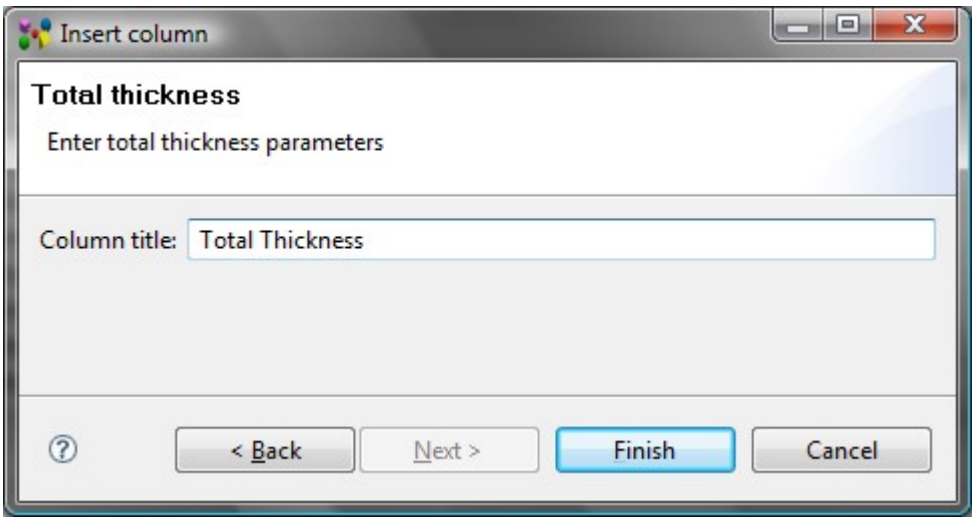

Figure 1. Insert column dialog to enter parameters for total thickness column

The column with a specified title appears in a table. You can not enter or edit a value in the column. The value is derivable. It summarizes the thickness of all layers, which are included in the referenced object. If there are created a few columns of the same type, all columns for the same object contain the same

data.

To top

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10.6.31.32. Unconnected Minimum Annular Ring

The *Unconnected minimum annular ring* column specifies the size of the unconnected minimum pad area that remains after a hole is drilled through the pad.
There are no additional parameters to enter in the wizard dialog page (see Figure 1), except for a column title. Enter a column title into the **Column title** field and click **Finish** button.

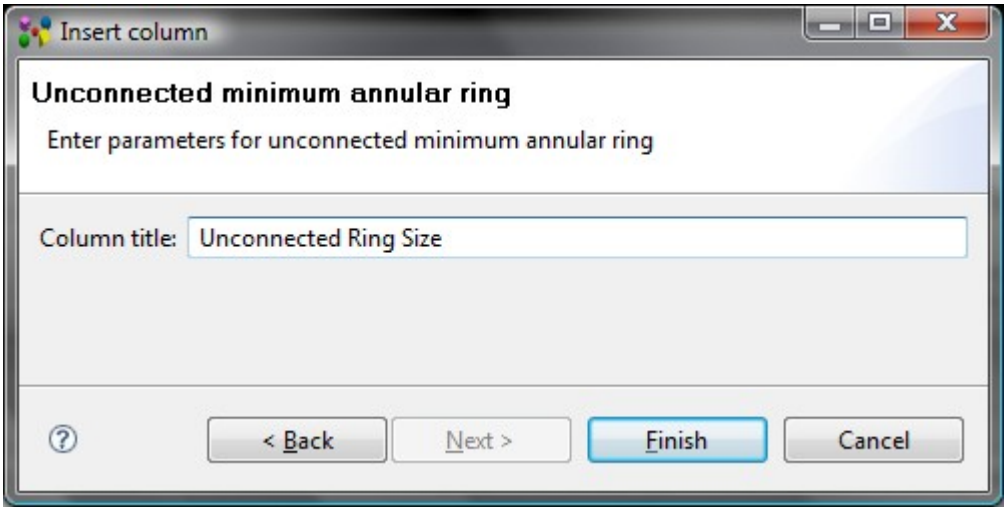

Figure 1. Insert column dialog to add parameters for unconnected minimum annular ring column

The column with a specified title appears in a table. To fill the column with value:

- Click with a mouse or press **F2** key on the column of type *unconnected minimum annular ring* to enter into edit mode.
- You can select an applicable unit from the list next to the text field. The units, and other parameters of the quantity, which appear in the column, are defined in the **Quantities and Units** preferences page (use main menu **Windows - > Preferences -> Numbers**).
- Enter a number into the edit field.
- Press **Enter** key. If current units are not the same as default units, the entered value is recalculated and displayed in default units.

If there are created a few columns of the same type, all columns for the same object contain the same data.

To top

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10.7. Columns Perspective

A perspective defines the initial set of columns in a table of a particular view. Each view has default perspective of columns. The user can create, select the created one or reset to default perspective.

To top

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10.7.1. Open Columns Perspective

Follow the routine to open other than default columns perspective (the perspective must be already created):

- Right click on the table in the current view. A context sensitive menu appears.
- Select **Columns Perspective -> <perspective_name>** menu item (perspective_name - the perspective created by the user) (see Figure 1).

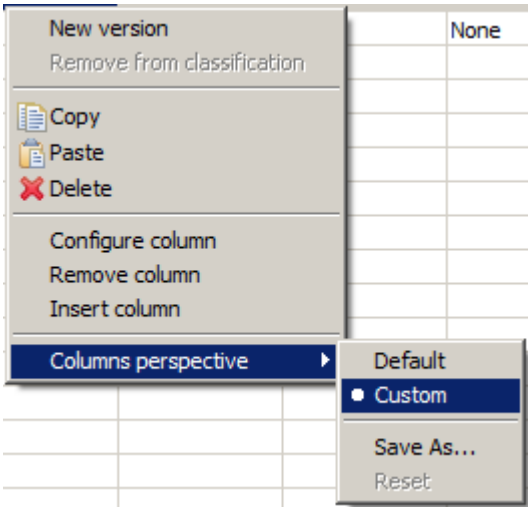

Figure 1. Selected 'Custom' perspective by using context sensitive menu

- The selected columns perspective is applied for the current table.

To top

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10.7.2. Create Columns Perspective

Follow the routine to create a new columns perspective:

- Open a view with a table.
- Add some new columns to the table.
- Right click on any table row. A context sensitive menu appears.
- Select **Columns Perspective -> Save as** menu item (see Figure 1).

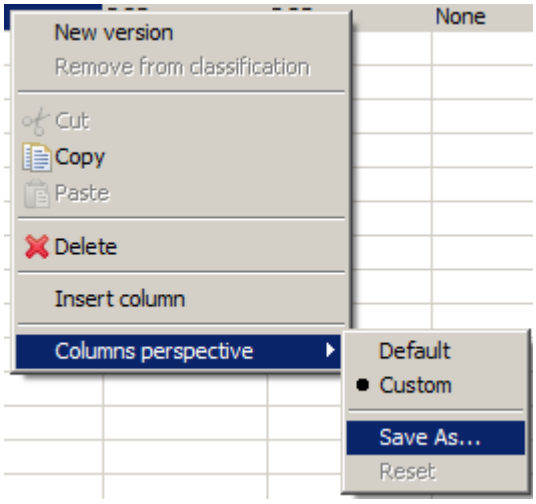

Figure 1. Selected Save as menu item

- The **Save Columns Perspective As** dialog window appears on the screen (see Figure 2):

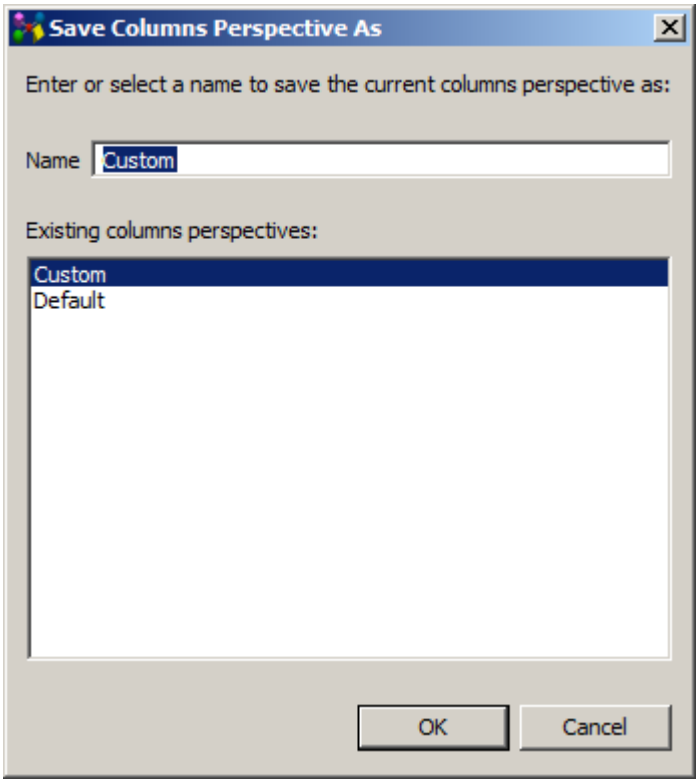

Figure 2. Save Columns Perspective As dialog window

- Enter a name of the columns perspective into the **Name** field .
- Click **Ok** button. The created perspective appears in the context sensitive menu list, marked with the bullet.

Or

- Reuse already created perspective name by selecting one from the **Existing columns perspectives** list:
    - The dialog window appears to confirm, whether you want to overwrite the perspective name or not (see Figure 3).

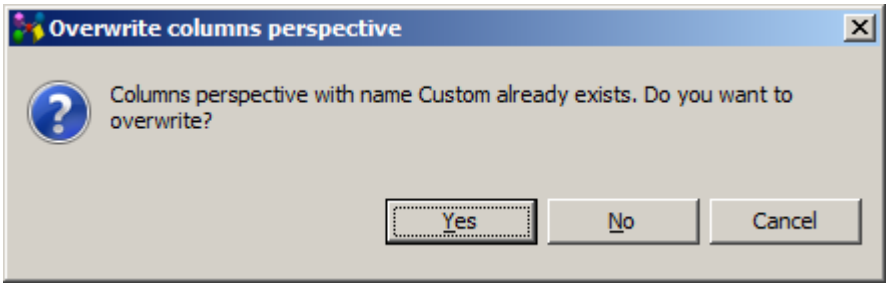

Figure 3. Overwrite columns perspective dialog window

- Click **Yes** button to overwrite the perspective name. The created perspective appears in the context sensitive menu list, marked with the bullet.
  Or
- Click **No** button to go back to the **Save Columns Perspective As** dialog window.
  Or
- Click **Cancel** button to quit the creating of a new columns perspective.

To top

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10.7.3. Rearrange Columns

Follow the routine to rearrange dynamic columns in a columns perspective:

- Right click on a table in a particular view.
- Select the **Columns perspective -> Rearrange** menu item in the appeared context sensitive menu. The **Rearrangement** dialog window appears on the screen (see Figure 1):

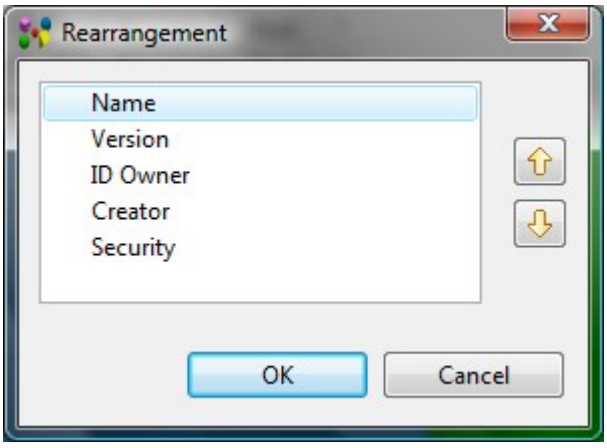

Figure 1. Rearrangement dialog window

- Select a name of the column you want to change the order in the table.
- Use the arrow buttons *up* or *down* on the right of the window to move the selected column up or down the list.
- Press **OK** button to confirm your changes or **Cancel** button to cancel the rearrangement.

To top

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10.7.4. Reset Default Columns Perspective

Follow the routine to reset the default columns perspective if you have made any changes to it:

- Right click on any table row. A context sensitive menu appears.
- Select the **Columns Perspective -> Reset** menu item.
- The initial defalt columns perspective layout appears on the screen.

Note. You can reset only the default columns perspective, for other columns perspectives the **Reset** menu item is disabled.
To top

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11. Printing

IDA-STEP provides possibility to print an active view or selected views, which are currently opened in IDA-STEP. You can select an applicable printing option in the main menu bar by selecting:

- **File ->** Print - this option is enabled if a particular view is activated in IDA-STEP window;
- **File ->** Print views.

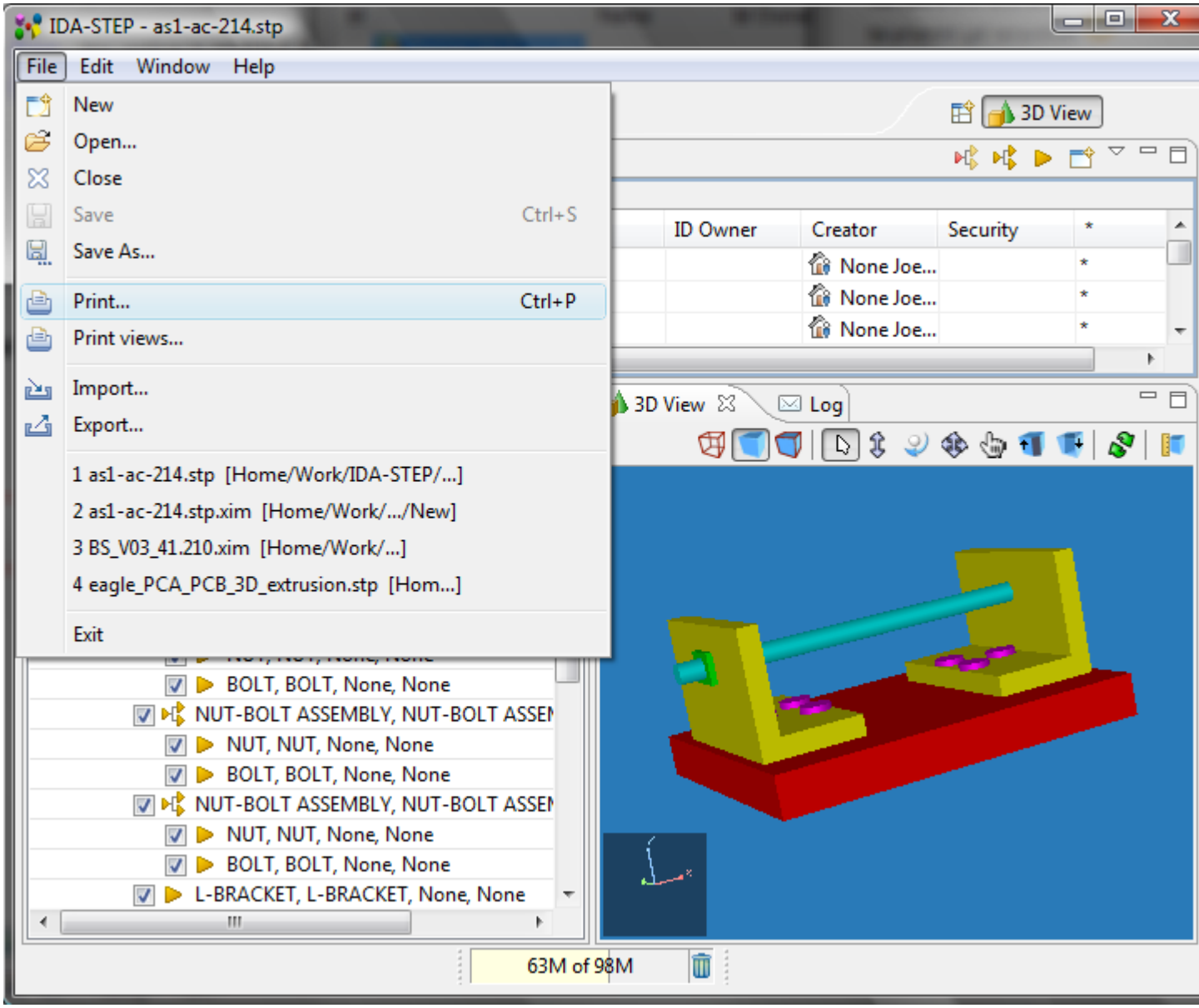

Figure 1. IDA-STEP window, File menu items.

Print

After the **Print** menu item is selected, only the currently activated view appears in the **Print preview** window (see Figure 2):

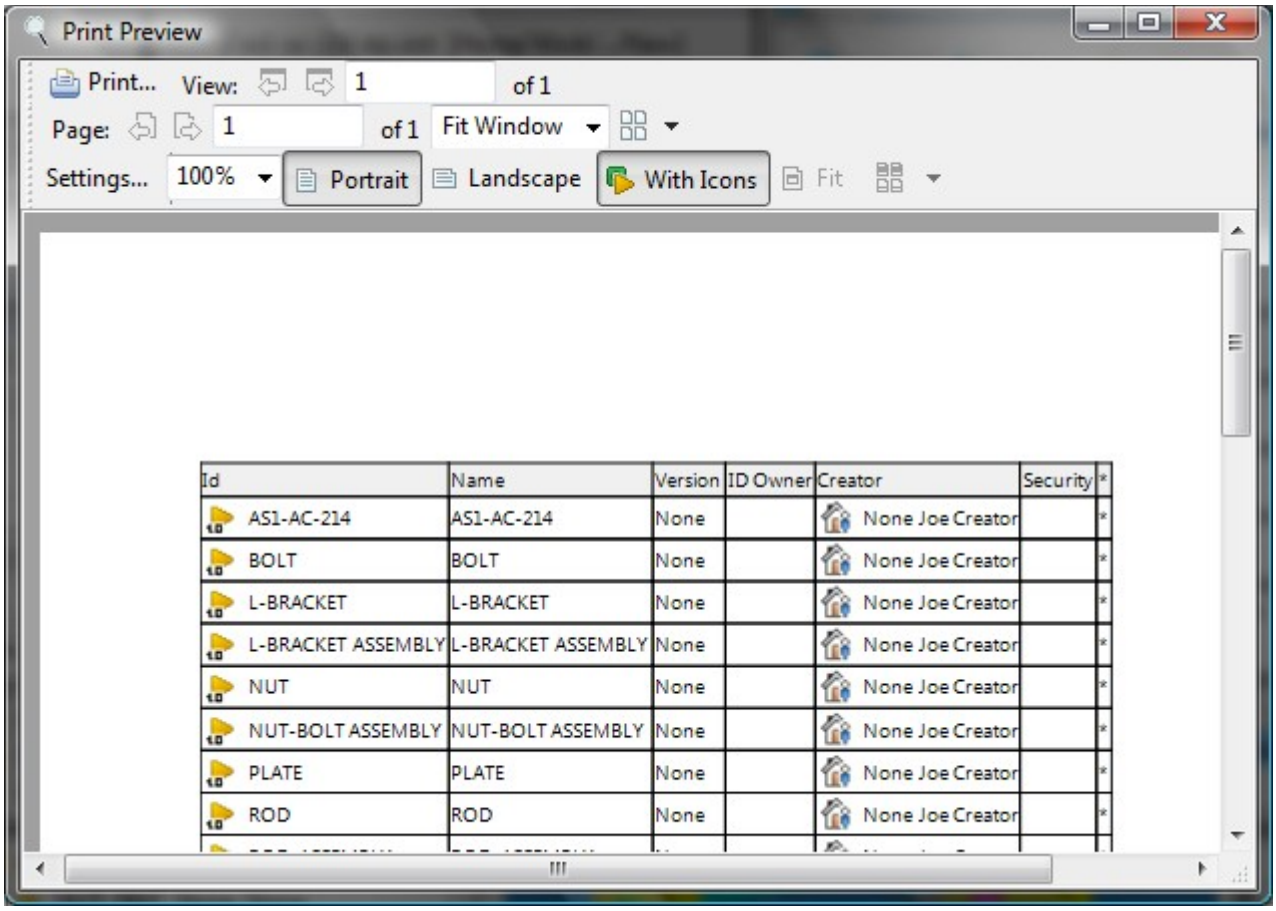

Figure 2. Print preview page of the current view

To top

Print views

If the **Print views** option is selected in the **File** menu, the **Select views** dialogue window to select applicable views appears on the screen (see Figure 3):

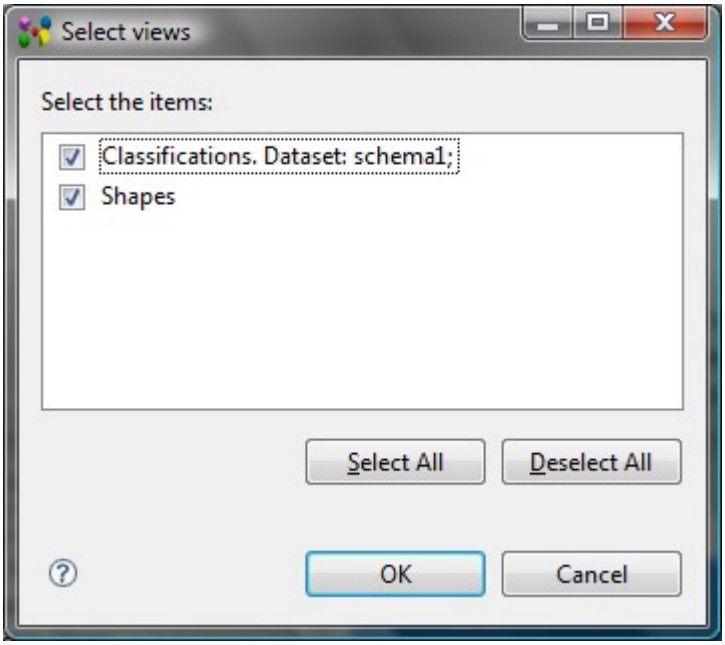

Figure 3. Select views dialogue window

- Choose views to be printed by selecting a check box in front of the particular view name in the **Select the items** list.

- Use **Select All** button to select all views, presented in the **Select the item** list. To unselect selected view(s), press **Deselect All** button.
- Press **OK** button to confirm your selection.

The **Print preview** window appears with recently selected views (see Figure 4).

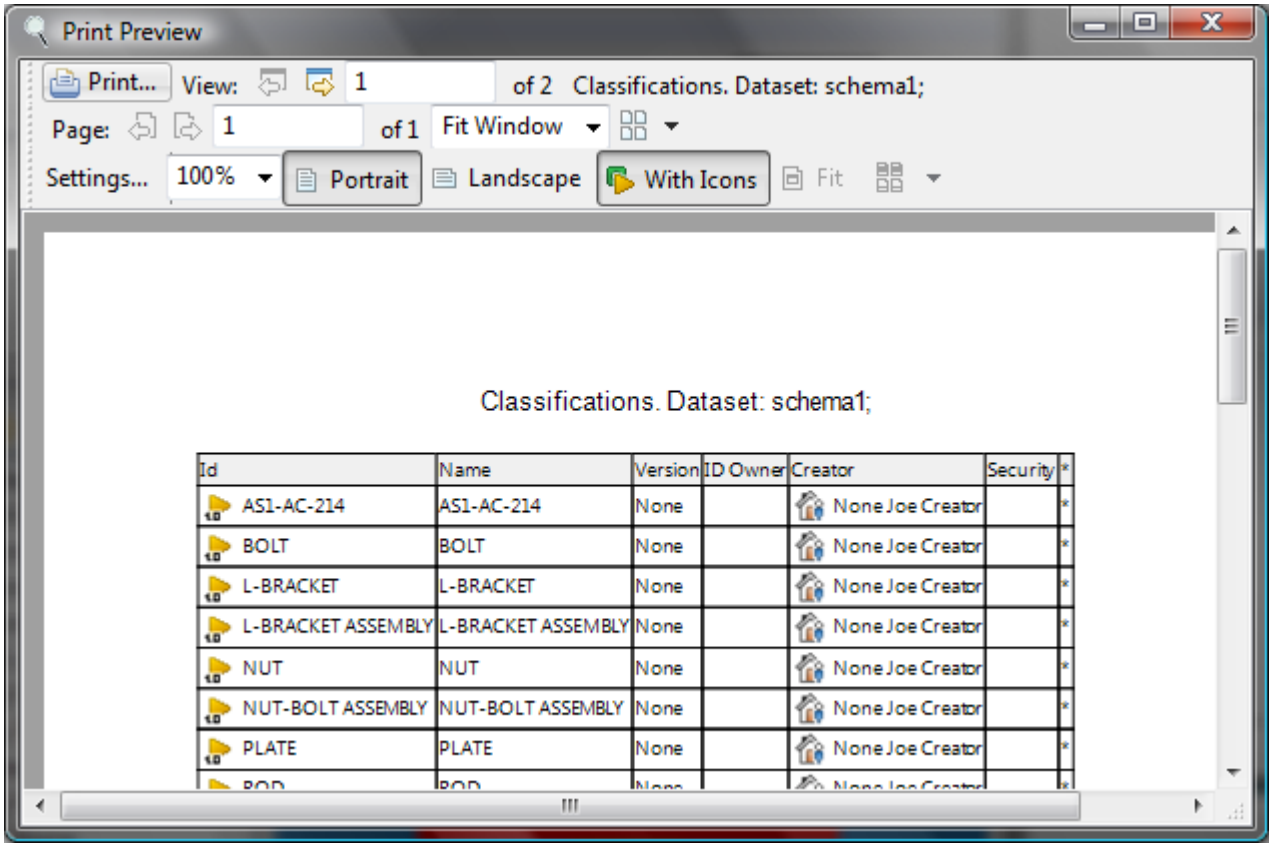

Figure 4. Print preview page with one of selected views

To top

Print preview settings

The **Print preview** page typically uses some default settings such as page size, layout and etc..

- **Print** button - the **Print** wizard is invoked to select a printer and change the selected printer settings.
- **Previous view** button - press the button to review the previous view.
- **Next view** button - press the button to review the next view.
- Text field next to the **Next view** button indicates the number of the current view in the preview page - you can enter a number of a view to be activated in the preview page. The read only label next to the text field presents the header of the view, which is presented.
- **Previous page** button - go to the previous page.
- **Next page** button - go to the next page.
- **Page number** text field next to the **Next page** button indicates the number of the page for the current view - you can enter the number of the page to be shown in the **Print preview** window.
- **Zoom** text field - enter an applicable zoom value or select the one from the drop down list. The preview page should be resized according entered/selected value.
- **Page Preview: Multiple Pages** - select a combination of pages the selected view to be previewed on. This is useful, if there is a need to preview a single view on multiple pages.
- **Settings** button - press this button to invoke the wizard page to change the printing parameters (see Figure 5):

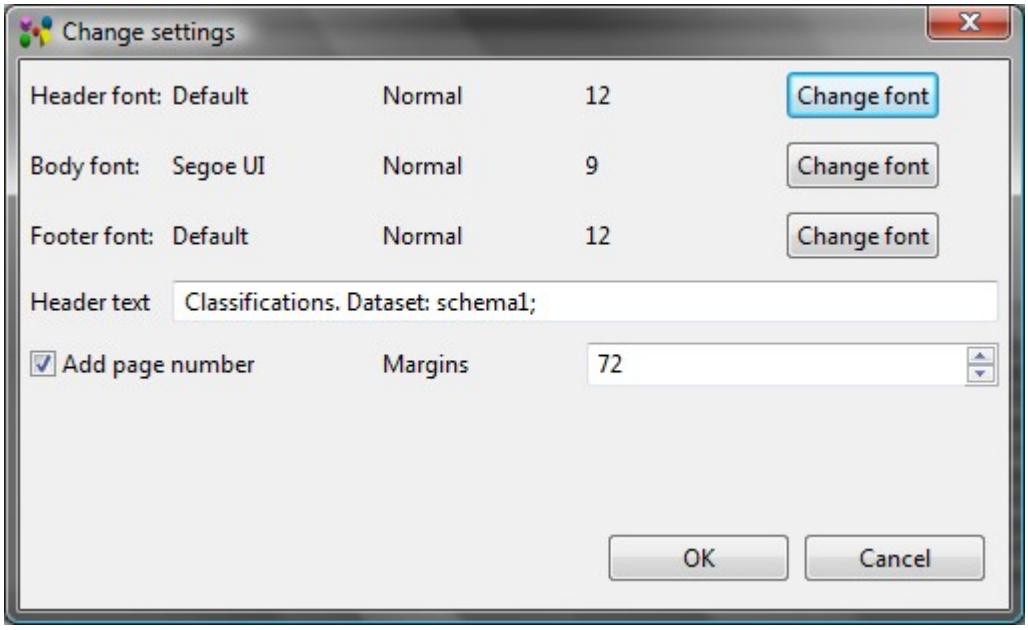

Figure 5. Change setting wizard page

- **Header font** - the page/view header font. Press **Change font** button next to change the font.
- **Body font** - the view/tree table font. Press **Change font** button next to change the font.
- **Footer text** - the page footer (page number) font. Press **Change font** button next to change the font.
- **Header text** field - enter a new or edit the currently defined text of the page/ view header.
- **Add page number** option - select the check box the page number to appear on the page.
- **Margins** field - to change margins of the page. Enter a number value or select an applicable value by using up and down arrows.
- **OK** button - press this button to confirm changes. To cancel changes use the **Cancel** button.

- **Scale** field - enter a new number value or select one from the list to scale a table/tree of the active view in the **Print preview** page.
- To change the layout of the page press one of the buttons **Portrait** or **Landscape**.
- **With Icons** button - to switch On/Off showing of icons in the table/tree of the active view preview page (see Figure 4, the option is switched On.).
- **Fit to page** button - to turn On/Off the fit to page option. A single drawing, in spite of it's size, is fit to one page (see Figure 6).

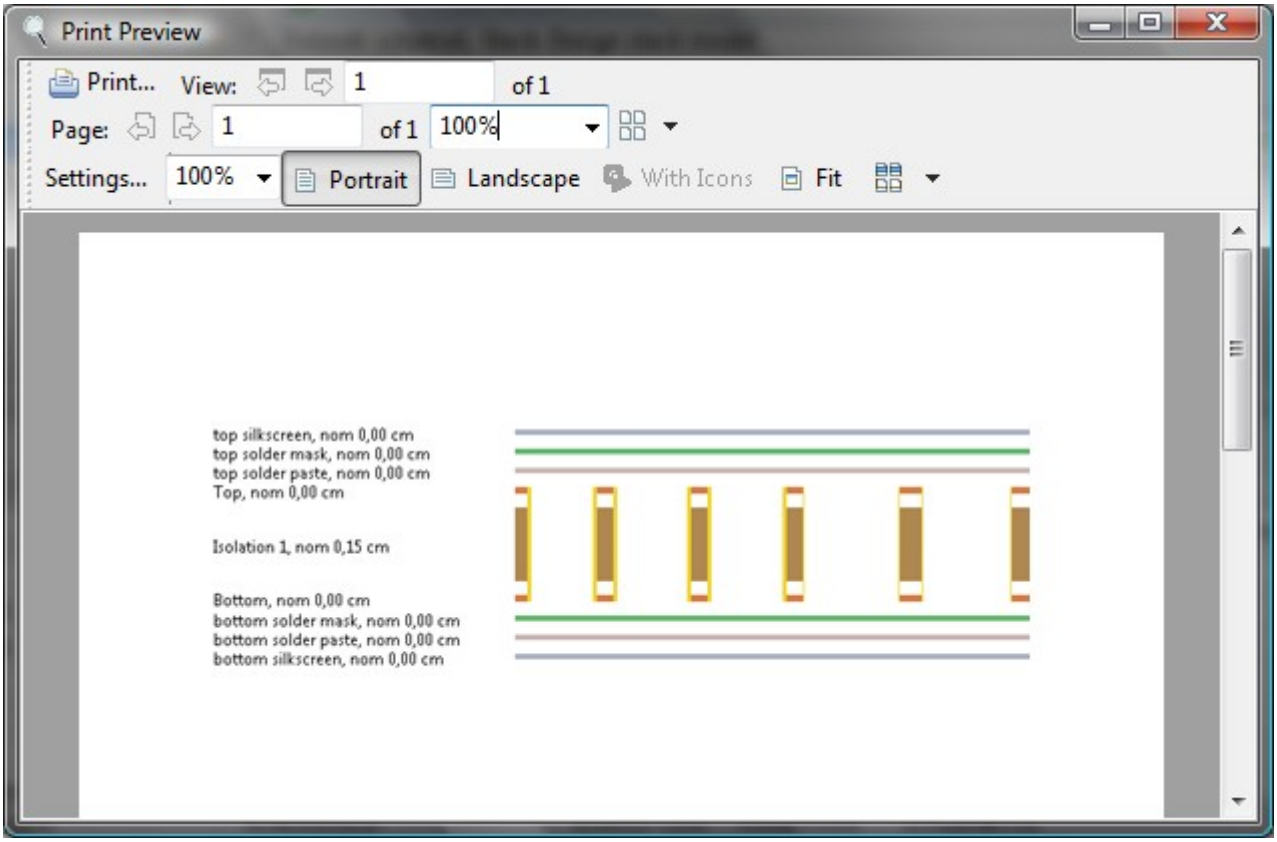

Figure 6. Drawing is fit to page

- **Draw On Multiple Pages** button - select a combination of pages the drawing of the selected view to be drawn on. This is useful, if there is a need to print a single drawing on multiple pages. The combination of the pages should be selected according the particular drawing size (if it is wide or tall). If the selected number of pages is too large, the "unused" pages are shown blank. You can switch on both buttons **Fit to page** and **Draw on Multiple Pages -** the drawing will be fit to all pages (see Figure 7).
  Note. **Fit to page** and **Draw on Multiple Pages** buttons are enabled only if the **Stratum Cross Section** view is the currently active view in the preview page.

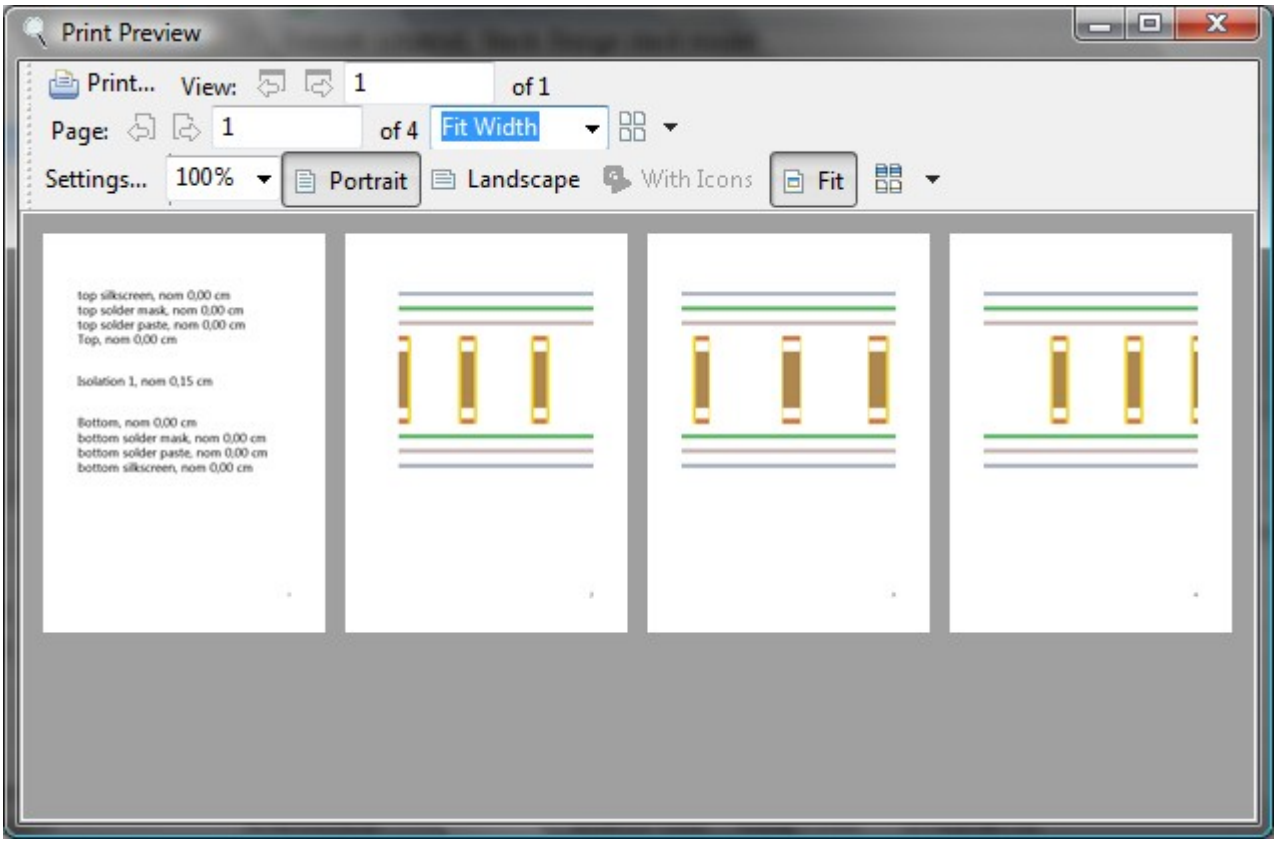

Figure 7. Single drawing is fit to 4 pages

The settings, defined in the **Print preview** page are applied only for the current printing/previewing. After this page is closed - the manually defined settings are not saved.

To top

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12.1. Reporting Errors

In case of an error condition please provide LKSoft with detailed information about the actual status of your IDA-STEP installation as described here:

1. Open **Log** view by selecting **Window --> Show View --> Other --> General --> Log** menu item.

    - The **Log** view is activated on the screen.
    - Copy all content by using keyboard *Ctrl+A*, *Ctrl+C* or use a context sensitive menu (*right click + Select All, right click+Copy* menu item).
    - Send the copied content to *support@lksoft.com*.

2. Select **About IDA-STEP** item in the **Help** menu. The About IDA-STEP dialog window appears (see Figure 1):

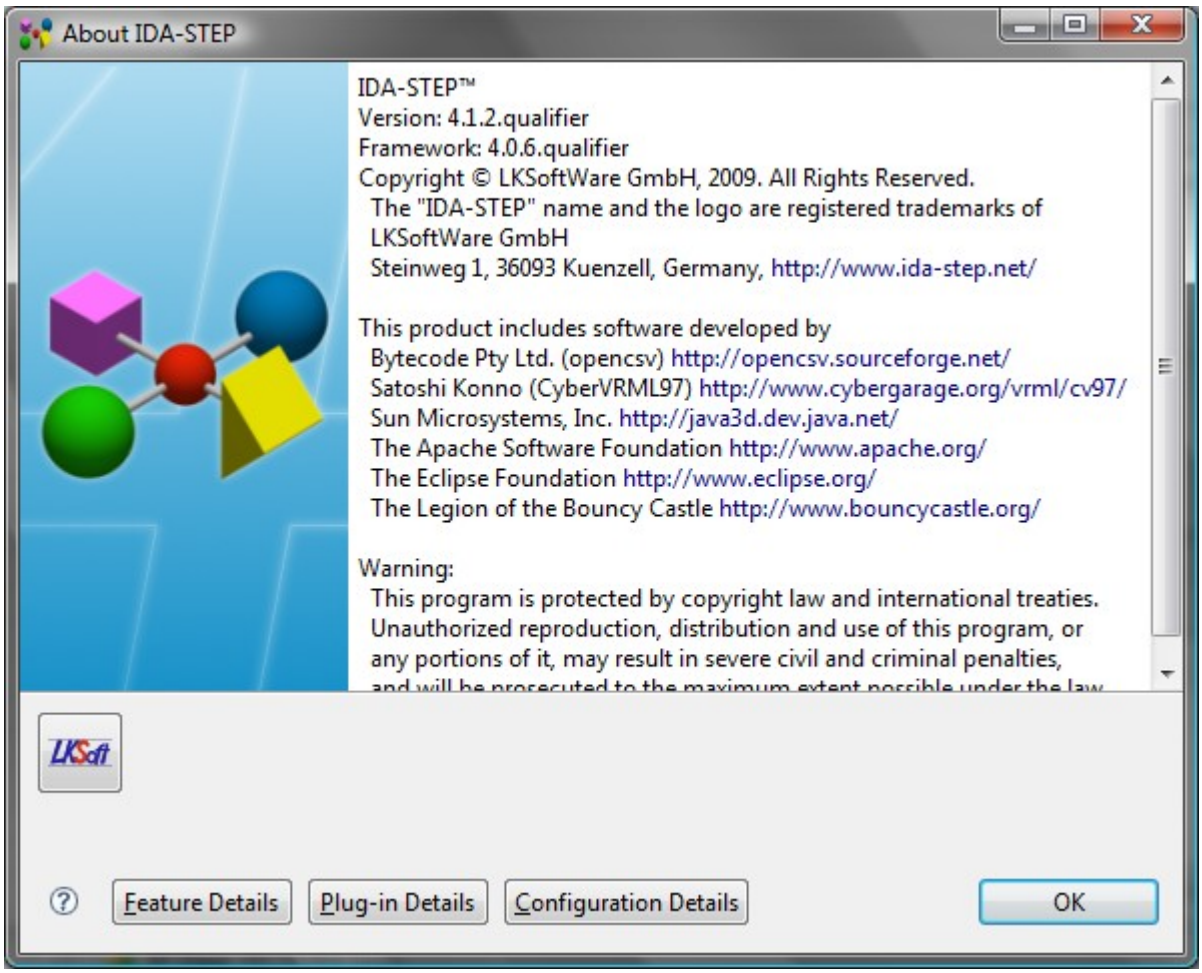

Figure 1. About IDA-STEP dialog window

- Click **Configuration Details** button. The **Configuration Details** window appears.

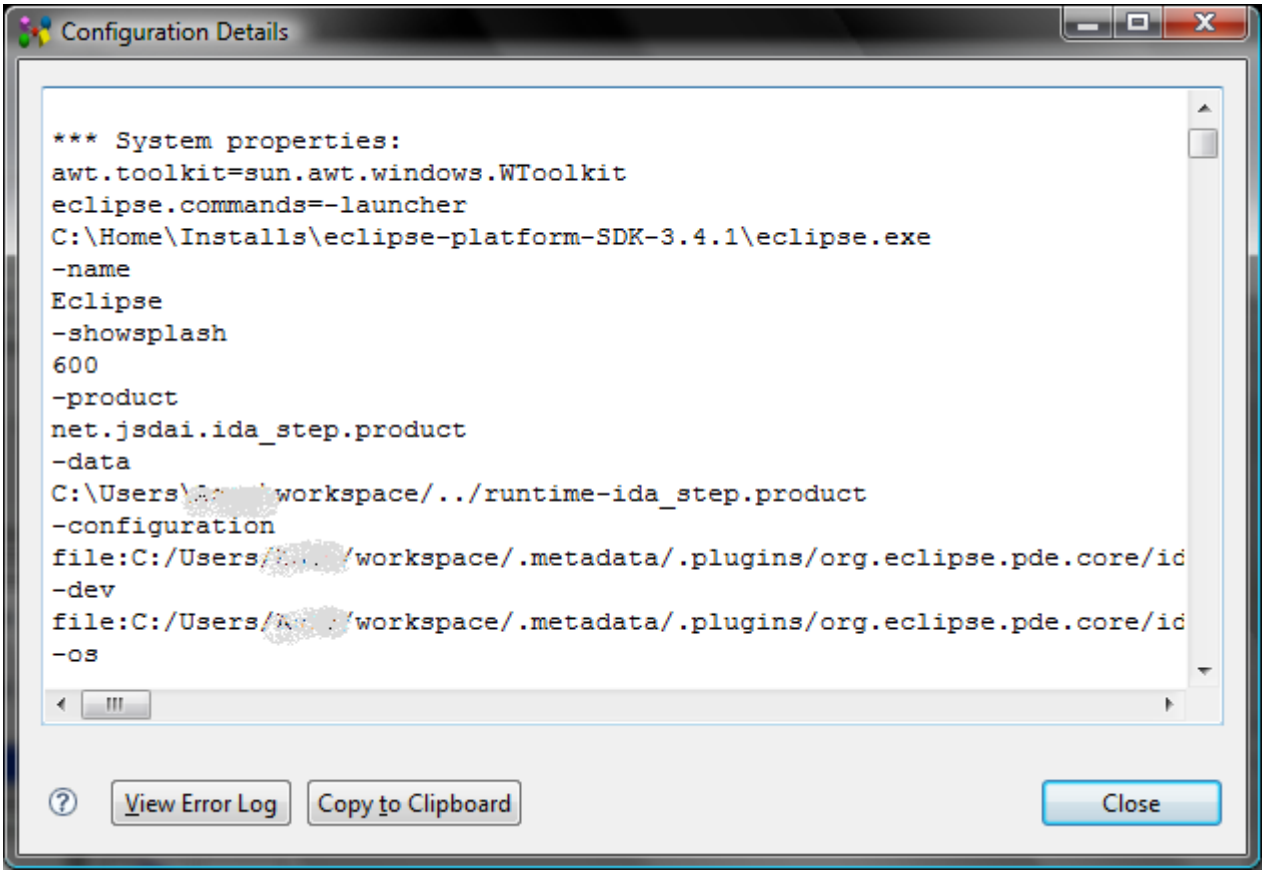

Figure 2. Configuration Details window

- Copy the content of the text box by using by **Copy to Clipboard** button.
- Send the content to IDA-STEP developers.

- Click **View Error** log button. Select the text editor in the appeared dialog window.
- Copy all content by using keyboard *Ctrl+A*, *Ctrl+C* or use a context sensitive menu (*right click + Select All, right click+Copy* menu item) and send it to IDA-STEP developers.

- Close the window after you finish with the error report by using **Close** button.

- Exit the **About IDA-STEP** dialog window by clicking OK button.

To top

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12.2.1. Error: Insufficient disc space on drive C:

While IDA-STEP installation is in progress, a message box with the following error appears: "Insufficient disc space on drive C:".

Cause

The installer requires about 100 megabytes of free disk space to expand in order to complete your IDA-STEP installation.

Solution

Delete files that you do not need from your C: drive (or whichever drive you are installing to), to create sufficient free space for the IDA-STEP installation.

To top

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12.2.2. Error: Failed to connect to https://services.ida-step.net/ through the Internet

While trying to install or update licensed features, the **Problem Occurred** dialog message can appear on the screen. Figure 1 shows one of the available error window variants.

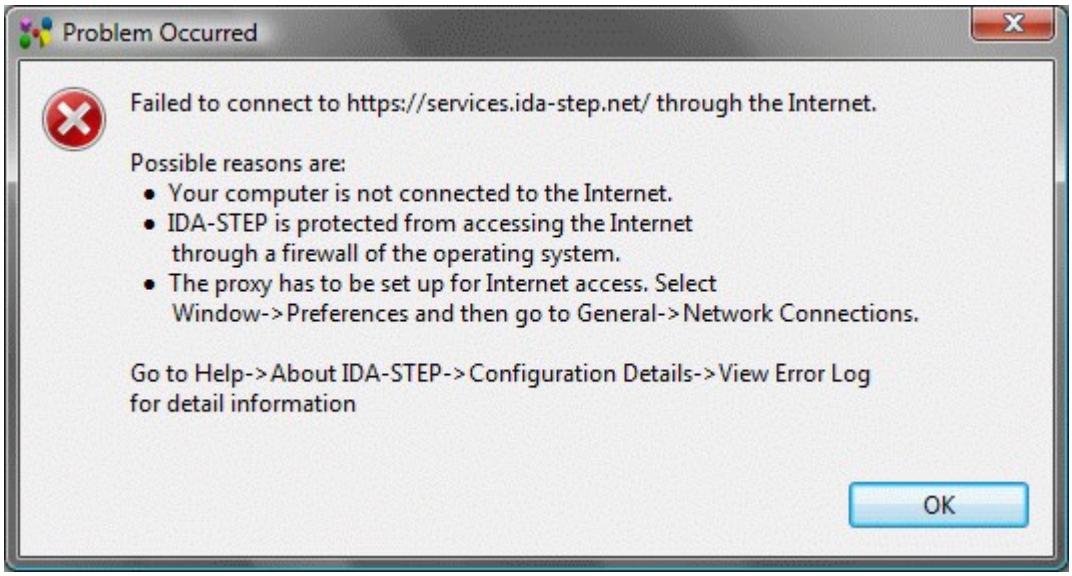

Figure 1. Problem Occurred message window

Cause

There may be the following causes for connection problems:

1. Your proxy possibly re-encoded data using it's own certificate which is not supported by IDA-STEP.
2. Possible reason is forged certificate by some kind of man in middle attack.
3. The failure was caused by HTTPS communication problem with the proxy.
4. The failure was caused by HTTPS communication problem.
5. Your computer is not connected to the Internet.
6. IDA-STEP is protected from accessing the Internet through a firewall of the operating system.
7. The proxy requires authentication therefore user name and password have to be entered.
8. The proxy has to be set up for the Internet access.

Solution

The solutions for causes specified above:

1. Go to Help -> About IDA-STEP -> Configuration Details -> View Error Log for detailed

information.
2. Go to Help -> About IDA-STEP -> Configuration Details -> View Error Log for detailed information.
3. Go to Help -> About IDA-STEP -> Configuration Details -> View Error Log for detailed information.
4. Go to Help -> About IDA-STEP -> Configuration Details -> View Error Log for detailed information.
5. Check your Internet connection, cable.
6. Change permission settings for a a firewall of the operating system.
7. Select **Window->Preferences** and then go to **General->Network Connections**. Enter a user name and password in the **Network Connections** preferences page.
8. Select **Window->Preferences** and then go to **General->Network Connections**. Set up the proxy for the Internet access.

If measures listed in the dialog box does not help, the user should go to **Help -> About IDA-STEP -> Configuration Details -> View Error Log** for detailed information and email the error log to IDA-STEP support (see Reporting errors).

To top

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12.3.1. Error: License ID unknown

After the license ID is entered in the **Register, Update and Install** dialog window **New license ID** field and the **Next** button is pressed, the A**ctivation Failed** message box with the error message 'License ID unknown' may appear on the screen (see Figure 1).

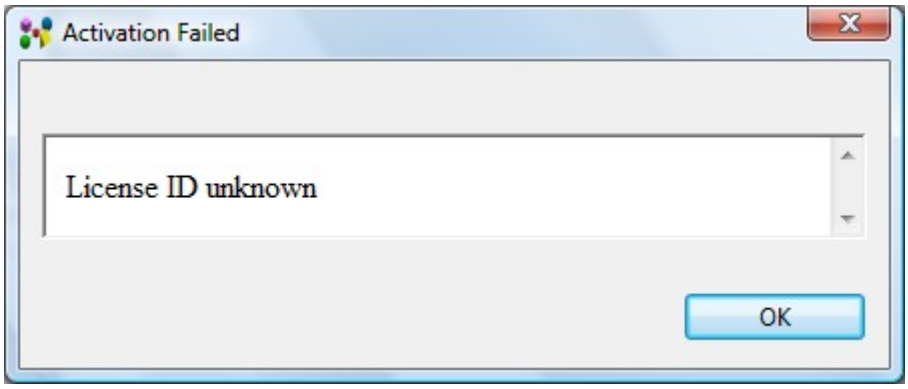

Figure 1. Activation failed dialog with error message "License ID unknown"

Cause

The license identifier is unknown.

Solution

Enter an available license ID or get a new one.

To top

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12.3.2. Error: License is no longer valid

While updating IDA-STEP by using **Register, Update and Install** dialog window, the **Activation Failed** message window with the error message 'This license is no longer valid' may appear (see Figure 1).

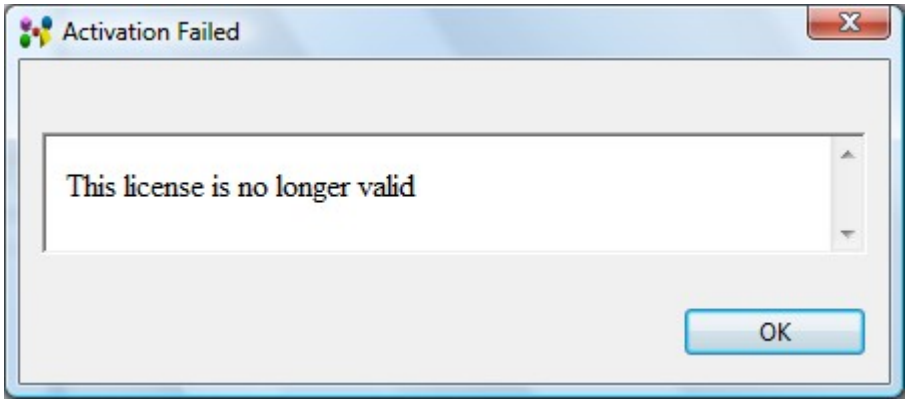

Figure 1. Activation Failed message box with error message

Cause

The current license is no longer valid (blocked).

Solution

Get a new license to install IDA-STEP.

To top

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12.3.3. Error: No rehostings left

While the same license is used for the different computer, the error message window appears (see Figure 1):

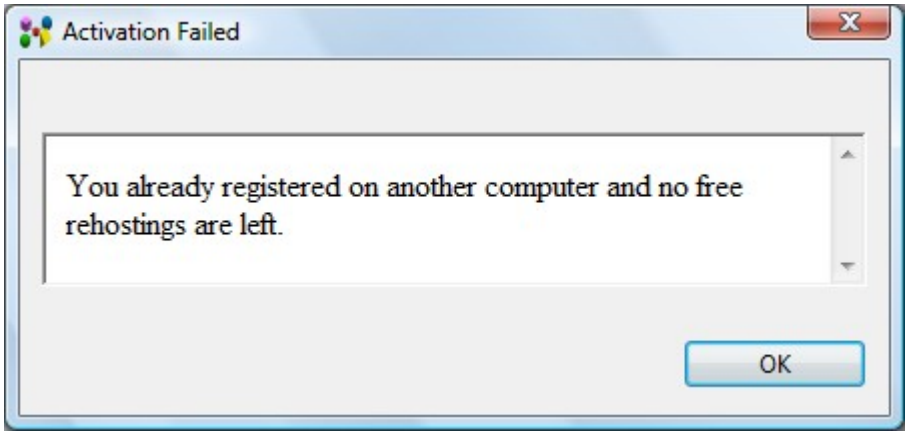

Figure 1. Activation failed dialog with error message

Cause

IDA-STEP is licensed for a single computer.

Solution

Get a new license to install IDA-STEP on the different computer.

To top

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12.4. Internal error

The **Error** message window is popped up on the screen (see Figure 1).

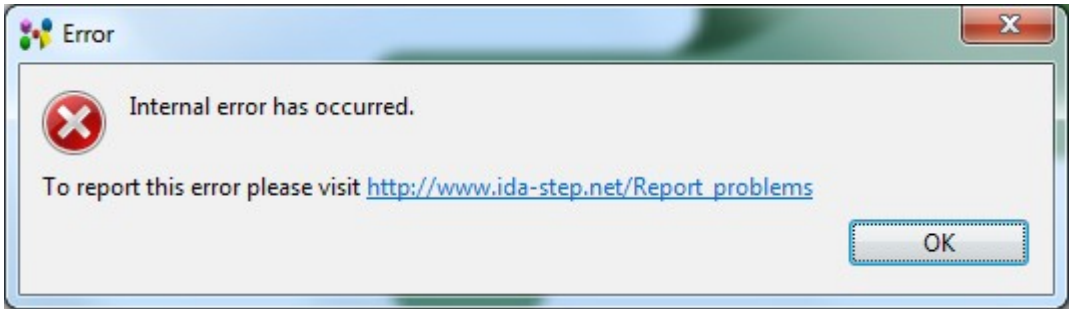

Figure 1. Error message window

Solution

Click on the hyper link http://www.ida-step.net/support/troubleshoot and follow instructions, presented in the **How to report problems with IDA-STEP** page.
Or
Go to **Help -> About IDA-STEP -> Configuration Details -> View Error Log** for detailed information and send by e-mail the error log to IDA-STEP support.

To top

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