



Tetra4D Reviewer

Version 2021

User Guide

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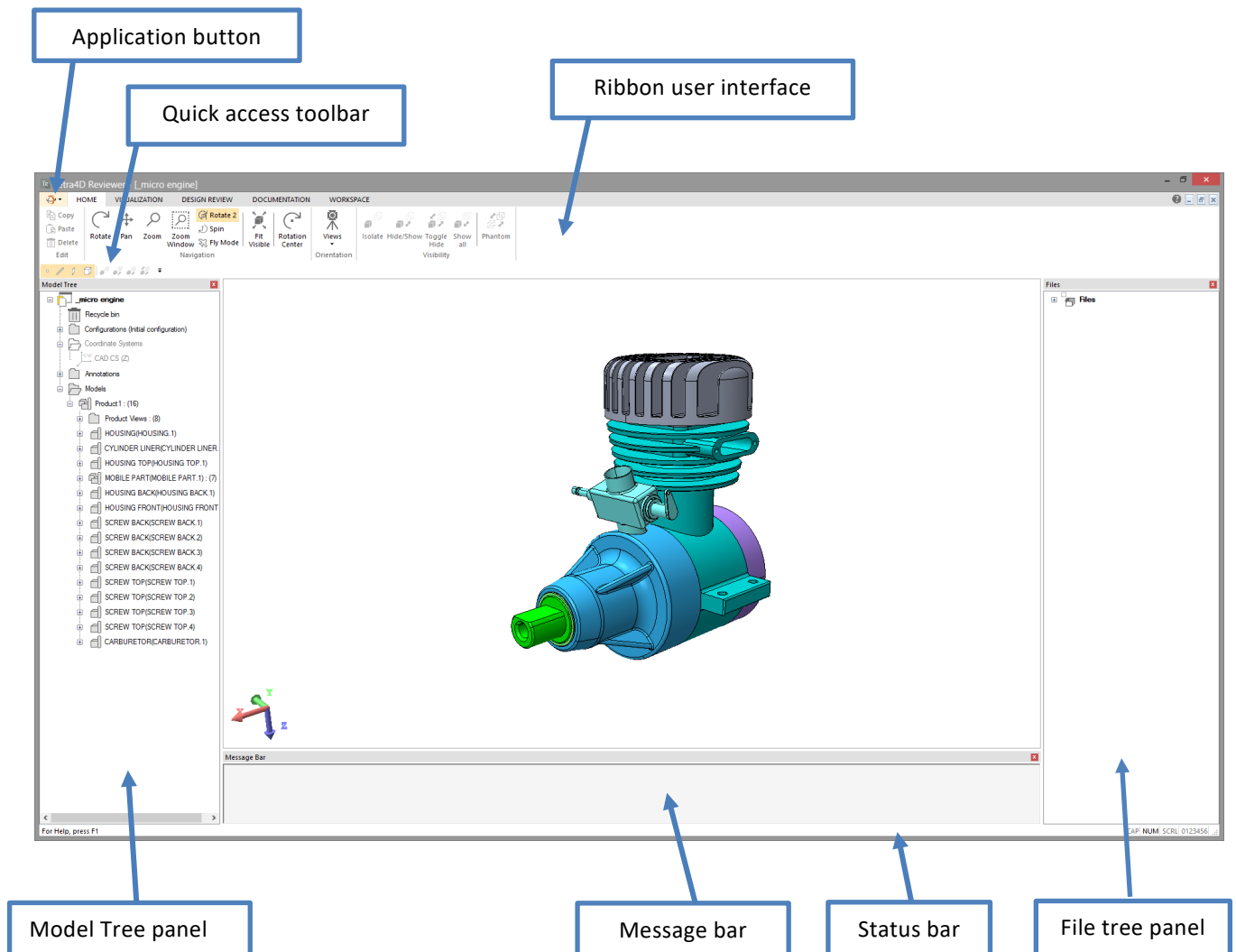
Chapter 1: Tetra4D Reviewer™ overview

Tetra4D Reviewer™ is a companion product of Tetra4D Converter™.

Tetra4D Converter™ allows you to create 3D PDF documents from a wide range of 3D CAD files or from existing PDFs while Tetra4D Reviewer™ provides some additional features dedicated to the extraction of accurate information (physical properties calculation, exact dimensioning, and sectioning...) and to the authoring of the 3D scene (merging and positioning different CAD models, changing part colors...).

Tetra4D Reviewer™ application

The Tetra4D Reviewer™ application is split in several areas. Each area has a specific purpose. The first time Tetra4D Reviewer™ opens, the areas are organized in their default states. The next time you open Tetra4D Reviewer™, the work areas appear as they did at the end of the previous work session.

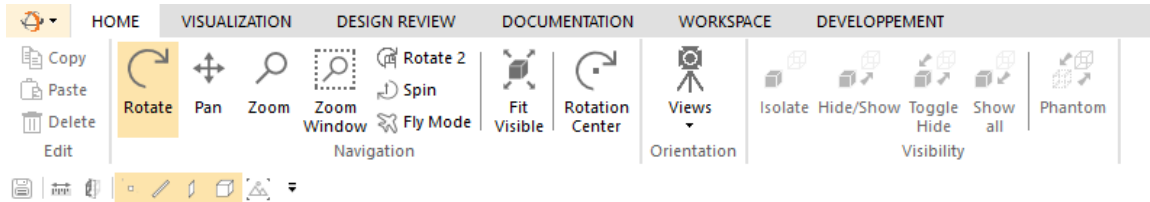


Tetra4D Reviewer™ user interface

The features of the software are accessible through a ribbon user interface.

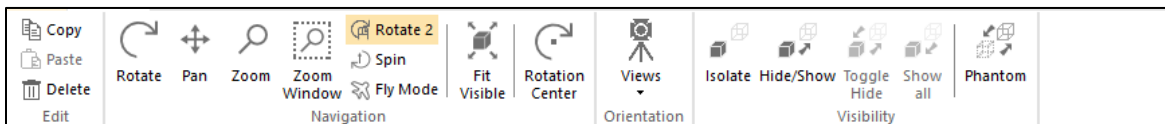
The ribbon is divided into several tabs, where all the product features are gathered to their usage.

Inside the ribbon, the features are grouped in panels.






“HOME” tab










The Home tab provides the access to 3D control features.












Edit panel

 Copy	 Paste	 Delete
Copies the selected entities in the clipboard	Pastes the clipboard into another document	Deletes the selected entities






Navigation panel

 Rotate	Main 3D navigation method
 Pan	Pan horizontally and vertically.
 Zoom	Dynamic zoom.
 Zoom Window	Zoom window
 Rotate 2	Second navigation method
 Spin	Spin (rotation with vertical rotation axis)
 Fly Mode	“Fly through” navigation mode
 Fit Visible	Fit the visible parts to the screen
 Rotation Center	Set a center of rotation

Orientation panel

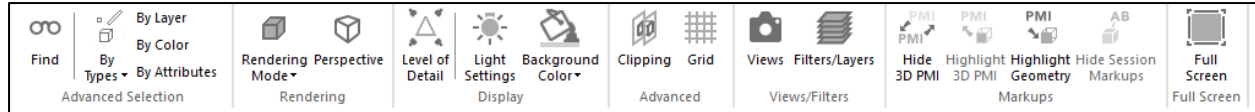
	Activates a default view
 Iso	Sets the 3D in isometric view
 Top  Bottom  Left  Right  Front  Back	Activates the selected view according to the activate “view” coordinates system
 Set Camera ▶	Sets the view orientation based on a selected 3D entity
	Restore Config. View From Selection From Curve/Edge From Face From Coordinate Axis Flip

Visibility panel



 Isolate	Makes the selected objects visible (hide all the unselected objects)
 Hide/Show	Hide the selected objects
 Toggle Hide	Toggles the visibility of all the objects
 Show all	Makes all the objects visible
 Phantom	Activates the “phantom display” mode

“VISUALIZATION” tab







The Visualization tab provides the access to the features to control the selection options, the 3D rendering settings and to access to the CAD defined views and 3D notes and Product Manufacturing Information (PMI).






Advanced Selection

 Find	Opens a dialog box to find entities
 By Types ▾	Selects entities based on their type Points Wires Coordinates Systems Planes Surfaces Solids Markups
By Layer By Color By Attributes	Select entities based on their layer, color, or attribute value



Rendering panel

 Rendering Mode ▾	Changes the rendering mode of the 3D model
 Solid  Solid Outline  Wireframe  Illustration	
 Perspective	Toggles the 3D display in perspective or orthographic projection



Display panel

 Level of Detail	Enables to change the level of details of the 3D model
 Light Settings	Enables to position the light(s) in the 3D scene
 Background Color▼	Enables to select a background color for the 3D scene





Advanced panel

 Clipping	Activates cutting planes for visualization
 Grid	Displays a 2D grid to enable quick measurements


Views/Filters panel

 Views	Provides the access to the CAD defined views.
 Filters/Layers	Provides the access to the CAD defined layers and filters.

Markups panel

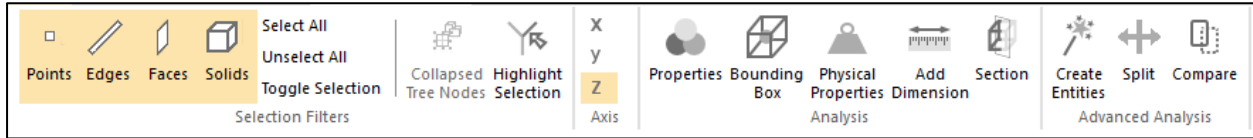
 PMI Hide 3D PMI	Toggles the visibility of the PMIs
 PMI Highlight 3D PMI	Highlights the PMIs that are linked to a selected entity
 PMI Highlight Geometry	Highlights the entities that are linked to a selected PMI
 AB Hide Session Markups	Toggles the visibility of the session markups

Full Screen panel

 Full Screen	Toggles the 3D window to Full Screen mode
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“DESIGN REVIEW” tab







The “Design Review” tab provides the access to the features that are dedicated to analyze and check the design of 3D models (measures, section, physical properties calculation...).



Selection filter

The selection filters define what type of information will be selected when the user clicks on a 3D object in the 3D window. When a specific selection filter is inactive, the corresponding entities can not be selected. Conversely, if the same selection filter is active, the these types of entities will be selected. When all selection filters are active, clicking selects any type of geometry close to the location you click.






Depending in the context of the feature, the selection filters may be automatically set. For example, when the Radius measurement tool is active, the Points, Faces, and Solids filters are deselected and only the Edges filter remains active so that only appropriate entities for a radius measurement can be selected by the user.

 Points	Selection of points
 Edges	Selection of edges
 Faces	Selection of faces
 Solids	Selection of solids
Select All Unselect All Toggle Selection	Selects all the visible entities (ignoring the selection filters) Unselects all the selected entities Toggles selection
 Collapsed Tree Nodes	Select the “collapsed tree node” when one of its component is selected in the 3D window
 Highlight Selection	Highlights the selectable entities when the mouse cursor hovers the 3D model




Axis

X Y Z	Sets the main active axis according to the current active coordinates system
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Analysis panel

 Properties	Provides general information for a selected object (type, color, attribute...)
 Bounding Box	Enables to calculate the bounding box for the current selection
 Physical Properties	Enables to calculate the physical properties (volume, weight) for the current selection
 Add Dimension	Enables to measure and create dimensions
 Section	Activates the section feature and make it possible to calculate sections

Advanced analysis panel





 Create Entities	Enables to create simple geometric entities based on the 3D models
 Split	Enables to analyze if a plastic part is suitable for mold design
 Compare	Enables to compare different version of the same 3D model

“DOCUMENTATION” tab



The “Documentation” tab provides the access to the features that are dedicated to analyze and check the design of 3D models (measures, section, physical properties calculation...).






Configuration panel

 Create	Saves the current settings of the 3D window (visible parts, camera position...) in a new configuration
 Show	Enable to create a new configuration with the current selection of objects
 Activate Previous	Activates the previous configuration
 Activate Next	Activates the next configuration

3D Markups panel

 Add Markup	Creates a 3D markup
 Create Balloons	Enables to create balloons attached to parts



Modify panel

 Drag & Drop	Allows to move components based on a 3D manipulator
 Move	Allows to move components based on various methods
 Restore Initial Position	Restore the initial position of a part that has been moved







Bill of Material panel

 Bill of Material	Enables to create a part list
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Animation panel

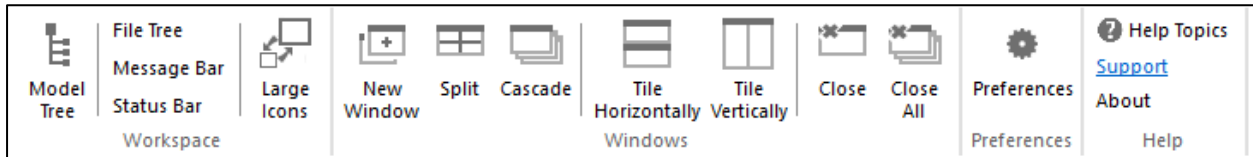
 Create Animation	Enables to create exploded views and animations
 Play Animation	Enables to play an existing animation

Screenshots panel



 Snapshot	Allows to create a screen shot (raster image) using one of the below methods
 Window to Raster	Saves an image of the full 3D window
 Region to Raster	Saves an image of a user defined sub-area of the 3D window
 Window to Clipboard	Adds the image of the full 3D window to the clipboard
 Quicksnap	Saves an image of the full 3D window without asking for name
 Window to Vector	Allows to create a vector graphics file of the full 3D window

“WORKSPACE” tab








The “Workspace” tab provides the access to the features that are dedicated to control the visible areas of the software and to manage the documents’ windows.




Workspace tab

 Model Tree	Allows to display or hide the model tree panel
File Tree Message Bar Status Bar	Allows to display or hide the file tree panel Allows to display or hide the message bar Allows to display or hide the status bar
 Large Icons	Allows to display large icons in the model tree and the file tree panels


Windows tab

 New Window	Creates a new window for the active document
 Split	Split the document’s window into 4 views
 Cascade	Arranges all the windows in cascade
 Tile Horizontally	Arranges all the windows horizontally
 Tile Vertically	Arranges all the windows vertically
 Close	Closes the active window
 Close All	Closes all windows

Preferences tab

 Preferences	Provides the access to the preferences settings
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Help tab

 Help Topics Support About	Opens the user guide Opens the web page of Tetra4D Reviewer™ support. Opens a dialog box showing product information
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“APPLICATION” button

This button provides the access to the “Files” related commands:

										
New	Open	Import	Save	Save back in PDF	Save As	Export	Close	Document LOG	Print	Exit

Document pane

Grids


Grids are available as part of the background for 2D and 3D views. You control the color, style settings, and positions of the lines and units in the Grid preferences. The active coordinates system determines the grid center point.

The grid lines adjust automatically when you change the zoom level. You can include the grid when you print.

Show or hide the grid

1. Click  , in VISUALIZATION tab.

Set grid options

1. Click  in WORKSPACE tab.
2. On the Grid tab, select options for the axes and scale.

Full Screen mode

Full Screen mode expands the document display to cover the entire screen. Full Screen mode hides the title bar, menus, toolbars, Model Tree, File Tree, Message bar, and Status bar. The only control visible is a floating toolbar with a single button. Clicking that button restores the standard work area view.

Activate Full Screen mode

1. Click  in VISUALIZATION tab.

Escape Full Screen mode

1. Press the ESCAPE key.




The Ctrl-L keyboard short cut toggles between Full Screen and standard views.

Change the detail level

The on-screen display of 3D objects approximates the curves and angles in those objects. The Level of Detail setting determines how closely the display matches the actual geometry.

The Level of detail directly affects the display. Indirectly, it affects a few other functions. For example, Level of Detail settings affect the selection modes when you take measurements or dimensions. Level of Detail also affects the results of calculations you make based on display information. It has no effect on calculations based on exact geometry.



1. Click  in VISUALIZATION tab.
2. Choose the level you want in Level of Detail.
3. Click Apply, or click Close to leave the Level of Detail unchanged.


Note: Level of Detail is available only for documents that contain actual geometry. The geometry depends on the content of the read CAD data.

Note: If you choose Advanced, the Advanced button opens a dialog box with the same options described for Physical Properties.

Change the background color

The background color is associated with the active configuration. You can set different background colors for individual configurations in a session file.



1. Click  in VISUALIZATION tab.
2. Select the color.

Note: Background changes are temporary if the active configuration is locked. To make the color change permanent, unlock the configuration after you change the background, and then immediately lock it again.

Advanced background options are available on the 3D Preferences tab.

Move the light source

The orientation of the light source associated with a configuration. You can set a different light source for individual configurations in a session file.



1. Button  in VISUALIZATION tab.

Note: A light icon appears in the document pane.

2. Drag the light  to change its position.

Note: As you drag the light, rotational axes appear that help you visualize the light position.

Arranging windows in the document pane

Tetra4D Reviewer is a multi-document application. It is possible to:

- Work with several documents opened in different windows.
- Copy and paste objects and entities between documents.
- Create different views from the same document. The Split window command displays different views in the original window. The New Window command displays views in separate windows.

The document windows for files follow behaviors that are common for virtually all Microsoft® Windows® applications and system windows. For example, you can use the buttons in the upper right

corner to minimize, maximize, and restore the window size. You can drag the title bars or edges of a file to resize it.

For more information on these basic functions, see Microsoft Windows Help system.

Showing multiple views of sessions

You can open an individual session file in multiple windows. Each window can show a view of the assembly that is independent of the views shown in the other windows. For example, when you can adjust the point of view, display mode, and zoom in one window, other windows for the same session file remain unchanged.

You can resize the window like any other window by dragging an edge or corner of the window. Splitting, cascading, or tiling automatically arranges the windows in the document pane so that you can see all open files simultaneously.

Note: Dimensions and the markups that you create in a window appear in that window but not in other windows of the same session file.

Split views within one session window

Splitting divides the window into four sections. Each section shows a different default view of the 3D objects.



1. Click **Split**, in WORKSPACE tab.
2. Adjust the sashing that separates the four views as needed:
 - Drag the sashing horizontally or vertically to resize all four sections.
 - Drag the sashing to the edge of the document pane to close two of the sections.
 - Double-click the intersection of the two sashings to close three of the sections, leaving only one view open.

Open a duplicate window for the active session file






1. Click **New Window** in WORKSPACE tab.

The appearance in the document pane doesn't change because new window fills the area and shows the same view as the original window. You can resize and arrange the window by the techniques common to most Microsoft Windows applications.

Arrange multiple windows

Choose one button of the followings:

-  **Cascade**: Arranges all currently open session files in identically sized windows that overlap.
-  **Tile Horizontally**: Arranges all currently open session files in equal horizontal rows.
-  **Tile Vertically**: Arranges all currently open session files in equal vertical columns.

Model Tree panel

About the Model Tree

The Model Tree is a hierarchical structure that organizes the elements of the document by type. When multiple CAD files are open, each document appears in the Model Tree.





The structure for each open document includes five major categories:

- **Recycle Bin:** Stores any entities you delete from that individual document.
- **Configurations:** Saved views of the parts and some session information, such as Bill of Material and Animations created in that configuration.
- **Coordinate Systems:** Coordinates systems defined by the CAD file and any custom coordinate system created by you.
- **Annotations:** Markups and dimensions you create.
- **Models:** The assembly and its component.

The Model Tree information is imported from the CAD source file.

Some Reading mode preferences may impact the Model Tree structure.

Model Tree icons indicate entity types. For example:

- an assembly (or sub-assembly) icon 
- a component icon 
- a solid 
- a surface .

Expanding and collapsing Model Tree structures

You can change your view of the Model Tree in two ways. You can expand individual levels, or you can simplify the tree structure by collapsing sub-levels of the hierarchy.

Plus [+] and minus [-] icons expand and collapse individual tiers of the Model Tree. By default, the plus and minus signs are available at all levels of loaded assemblies.

The Collapse command simplifies the tree structure by limiting the number of levels that appear. The Expand command reverses the Collapse command so that you can open all levels of the structure.

Note: Plus and minus signs don't appear under Models for part levels you've collapsed with a context menu command.

Expand and collapse individual elements

1. Click the plus sign [+] to expand a branch by one level.
2. Click the minus sign [-] to collapse all levels within that branch.

Note: In the Models section of the Model Tree, plus and minus signs appear only for loaded parts. If you load only an assembly structure, only the top assembly level will appear in the hierarchy.

Expand or collapse one or more levels

Expand commands on the context menu reveal the plus sign icons [+] for selected branches. Collapse commands collapse branches and hide the plus and minus signs for the selected elements.

Right-click a branch of the Models hierarchy and choose an Expand or Collapse command:

- **One Level:** To restore the plus sign [+] to the next tier of the branch structure of the selected entity. (Available only on the Expand menu.)
- **Component:** To change the availability of the plus [+] and minus [-] signs for entities nested under the selected element.
- **All:** To change the availability of the plus [+] and minus [-] signs for the selected branch and the entities nested under that branch.

Note: Context menus for Models entities, such as assembly parts, bounding boxes, and sections, have Expand and Collapse commands. These commands don't appear on context menus for other Model Tree entities, such as configurations and annotations.

Models hierarchies

The Models structure in the Model Tree expands to show several types of entities and information. The plus [+] and minus [-] icons beside the structure names expand and collapse the individual levels. Primarily, the Models shows the assembly structure for the 3D objects. The arrangement of subassemblies, component parts, and items reflects the structure read from the CAD file. The assembly structure also contains structures for any Product Views and 3D PMI markups read from the CAD file.

Any Sections, Bounding Boxes, created entities, and imported or pasted-in parts appear under Models at the same level as the assembly.

About attributes

An attribute is non-geometric information linked to a part. The CAD data defines attributes information. That information is included when you import the CAD file into Tetra4D Reviewer. Tetra4D Reviewer can include some attribute values in text that identifies individual parts of an assembly. This information appears at the fully expanded level for each individual component. Showing or hiding attributes is set at the document level. You can select which attributes to show or hide.

Show attributes in the Model Tree

1. In the Model Tree, right-click the document name and choose Show Attributes.
2. Under Attributes, select the attributes that you want to show:
 - Select and deselect check boxes for individual attributes.
 - Select the Select All ☒ or Unselect All ☐ for all attributes.

Note: Clicking an attribute name in the Attributes pane makes any known values for that attribute appear in the Values pane.

Recycle Bins

Recycle bins in the Model Tree stores entities that you delete from session files. Each open file has its own recycle bin. You can restore deleted items to the document. You can also delete items permanently by emptying the recycle bin.

Note: The recycle bins within Tetra4D Reviewer are independent of the Recycle Bin on the Windows desktop.

Restore deleted entities

1. Click the plus sign [+] to expand the Recycle Bin for the current document.
2. Right-click the entities that you want to restore and choose Restore.

Empty a recycle bin

1. Right-click the Recycle Bin that you want to empty, and choose Empty Recycle Bin.

File Tree panel

The File Tree panel provides you with the access to the incremental loading features. By default, the File Tree is closed when Tetra4D Reviewer starts.

You can open or close the File with button **File Tree** in the WORKSPACE tab

Show or hide information bars

The Message bar appears at the bottom of the work area, just above the Status bar. The Message bar can show information about any reading problems or direct your attention to part of the work area. The Status bar appears across the bottom of the work area. In some contexts, the Status bar contains information about the current process. For example, the Status bar can provide cues about actions needed during the process of a specific task.

Show or hide an information bar:

- Click **Message Bar**, in WORKSPACE tab.
- Click **Status Bar**, in WORKSPACE tab.

Chapter 2: Opening and importing files

Tetra 4D Reviewer supports the reading of multiple 3D CAD formats, some 2D CAD formats, PDFs, and PRW files. PRW files are Tetra4D Reviewer™ session files.

You can open 3D data in new documents or import them into an existing session file.



Complex CAD files may require large amounts of memory. You can load files incrementally so that only some chosen parts open in Tetra4D Reviewer™. This option makes the loading more efficient and speeds up the reading process. Incremental loading is ideal if you want to work only on a few parts of a large assembly.

Opening a CAD file from within Tetra4D Reviewer™

Computer Aided Design (CAD) applications create the 3D models, using a variety of formats. You can view 3D models in Tetra4D Reviewer™ by opening CAD files or by importing or pasting parts from other files.

You can open the 3D models embedded in PDF files in Tetra4D Reviewer™. If the PDF contains multiple 3D models, you can open the models in new documents or add them to existing documents. Tetra4D Reviewer™ doesn't display other types of PDF content that are not within 3D models, such as surrounding text blocks or layout graphics.

The same basic method applies whether you want to open a CAD file, a Tetra4D Reviewer™ session file, or a 3D model embedded in a PDF.

1. Click  in the  application button and select the file.

Note: The Options button at the lower left of the Open dialog box opens the Preferences dialog, providing general reading modes options as well as format specific options.

Open a PDF that contains multiple 3D models

When you open a PDF with multiple 3D models, the document pane remains empty until you select which model you want to load.

1. In Tetra4D Reviewer™, button **File Tree** in WORKSPACE tab, if necessary.
2. Expand the Files structure and the PDF folder.
3. Right-click the model that you want to open, and choose where you want to open it:
 - Insert in Current Product: Adds the model to the current open document.
 - Create a New Product: Opens the model in a new document.

Note: A PDF can include multiple 3D models and other types of content, such as ordinary graphics and text blocks. These other types of content do not appear when you open the file in Tetra4D Reviewer™.

Open a model from a “multi-model” CAD file

Some CAD formats can include multiple configurations in a file. When you open this type of multi-configuration file in Tetra 4D Reviewer, you have first to select the model that you want to activate.

1. If necessary, click the **File Tree** button in the WORKSPACE tab to open the File Tree pane.
2. Click plus signs [+] to expand the Files Tree and access to the format of the models.

Note: A number, (x), after a format name indicates the number of models in that format.

3. Right-click the model that you want to load, and choose an option:
 - **Insert in Current Product:** Opens the selected model in the active document.
 - **Create a New Product:** Opens the selected model in a new document.

Open a 3D PDF from within Tetra4D Converter™

1. In Acrobat® ProDC/2017, activate the 3D model in the PDF.
2. Expand Tetra4D Converter™ plugin in the tools pane, then select "Edit in Tetra4D Reviewer™".

Open a file from the desktop

Select the file and open it:

1. Double-click to open a PRW session file.
2. Right-click another type of file with 3D content and choose Open With > Tetra4D Reviewer™ Application.

Note: If the “Open with” command is not available, select Open. Then select “This Program From a list”, and choose Tetra4D Reviewer™.

Open a file by drag and drop

You can drag CAD files and PDFs with 3D content into Tetra4D Reviewer™ to open them.

1. From the desktop or Windows Explorer, drag the CAD file or the PDF into the Tetra4D Reviewer™ document pane.
2. If the Import/Open dialog box opens (because another document is open), select options:
 - **Import:** To add the 3D model to an already open document.
 - **Open:** To open the 3D model as a separate document.
 - **Options:** To open Preferences for Reading Mode and Read/Write CAD.
 - **Always do the same action:** Sets your selection of Import or Open as the default behavior for dragging files into Tetra 4D Reviewer. If selected, the Import/Open dialog box won't appear the next time you drag a file into Tetra 4D Reviewer.

Note: You can turn off the “Always do the same” setting later by clicking it on the General tab of Preferences.

Opening PDF files with mixed content or multiple 3D models requires an additional step: selecting a method for opening from the File Tree context menu.

2D files

Tetra4D Reviewer™ can read some native 2D CAD files. The available features and the behavior for 2D files vary from the 3D files.



All 2D CAD file drawings open as separate documents, regardless of how you open them. A 2D file can contain several pages, each of which appears on a separate tab in the document pane. The structure of the 2D file appears in the Model Tree.

You can't save 2D files as session files (PRW), but you can export them as PDF or PostScript® files. Within Tetra4D Reviewer™, you can apply default views, select markups, show and hide certain entities, apply colors, and take raster snapshots.

Other behaviors depend on the specific format of the read file and the features created in the authoring application.

Import a model into an open file

You can import into existing 3D files or into a new, empty document. Starting by importing into an empty document is useful when you intend to copy and paste selected geometry from another document.




1. Click  in APPLICATION button and select an existing file to open.
Note: This first file will be the target document.
2. Click  in APPLICATION button, and select the file you want to import.
3. Select the coordinates system for the positioning of the imported model:
 - Global axis: Default option, which is the global CAD coordinate system of the active file.
 - Main axis: The activated coordinates system of the active file.

Note: The added file appears in the Model Tree under Models. In the File Tree, it appears under the format of the CAD file.

Note: You can set reading Preferences for importing some types of files. If the Options button is available in the Import dialog box, you can click it to open the related Preferences tab.

Copy and paste entities

Ordinary copying and pasting is a simple way to add parts to a document. You can copy and paste parts only from one document to another, not within the same document.

1. Click  in APPLICATION button and select the document with the parts you want to add to another document.
2. In the first document, select the parts and click  **Copy** in HOME tab.
3. Activate the target document where the parts will be pasted.
4. Click  **Paste** in HOME tab.

Note: If the two documents use different units, a message appears, asking if you want to apply the current document units to the part. If you select No, the part resizes itself so that the new units apply to the original size values.

Incremental loading of CAD files

About incremental loading

The file size of complex product files can be large. Loading only the parts of the assembly needed for your current tasks can improve performances and reduce the loading time.

You can change the loading status of individual parts and assemblies from within an open CAD file. After you close the document as a session file (PRW) or PDF and exit Tetra4D Reviewer™, links to the native CAD files are lost.

Further loading and unloading are no longer possible in the saved session files and PDFs.



Be careful about unloading parts temporarily because you want to work on another part of the assembly. If the unloaded parts are associated with any markups or dimensions, those annotations are lost when the part is unloaded.

Instead, save a copy of the document under a different filename or location, and then load and unload parts before you start working on the data.

Loading parts of a 3D file

You can set up your Preferences to load specific types of files in two stages:

- In the first phase, Tetra4D Reviewer™ loads only the structure of the assembly
- In the second phase, the components or subassemblies that you select are loaded.

This type of loading is useful to partially load a large assembly, while accessing to the full assembly structure.

Phase 1: Load the assembly structure

To do a partial loading, you have to set up reading options before you open or import the file. These options are format- specific. The incremental loading is available only for some types of files.

1. Choose Edit > Preferences, and select the Read/Write CAD tab.
2. In the File Formats list, select the format of the CAD file you want to open.
3. Click Assembly to open the Assembly Options dialog box.

Note: If the Assembly button doesn't appear, partial loading is not available for the selected format.

4. Select Load Product Structure Only, and click OK to close the Assembly Options and Preferences.
5. Open the file, using any method.

At this point, no 3D window is created, because only the structure of the assembly has been read. This structure is accessible in the File Tree where a Plus sign appears aside the read format.

Phase 2: Load selected parts and assemblies

In this phase, you load individual parts or subassemblies by selecting them in the loaded assembly structure. When the selected items are open, they appear in the model tree and in the document pane.


1. If necessary, choose Workspace > File Tree.

2. In the File Tree, click the plus signs [+] to expand the Files structure down to the loaded assembly structure.
3. Click and Ctrl-click, as needed, to select the parts and assemblies that you want to load.
4. Right-click the selected items in the File Tree and choose an action:
 - **Insert In Current Product:** Loads the items in the active document as new parts.
 - **Create A New Product:** Opens as a session file that you can save independently of the original file.

Note: The Properties command is also available on the context menu when you select only one part or assembly. Choosing Properties opens a read-only window for the selected item.

All parts appear under Models in the Model Tree, but only the loaded parts have colored icons.

Unloaded parts are listed with gray icons.




In the File Tree, a dot identifies partially or fully loaded structures. A check mark  appears above the icon for each loaded part.

Phase 3: Load and unload additional parts

Once a part at least has been loaded, you can load other parts directly from the Model Tree. In the Model Tree, right-click an unloaded part and choose Load/Unload.

Note: You can also use the process described in Phase 2 to load more parts.

Unload and reload parts

Unload commands appear in the Model Tree context menus for product definitions  and assemblies . The commands don't appear on context menus for objects .

In the Model Tree, select the components that you want to unload:

- To unload an individual instance of a part or a sub-assembly, click and Ctrl-click to select those components. Then right-click the selection and choose Load/Unload.
- To delete all instances of specific parts or sub-assemblies, select one instance of that component. Then right-click the component and choose Unload All.

To reload the parts, select them again in the Model Tree, right-click, and choose Load/Unload. Unloaded parts remain listed in both the Model Tree and File Tree, but the icons for unloaded elements are dimmed.

Note: You can distinguish between unloaded items and items that are merely hidden by examining the File Tree. The icons for hidden parts are dimmed in the Model Tree but not in the File Tree.

Expand a reloaded structure

Special circumstances apply after you actively unload parts or assemblies. When you reload that same item, the listing under the Model Tree doesn't include a plus sign [+] for expanding that branch of the Models structure.

Right-click the reloaded assembly and choose an Expand command:

- **One Level:** Shows the next tier of the assembly or part structure.
- **Component:** Expands all tiers within the selected part or assembly down to the component level.
- **All:** Expands all tiers for the selected part or assembly.

View a Properties log

1. On the File Tree, right-click the part or assembly and choose Properties.
2. Click buttons at the bottom of the window to see information:
 - **Log (Default):** Shows the complete information log for the part or assembly.
 - **Summary:** Shows limited information from the log.
 - **Missing File:** Generates a list of any not-found files for the selected assembly.
 - **Save:** Opens the Save As dialog box, for saving a copy of the log as a text file (.txt).

Creating and opening session files

Files you save in Tetra4D Reviewer are called session files and have the .prw extension. The PRW file contains all the originally read CAD data. It also saves all your work on the objects and assemblies, including any measurements, animations, or configuration views that you define.

Opening a session file from the desktop, Windows Explorer, or Start/Documents menu starts Tetra4D Reviewer.

Note: The type of information that are stored in a session file corresponds to the Reading Mode preferences that have been set to import or open the CAD file.

Save a session file

1. Click  or  in APPLICATION button.

Note: If this is the first save operation with the document, the Save As dialog box opens. The Save As dialog box contains the usual options for saving files, such as the location and filename.

Open a session files

To open an existing session file, do one of the following:

- In Tetra4D Reviewer, choose File > Open; then find and select the session files that you want to open.
- In Windows Explorer, double-click a session file.

When you open multiple files simultaneously, each session opens as a separate document.

Note: You can also drag session files from Windows Explorer into the Tetra4D Reviewer workspace.

Chapter 3: Managing entities and views

For Tetra4D Reviewer, the word “entity” refers to various kinds of information. Some entities are read from the CAD data, like physical parts, annotations, and PMI information. The entities read from the original CAD file flow into a hierarchical list in the Model Tree.

Views combines the visibility of the 3D objects with camera definition, zoom level, and other factors. Views can be saved as configurations.

Entities in 3D documents

The Model Tree positions entities under categories, according to their type. For example, markups and measurements that you create in sessions appear under Annotations. Items read from the CAD file, such as the component parts of an assembly and CAD PMI, appear under Models.

The appearances of the icons in the Model Tree give clues about entity status. Colored icons represent entities that are visible in the document pane. Dimmed icons represent hidden entities. A small funnel on a dimmed icon indicates that the entity is hidden because it is on an inactive layer. You can't create physical parts in Tetra4D Reviewer, but you can create various other types of entities. For example, annotations and bounding boxes are considered entities, but they aren't physical parts of an assembly. Typically, you create these types of entities during an analysis or publishing workflow. You can also add selected parts from one document to another by importing or ordinary copying and pasting.

Renaming an entity

1. Right-click the entity in the Model Tree and choose Rename.
2. Type a name, and press Enter.

Note: You can change entity names by two other methods. You can double-click the name in the Model Tree and type to overwrite the name. Or, you can change it on the General tab of the entity Properties.

Removing entities

You can delete unwanted entities from a document, either temporarily or permanently. All types of entities can be removed.

When you delete an entity temporarily, the entity appears in the Model Tree under the Recycle Bin for that document. Markups and dimensions associated with that entity move automatically to the Recycle Bin. You can restore the entity from the Recycle Bin and then restore its associated markups and dimensions.

Deleting an entity that is already in the Recycle Bin deletes it permanently. If the deleted entity was a physical component of the product assembly, you can import it again from the original source file, if available.

Note: Instead of removing components from a file, you can hide them. The components remain in the session file and can be easily shown again.

Delete entities from a document

1. Select the entities that you want to delete.
2. Click Delete button in HOME tab, to remove the entity and store it in the Model Tree Recycle Bin.

Restore deleted entities

1. Expand the Recycle Bin for the file in the Model Tree panel.
2. Right-click the entity, and choose Restore.

Selecting entities

About selecting entities

An entity is anything in a 3D model that you can select. Selected entities are activated so that you can change their properties, move them, change their visibility, and more.

Tetra4D Reviewer offers advanced selection features, beyond the common clicking techniques. For example, you can select entities that are hidden in the document display.

Select entities manually

- To select visible entities in the document pane, click, Ctrl-click, or hold down Shift and drag a selection box around entities.
- To select entities in the Model Tree, click, Shift-click, and Ctrl-click to select visible and hidden entities. Ctrl-clicks select non-adjacent entities. Shift-clicks select all adjacent entities within a range.

Note: The entities that can be selected in the display area depend on the active Selection Filters.


Selecting entities by attributes

It takes only one step to select all entities that share a trait. For example, you could select all solid entities or all markups. You can also select all the entities in a document in one action.

Selecting all displayed entities

1. Click **Select All** in DESIGN REVIEW tab.

Selecting entities by type

1. Click  in VISUALIZATION tab, and choose the wished type.
2. Hold down Ctrl and choose another type again to add a second type to the selection.

Note: The types include Point, Wire, Coordinates System, Planes, Surfaces, and Solids. Dimmed items on the By Type submenu indicate that no entities in the current document correspond to that type.

Selecting entities by layer

The list of layers depends on the layers that were read in the files present in the current document.

1. Click **By Layer** in VISUALIZATION tab.
2. In the Select a Layer dialog box, select a layer or Ctrl-click to select more than one layer. When you close the dialog box, entities on the selected layers are selected.

Note: The layers that have been created in the CAD application appear in Tetra4D Reviewer session files only if the import options are set to address them.

Selecting entities by color

Use this method to select components that have the same color property.

1. Click **By Color** in VISUALIZATION tab.

Note: The Select a Color dialog box shows all the colors in use in the document.

2. Click User Selection to activate the selection process.
3. Select colors:
 - Select a part in the document pane to select all parts of that color.
 - Select the name of a color in the dialog box to select all parts of that color.
 - Select one color in the dialog box and Shift-click another color name to select all parts of all colors in the range.
 - Ctrl-click to select multiple color names or multiple parts.


Note: To restart the selection process, click User Selection twice.

Selecting entities by Attribute value

1. Click **By Attributes** in VISUALIZATION tab.
2. Select an attribute in the Attribute list or type the attribute name in the box below the list to jump to that attribute.
 - Select an item in the Attributes list.
 - Type the name of an attribute in the box below the list.
3. In the Values pane, select a value for that attribute or type the value name to jump to that value.
4. Click Select.

Selecting an entity by a search

You can use the Find feature to search for some kinds of entities, such as physical parts and assemblies, balloons, and annotations.

1. Click  **Find** in Visualization tab.
2. In Find What, type part or the full name for the entity.
3. Select any options you want to apply to the search:
 - **Match Whole Word Only:** Ignores occurrences within longer words.
 - **Ignore Hidden:** Ignores entities that are hidden.

- **Match Case:** Ignores occurrences with different capitalization of text.
 - **Ignore Unloaded:** Ignores instances that appear in the Model Tree but that are not currently loaded.
4. Click Find Next or FindAll. Found entities are automatically selected.
- Note:** You can search for CAD-read Product Views and 3D PMIs. The Find feature ignores configuration names, animation actions, and bills of material.

Changing a current entity selection

After you select multiple entities by any process, you can change that selection. You can manually change the selection status of individual entities. You can deselect all entities. You can also switch the status of all visible entities in one step. This action automatically selects all previously unselected entities and deselects all entities that had been selected.

Editing an entity selection

1. In the document pane or the Model Tree, Ctrl-click each entity that you want to select or deselect.

Deselecting entities

1. Click. **Unselect All** in DESIGN REVIEW tab



You can also cancel all selections by clicking an empty area of the document pane.

Toggling the selected entities

1. Click **Toggle Selection** in DESIGN REVIEW tab.

Note: This inversion of selection is independent of the selection filters. Toggling reverses the selection status of all visible entities. Hidden entities are not affected.

Entity properties

About entity properties

The Properties dialog box contains information about a selected entity or entities. The information comes from one of two sources: the original CAD design file or default information applied by Tetra4D Reviewer. CAD information always takes priority over default information.

Some of the information in the Properties dialog box is read only. Non-interactive information appears with a gray background.


You can change some Properties information for entities. The changes you apply to the selected entities is saved with the session file.

A related dialog box, Physical Properties, has a different function from the Properties dialog box. For example, Physical Properties includes features for making calculations of area, volume, and mass. These calculations are based on information in the Properties dialog box.

Viewing and editing entity properties

Properties dialog boxes are available for all types of entities.

You can change some entity properties. Most of the editable properties are accessible in the Color/Material tab. Properties that are dimmed in the dialog box can't be edited.

1. In the Model Tree or document pane, select the entity.
2. Click  in DESIGN REVIEW tab.
3. Change the properties on each of the four tabs, as needed.
4. Click Apply to apply the change to the entities without closing the Properties and examine the entity appearance in the document pane.
5. Continue adjusting settings, as needed.
6. When you finish, either close the Properties or click Apply and select the next entity that you want to edit.

Note: Configurations, Coordinate Systems, and the file itself do not have Properties dialog boxes. You can open an information display for the file itself by right-clicking the file in the Model Tree and choosing Document Information.

Adding and deleting user-defined materials

Tetra4D Reviewer includes several default material definitions, such as Aluminum, Copper, and Steel. Each material definition incorporates specific density, color qualities, and transparency settings. When you apply a Material option to an entity, the entity assumes all of the material characteristics. You can define, save, and apply custom materials. These user-defined materials appear on the Properties dialog box Material menu and are saved with the session file.

After you apply a material, you can adjust the settings for that entity. Those changes apply only to the selected entity and do not alter the saved definition for the material. If you want to change the material definition, define a new material with the settings you want and delete the old material.




The attributes you choose apply only to the selected entities. If you want to make this set of attributes available for future use, create a user-defined material before you close the dialog box. User-defined materials are saved with the session file.

Adding a user-defined material

1. Select an entity, and click  in DESIGN REVIEW tab, and select the Color/Material tab.
2. Specify the Colors, Transparency, and Density settings for the material.
3. Choose User Defined for Material and click the plus sign .
4. Type a name for the custom material in the Material Name dialog box, and click OK.



Deleting a user-defined material

1. In the Material pop-up menu on the Color/Material tab, choose the user-defined material that you want to delete.
2. Click the minus sign .


Note: The minus sign button is not available for the default materials because they can't be deleted.

Edit a user-defined material

Once you create a user-defined material, its properties are permanent and can't be edited. However, you can easily define a new material based on the existing material properties.

1. Select a part and click  in DESIGN REVIEW tab.
2. On the Color/Material tab, select the Material definition that you want to replace, and change the settings, as needed.
3. Click the plus sign , type a name to define the new material, and click OK.

Note: Reusing the name of an existing user-defined material does not overwrite it or alter its settings. Both instances of the name appear on the Material menu until you individually delete them.

4. In Material, select the user-defined material that is now obsolete, and click the minus sign  to delete it.

Properties options

Four tabs organize the Properties information. Some contain read-only information and some include interactive options.

General tab

- **Name:** Name of the item, as shown in the Models structure.
- **Text area:** Descriptions of attributes, if any exist. Look here for information about geometric characteristics and the number of faces, for example.

Color/Material tab

- **Material:** Options for predefined and user-defined substances, such as copper, steel, and aluminum.
- **Simple Color:** Limits the available color options to Diffuse.
- **Material Colors:** Makes options available for individual Diffuse, Ambient, Emissive, and Specular colors and for Shininess.
- **Transparency:** Sets the opacity of the selected entity. Lower percentages produce more transparent views of the entity. At 0% Transparency, the entity is invisible.
- **Density Sets:** the physical property for the selected material, using the specified units of weight and volume.

Position tab

- **Matrix:** Indicates the translation and rotation information for each axis (Tx, Ty, Tz, and Rx, Ry, and Rz in relation to the Tx, Ty, and Tz values).
- **Sx, Sy, and Sz:** Indicated the values of scaling.
- **Restore:** Resets position options.
- **Is Mirror:** Indicates if a symmetry is applied to the entity.
- **Is Unit:** Indicates if transformations are applied on the entity.

Source tab

The Source tab provides information related to the original CAD file. The tab lists the original filename, the design application used to create the file, and the units of measurements.

Changing appearances of entities

Hiding and showing entities


You can change hide or show any individual entities in the document pane. Visibility does not affect the structure of the Model Tree. The Model Tree icons for hidden (invisible) entities become light-gray. Icons for visible entities are gray.

You can change the appearance of entities in several ways:


- Altering visibility by hiding and showing selected entities.
- Adjusting the transparency of selected objects.
- Setting an object to phantom mode.

Isolating an entity


Isolating will hide all unselected objects so that only the selected ones remain visible.

1. Select the entities that you want to isolate.
2. Click  **Isolate** in HOME tab.

Hiding or showing individual entities

1. Select the entities that you want to hide or show.
2. Click  **Hide/Show** in HOME tab.

Toggling all entities between hidden and shown



1. Click  **Toggle Hide** in HOME tab.

Displaying all entities

1. Click  **Show all** in HOME tab.

Hiding or showing all annotations by type

Session annotations are 3D markups and dimensions you create in Tetra4D Reviewer. The context menu Hide/Show command changes the visibility of annotations.

1. Click  in VISUALIZATION tab to hide all PMI markups.
2. Click  in VISUALIZATION tab, to hide all dimensions and markups created in Tetra4D Reviewer.


Note: Individually hidden 3D PMIs are dimmed in the data tree. When you choose Hide 3D PMI, the PMI markup icons aren't dimmed even though the markups aren't visible.

Changing the transparency of entities

Changing transparency adjusts the opacity of an entity so that you can see it and see through it. You can change transparency temporarily, to get a quick glance at the underlying parts or to do other work. Temporarily transparent objects aren't selectable.

You can also change the Transparency property. This type of transparency leaves the entity selectable. The entity remains semi-visible until you actively reverse the transparency setting. Phantom mode applies a preset level of transparency to the selected entity. You can't select an entity that is in Phantom mode.

Note: Phantom mode is a different type of transparency: It is persistent until you actively reverse it and the objects are unselectable.

1. Select the entities.
2. Click  in DESIGN REVIEW tab.
3. On the Color/Material tab, drag the Transparency slider or enter a value for the percentage of transparency.
4. As needed, click Apply and repeat the preceding steps to set other entities at a different percentage of transparency.
5. Click OK to apply the changes and close the Properties dialog box.

Apply transparency temporarily

This technique applies transparency without requiring you to select entities first. While a part is temporarily transparent, it is unselectable. However, you can select other parts, take dimensions, and do other work without reversing the temporary transparency.

1. Move the pointer over the part that you want to make transparent.
2. Press H key.
3. As needed, press H again to make more parts transparent temporarily.

Note: You reverse temporary transparency actions one step at a time by pressing S.

4. Click anywhere in the background to remove the temporary transparency and restore selectability.

Phantom mode

In Phantom mode, selected parts appear semi-transparent and can't be selected. You can apply this mode to any geometric entity.



Displaying entities in Phantom mode


1. Select the parts in the Model Tree or document pane.



2. Click  Phantom in HOME tab.

Note: The phantom mode remains in effect until you choose View > Phantom again to exit Phantom mode.






Adding and removing entities from Phantom mode

1. Click  Phantom in HOME tab.
2. Change the selection, as needed:
 - a. Ctrl-click to add or remove parts from the current selection.
 - b. Click outside the current selection to deselect, and then make a new selection.
3. Click  Phantom in HOME tab.


 When adding entities to a set of entities that are already displayed in phantom mode, select those entities first. Then choose View > Phantom twice or click the Phantom toolbar

 button  twice. These actions reapply the phantom mode to both the already displayed in phantom mode entities and to the ones you selected later.

Changing the display mode

1. Click  in VISUALIZATION tab and select the rendering mode among the 4 following renderings:
 - **Solid:** Click  Solid in VISUALIZATION tab, shades in the entire surface area of the parts.
 - **Solid outline:** Click  Solid Outline in VISUALIZATION tab, shades in the entire surface area of the parts and outlines the edges of the parts.
 - **Wireframe:** Click  Wireframe in VISUALIZATION tab, outlines the edges of the parts.
 - **Illustration:** Click  Illustration in VISUALIZATION tab, shows the entire surface area of the parts in flat white and outlines the edges of the parts.

Perspective

1. Click  Perspective in VISUALIZATION tab, adjusts Solid, Wireframe, or Illustration mode so that the geometry resembles a physical view.


Accessing the PMIs

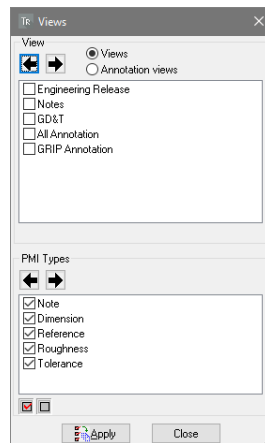
The Product Manufacturing Instructions (PMIs) read from CAD files can be numerous and of various types. PMIs are often associated with Product Views read from the CAD files. Product Views appear in a folder under Models in the Model Tree.

Product Views are like Configurations that you define and save in Tetra4D Reviewer. A Product View sets the orientation, zoom level, and visibility of the 3D objects and 3D PMI markups.



Views created in CAD can also be associated with the 2D environment, as views defined in a drawing.

Activating a PMI Product View

1. Click  in VISUALIZATION tab opens the following dialog:



2. Select Views or Annotation Views (if available).
3. Under PMI Types, select the types of PMI markups you want to show: Notes and References (if available).
4. Under View, select the view you want to see:
5. Click the arrows to step through the list, which automatically applies each Product View as you progress.
6. Click a check box to select a view, and click Apply.
7. Close the Views window.

Note: You can select or deselect all PMI Types with the Select All  and Unselect All  buttons.


Moving parts and assemblies

About moving components

You can change the positions of individual parts and assemblies in the document. This feature is especially useful if parts from different source files appear superimposed at the same location. The new positions are associated with the currently active configuration. When you switch to a different configuration, moved parts appear in their original positions. When you reactivate the configuration with the moved parts, they jump back into their moved locations.

Note: You can move all parts simultaneously in different directions by creating an exploded view. Exploded views can also be saved as part of a configuration.

Activating the “Drag and Drop” displacement control

1. If a locked configuration is active, switch to an unlocked one or create a new configuration.
2. Select the parts to be moved.
3. Click  in DOCUMENTATION tab.

Note: The displacement manipulator appears around the selected parts.

4. Use the manipulator to move the parts as described below.

Moving parts freely with direct displacement controls

1. Click the yellow dot that appears in the center of the manipulator.
2. Without pressing the mouse button, slide the manipulator to move the selected parts in any direction.
3. Click once to release the parts in this new position.

Moving parts along axes

1. Click one axis of the manipulator.
2. Without pressing the mouse button, slide the manipulator along the direction of the arrow.

Note: As you slide the pointer, a small box displays the value of the displacement along the axis.

3. Click once to release the parts in this new position.
4. As needed, continue to move and rotate the parts using the other activated controls.

Rotating parts along axes

1. Click one rotation arrow of the manipulator.
2. Without pressing the mouse button, slide the pointer around within the visible circle to rotate the part.

Note: As you slide the pointer, a small box displays the angle (in degrees) between the original position and the rotated position.

3. Click once to release the parts at the rotated position.
4. As needed, continue to move and rotate the parts using the other activated controls.

Moving parts with the Move feature

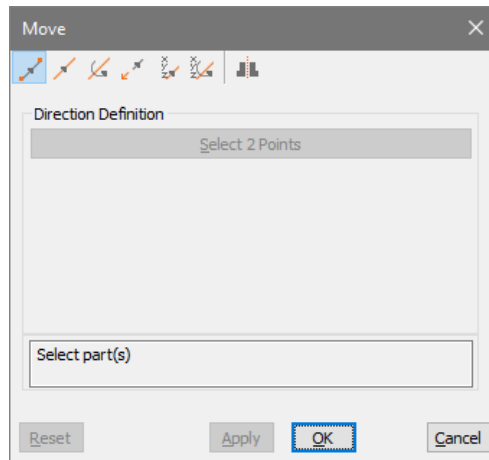
You can define displacements through several methods available in the Move controls window. The Move feature is available only when an unlocked configuration is active.



As you work with the Move feature, notice the text area near the bottom of the window. This area displays context-sensitive information about actions that are required as you complete steps in the process.

1. If necessary, activate a new or an existing unlocked configuration.

2. Click  in DOCUMENTATION tab to activate the feature



3. Select a method at the top of the Move controls dialog.
4. In the Direction Definition area, select a Method if available: Main CS, 2 Points, Edge, or Plane.
5. Drag the Displacement slider or type a value to set the displacement value

Note: If you aren't satisfied with your selections, click Reset to start over from the original position.

6. When you are satisfied with the position change, continue working:
 - Click Apply to save the results, and continue to move other parts.
 - Click OK to save the results and close the Move controls window.

Methods to move parts

Move from point to point

- Moves the parts along a direction that you define by clicking two reference points in the document pane.

Move along axis

- Moves the parts along an axis defined for the selected Method option.

Rotate around axis

- Moves the parts along an axis defined for the selected Method option.

Move from axis to axis

- Moves the parts in relation to your selection of an initial coordinates axis system and a target coordinates system axis.

Move with coordinates

- Moves the parts according to the x, y, and z coordinates that you specify.

Rotate with angles

- Moves the parts according to angles you specify to the x, y, and z axes.

Mirror

- Creates a mirror-image copy of the selected part, naming the new version “SYM [part name].” The part appears on the opposite side of a symmetry plane that you define.


Methods to define a displacement direction

- The Method menu is available for Move Along Axis and Rotate Around Axis. The menu also appears for Mirror, but only Main CS and Plane are listed.

Main CS

- Moves along the active axis of the designated Main CS.

2 Points

- Sets the direction by two points that you select in the document pane. The Accept/Reject  pointer appears as you select points so that you can confirm the position or toggle to another possible location.

Edge

- Sets the direction parallel to an edge that you select in the document pane.


Plane





- Sets the direction along a plane that you select in the document pane.


Note: Clicking Select Direction cancels any previous reference items and restarts the process of setting a direction.

Stack up displacements for multiple parts

Stack Up option will simultaneously move multiple parts by different amounts of displacement. The order in which you select the parts determines how far they move. For example, if the first part moves by x amount, the second part moves $2x$, the third part moves $3x$, and so on.

1. Click  in DOCUMENTATION tab.
2. Select a type of action, action options, and displacement, as usual.

Note: StackUp is available only for move along axis , rotate around axis , rotate with angles , and Move with coordinates .


3. Select Stack Up .
4. Ctrl-click to select parts, selecting the part you want to move least first. Continue selecting in order of increasing multiples of the displacement.
5. Set the displacement value, and click Apply or OK.

Note: The Stack Up feature is also available for custom animations.

Restore the initial positions of moved parts

After you move one or more parts, you can restore them individually to their previous positions.

Note: If you locked the configuration after moving the part, unlock it. When it is unlocked, you can restore parts to earlier positions.

1. Select the moved parts.
2. Click  in DOCUMENTATION tab.
3. When a message appears asking you to confirm the repositioning, click OK.

Note: If you have moved a part several times, you can only restore the most recent previous position.

Locate entities visually

Complex assembly displays can make it challenging to locate specific parts and other entities, such as markups and balloons.

One method of identifying an entity is to select it. Selecting highlights entities in both the document pane and the Model Tree.

Another method is to use the Focus and Show Me features. These features are especially useful when you want to locate an object without selecting it.

You can also locate an entity by searching it.

Show me (Highlight an entity)

Highlighting draws a red, translucent sphere around the image of the selected entities in the document pane.

1. In the Model Tree, right-click the entity and choose Show Me. Clicking anywhere in the document pane removes the highlight.

Focus on an entity

Focusing does two things: It adjusts the zoom level to the maximum level at which you see all selected entities. It places the center of rotation to correspond with the center of a bounding box for the selected entities.

1. In the Model Tree or document pane, select one or more entities.
2. Right-click a selected entity and choose Focus.

Note: The zoom level and center of rotation set by focusing remain in effect until you change them.

Creating geometric entities

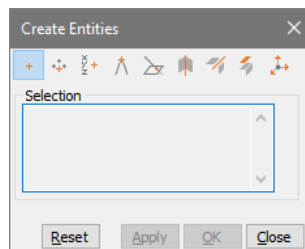
You can generate some simple entities based on existing geometry. For example, you can create a reference point, an intersection, or an axis, or define a new coordinate system.

These entities are permanent and appear in the Model Tree. The location of the entity in the Model Tree and the type of icon appearing with it depend on the entity type. You can use these entities for other operations, such as measuring.

Note: Created entities are not solids or components in the 3D assembly. Physical parts and assembly entities must be created in a CAD application and can be imported into a session file.

Creating an entity


1. Click  in DESIGN REVIEW tab.



2. Select the type of entity that you want to create.
3. Follow the cues that appear displayed in the status bar at the bottom of the application window to make selections in the document pane.
4. Create the entity:
 - Click Apply to create the entity and leave Create Entities open.
 - Click OK to create the entity and close Create Entities.
 - Click Close to close Create Entities without creating an entity.
 - Reset to deselect and start creating a different entity.

Types of created entities

The Create Entities control window provides tools for creating nine types of entities. These entities are geometrics.

In the same way as with measurement tools, several Create Entities tools ask you to confirm the locations you select. The pointer changes to the Accept/Reject  pointer, indicating that you can right-click to reject the proposed location and see another proposed location.

Creating a point from selection

- Creates a Point entity at a position you select after selecting this tool.

Creating a point as a center of a circle defined by 3 points

- Creates a Point entity from three points you select in the document display after selecting this tool.

Creating a point by coordinates



- Creates a point from the values you type in X, Y, and Z, using the CAD coordinates system.

Creating a point as an intersection of segments



- Creates a point where two selected line segments cross.

Creating a point as an intersection of a face and a segment



- Creates a point where a selected line segment crosses a selected face.

Creating an axis from a cylinder



- Creates a wireframe entity from the axis of a cylindrical part or entity.

Creating a curve as an intersection of faces



- Creates intersecting curves from faces that you select.

Creating an entity by extraction



Retrieves information from the selected items as entities under an entity named “[PARTNAME] Extraction.” Extracted entities can be based on any combination of the three Selection options:

- **Basis Surface:** Generates the original surface for the selected face of the solid or the surface as it was before any trimming operations.
- **Face:** Generates a copy of the selected face from the solid or the surface.
- **Wire:** Generates subcurves and characteristic points from a selection of composite curves. Generates original and characteristic curves for a selected face. Original curves refer to trimming curves. Characteristic curves refer to elements such as the axis of a cylinder.

Creating a Coordinates System

Defines a system of coordinates based the selected reference option and elements you select in the document pane. The new coordinates system appears as Axis [#] under the Coordinates System section of the Model Tree.

Options for creating coordinates systems

Normal to surface

- Sets the origin at the point you click. Sets the Z axis normal to the surface at the origin. The orientation of X and Y axes are set automatically in relation to iso-curves of the surface, and can't be edited.

Points

- Sets the origin at the first point you click. Your second click sets the X axis. Your third click defines the Y axis and XOY plane.

Note: The vector defined by the points 2 and 3 isn't necessarily perpendicular to the first vector defined by points 1 and 2.

2 Edges

- Sets the origin at the intersection of the two edges. The first edge you select sets the X axis. The second edge defines the XOY plane.

Note: The two selected edges aren't required to be perpendicular, but they must be linear, nonparallel, and secant.

Curve + Point

- The first point clicked sets the origin and the Z axis. The second click sets the X axis.

Note: Cylinder parts present a special case for the Curve + Point method. Selecting the cylinder sets the cylinder axis as the Z axis. The second click sets the position of the origin and the direction of the X axis.

From CS

- Creates a copy of the selected coordinates system.

Naming rule for created entities

Tetra4D Reviewer assigns default names to created entities. The default name varies with the type of entity created. Some names include a number that indicates the order in which it was created. Numbers in parentheses following the entity name indicate how many individual entities are included in the created item.

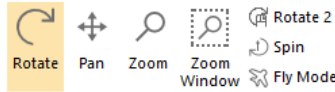
Different created entities can have identical names. You can rename entities to make it easier to tell them apart, as needed.

Operating with the model in the 3D window

About 3D controls in the 3D window for the document



You adjust the visualization of the 3D entities using 3D Control tools, which are accessible in the HOME tab.

The selected 3D tool determines what happens when you drag the mouse within the 3D window.




Rotate the 3D model

This control rotates the model around a specific point in the 3D window. The Rotate tool is active by default.

1. Click  in HOME tab.
2. In the document pane, set the center of rotation:
 - Drag from any position in the background to rotate around the center of a bounding box that encloses all visible parts.
 - Drag from a position on a component to rotate around that starting point, as indicated by across hair  that appears.

Lock a rotation axis

By default, the rotation can be performed freely around any axes.

1. Click  in HOME tab.
2. Using the right mouse button, start dragging to rotate the 3D objects.
3. As you drag, press the spacebar repeatedly to lock a specific axis:
 - One time Locks the X axis.
 - Two times Locks the Y axis.
 - Three times Locks the Z axis.
 - Four times Unlocks all axes so that you can rotate freely around any axis.

A double arrowhead in the View CS display in the lower left corner of the document pane appears as a reminder of the locked axis. The axis remains locked after you stop dragging, even if you activate a 3D Controls tool.

You unlock all axes by repeating this process, pressing the spacebar until no axis has a double arrowhead.

Set a fixed center point for rotating and spinning

By default, the center of rotation is automatically set as the center point of the bounding box calculated including all the visible entities.

You can set a different center of rotation that does not change as you manipulate the view in different ways.

You can set a fixed center of rotation at any position on any part but not on the background or in empty space between parts.

1. Click  in HOME tab.


Note: The Rotation Center tool is available only when one of the Rotate tool or Spin tool is active.

2. Click a point on one visible part that you want to serve as the center of rotation.

Note: A white cross hair marks the position in the display, even if you switch to a different type of tool.


3. Drag to spin or rotate the 3D object.

Note: Notice that the Rotation Center tool remains active as you work, even if you switch back and forth to the 3D Controls tools.

4. Push  again to cancel the fixed center.


Spin a 3D model

Spinning rotates in relation to the Z axis of the View CS. It is especially appropriate for turning images set on a fixed horizontal plane, such as buildings.

1. Click  in HOME tab.
2. Drag in the document pane.

Pan horizontally and vertically


Panning changes the position of the 3D objects in the 3D window while keeping the orientation.

1. Click  in HOME tab.
2. Drag the 3D object left, right, up, or down.


Zoom in and zoom out

Zooming in enlarges or shrinks the view in the document pane.


Zoom in or out


1. Click  in HOME tab.
2. In the document display, drag up to zoom in and drag down to zoom out.

Note: Clicking the Zoom tool and dragging horizontally do not change the zoom level.

 If your mouse has a wheel button, you can zoom without selecting the Zoom tool. Instead, turn the wheel forward and backward to change the zoom level. Wheel-button zooming centers around the current pointer position in the document pane.

Zoom window


1. Button  in HOME tab.
2. In the document display, draw a rectangle to define the zoom area.

 You can switch temporarily to the Zoom Window tool by holding down Shift and dragging. When you stop dragging and release the Shift key, the previously active tool is reinstated.

Fit the 3D model into the display

The Fit Visible feature shows the model at the maximum magnification that keeps all visible components in the document pane window.

1. Click  in HOME tab.

 Double-clicking the document display with any tool also changes the view to Fit Visible. In some contexts, you can right-click and choose Fit Visible to show all visible parts.

Activating document views


Tetra4D Reviewer offers several features to change views.

- **Default views** Align 3D objects in various orientations to the axes of the active view coordinates system (View CS).
- **Camera** Aligns the view in relation to a selected element in the geometry.

Configurations bear a similarity to default views in that they quickly shift the view. You can create custom configurations, lock or unlock them, and use them in later work sessions.

Set a default view

Applying a default view leaves the zoom level and other viewing characteristics unchanged. The default views appear both on the Views menu and in the Default Views toolbar.

1. Click  in HOME tab, and one of the following:
 - **Top, Front, Back, Bottom, Left, or Right:** Sets the view normal based on the active View CS.
 - **Iso** Sets the 3D objects in a standard isometric view, based on the active View CS.



The Smooth Transitions preferences can help you visualize how the view changes. Selecting these preferences animates the 3D objects as you apply built-in views. If these preferences are deselected, the view changes in one step.



Set the camera

A *camera setting* refers to a combination of zoom level and orientation. Some camera settings are independent entities. Other camera settings change the view according to the entity that is currently selected.

1. If appropriate, select one or more entities.



Set the appropriate Selection Filters for the type of selection you want to make, such as a curve, face, or axes.

2. Click  in HOME tab, then click  **Set Camera** and choose one of the following settings:
 - **From Selection** Aligns the view to the currently selected entity or to the next entity that you select.
 - **Restore Config.** View Resets the orientation to either the initial settings for the current configuration or the initial view when the file opened.
 - **From Curve/Edge** Sets the view perpendicular to the next curve or edge that you select.
 - **From Face** Sets the view parallel to the next planar face that you select. Or, if the face is not planar, sets the view normal to the face at the point that you select.
 - **From Coordinate Axis** Sets the view perpendicular to an axis (X, Y, or Z) that you select next.
 - **Flip** Rotates the view to the opposite side of the assembly or part.


Saving views as configurations

About configurations

A configuration is a view of a 3D model. A configuration includes basic characteristics, such as zoom level, orientation and positions of the 3D objects, visibility of entities, lighting, and background color. Configurations also apply specific types of features created in that configuration. These features include bills of material, balloons, and sections.

When you start a Tetra4D Reviewer session or open a new document, the default, Initial Configuration, is active. Unless Initial Configuration is locked, objects open as they appeared at the end of your most recent work session.




Creating a configuration

1. Adjust the orientation, zoom level, visibility settings, and other characteristics for the view, as needed.
2. In the Model Tree, right-click Configuration and choose Add Configuration, or click  in DOCUMENTATION tab.
3. Type a name for the new configuration.

The saved view becomes the active configuration.

Activating a configuration

1. In the Model Tree, right-click the configuration and choose Activate Configuration.

 You can also use the buttons  and  to step through configurations, or just double-click a configuration in the Model Tree to activate that view.

Renaming a configuration

1. In the Model Tree, right-click the configuration and choose Rename.
2. Type a new name.

Deleting a configuration

1. In the Model Tree, select the configuration.
2. Click Delete from the context menu, or press Delete key.

Locking or unlocking an existing configuration

1. In the Model Tree, right-click the configuration and choose Lock or Unlock.

About locked configurations

When a defined configuration is active and unlocked, any changes to the zoom level, orientation, and so forth, are incorporated into the configuration definition. Locking a configuration prevents changes from altering the configuration definition. Each configuration is individually locked and unlocked. You can lock the default Initial Configuration and custom configurations.

You can change the zoom, rotate, hide parts, and make other view adjustments when a locked configuration is active. When you switch to a different configuration and then back to the locked one, the locked configuration appears in its original state.

Some features are disabled or can't be applied to a locked configuration:


- Positioning tools
- 3D Markups
- Animation controls

Locking a configuration doesn't affect your ability to create balloons and bills of material in that configuration.

Hiding and showing entities by layers and filters

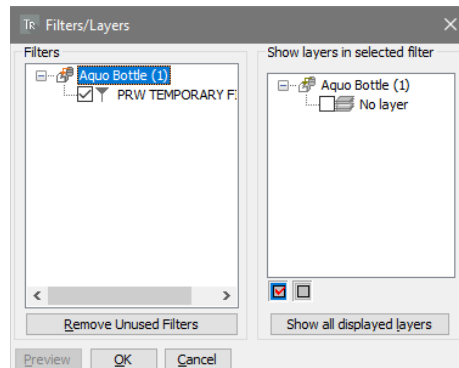
Layers and filters are CAD structural features created within certain file formats. When the CAD file is imported into Tetra4D Reviewer, the filters and layers are imported too. You can't modify the filters and layers in Tetra4D Reviewer.

However, you can use them to select parts or to change the view of the 3D objects.



The Model Tree for a part on a hidden layer isn't colored and includes a funnel shape . The icon serves as a reminder that you can't use the ordinary Hide/Show command to display the part.

Hiding and showing parts by layers

1. Click  in VISUALIZATION tab



Note: Immediately after the Filters/Layers window opens, you can click Remove Unused Filters to remove the filters that don't refer to any layer.


2. Select and deselect layers until only the layers you want to appear are selected, using any combination of methods:
 - Click the layers check boxes to select and deselect individual layers.
 - Select Filters in the left pane to select layers associated with that component part or assembly.
 - Click Select All  to select all layers.
 - Click Unselect All  to deselect all layers.
 - Click Show Layers in Selected Filter to select layers currently displayed in the document pane.
3. Click Preview, and review the results of your selection in the document pane and Model Tree. If necessary, adjust your layer selections and click Preview again until you are satisfied with the results.

Note: The Preview is temporary. If you close the Filters/Layers dialog box without clicking OK, the view resumes its previous condition.

Showing a part on a hidden layer


The usual Hide/Show command doesn't affect the visibility of parts on hidden layers. Layer visibility can be reversed only in the Filters/Layers dialog box.



1. Click  in VISUALIZATION tab.
2. Select the layer that includes the part you want to bring out of hiding.
3. Click Preview to confirm the layer visibility settings, make any adjustments, and click OK.

Fly through mode

Activating the Fly through mode

1. Click  **Fly Mode** in HOME tab.
2. Press and hold the following keys to control Fly Mode direction and speed:
 - **Q and D** Move left and right
 - **Z and S** Move up and down, left and right (parallel movement without rotation)
 - **A and E** Rotate left and right
 - **R and F** Forward and backward.
 - **Spacebar** Stop movement.

Note: You can also use your mouse to move up, down, right, and left. Use the center mouse wheel to increase and decrease Fly Mode speed.

3. Release the key to stop the fly through motion. You can also use the mouse wheel to increase and decrease fly through speed.

To see a complete list of keys to use in Fly Mode, open the Preferences dialog box, and click the Fly Through tab.

To exit Fly Mode, press ESC or right-click in the model and choose End Fly Mode.

Managing Fly through mode views

With Fly Mode, you can view a model at different angles and magnifications. You can save these views, or positions, to use later. You can also constrain Fly Mode movement to a horizontal plane so that altitude doesn't change as you move. This feature is useful for moving through architectural models.

Saving a Fly through position

1. In Fly Mode, position the model as you want to save it.
2. Right-click the model, and then choose Position > Store Current Position.
3. Type a name of the position in the dialog box, and click OK.

Modifying a saved position

After you create a position, you can rename it, activate it, or delete it.

1. Right-click inside the model and choose Position > User Positions.
2. Select a user position, and then select what you want to do with the position.
3. Click Close.

Returning to the default position

The default the position in Fly Mode is <Automatic> and always corresponds to the Isometric view.

1. In Fly Mode, click in the document pane, and then right-click in the model.
2. Choose Position > Activate Default Position.

Setting a reference plane for Fly through

You can set a horizontal reference in Fly Mode to retain the same height as you move through a model. This feature is called walk through mode. Altitude doesn't change in walk through mode.

Note: You can change the default height and reference plane settings on the Fly Through tab of the Preferences dialog box.

1. In Fly Mode, right-click inside the model.
2. Choose Reference, and then select the reference type to use:
 - **Plane** Uses a horizontal plane as the reference point. The camera position is constrained to the reference plane and the height is specified in the Fly Through preferences.
 - **Main UCS** Uses the main coordinate system to define a new reference. When selecting this option, the reference plane is the one that is perpendicular to the active axis (X, Y, or Z). The camera is located at the specified height, which is constant.
 - **Curve or Axis** Constrains movement to the length of the curve. The axis is selected automatically by selecting a cylinder face. For an axis or line, you can move anywhere using the keyboard controls (except A and E).
 - **Free** Deactivates the previously set reference and restores default Fly Mode behavior.
 - **Invert Direction** When Plane of Main UCS is used, Invert Direction changes the orientation of the reference to be opposite the current orientation.

Changing the reference plane

1. In the Preferences dialog box, click the Fly Through tab.
2. In the walk-through section, do the following:
 - To change the height of the horizontal plane, set the height value and corresponding measurement unit.
 - To hide the reference plane, clear the Display Plane option in the walk-through section.

Restoring the original reference plane

Apply a horizontal reference plane to restore an initial state.

1. In Fly Mode, click in the document pane and then right-click inside the model.

Note: The model position remains the same, but the direction moves horizontally.
2. Choose Horizontal from the context menu.

Graphic rotation controls

The graphic rotation controls are displayed in the lower-left corner of the model. Use the graphic controls for quick access to user-defined positions and rotation functions.

1. In Fly Mode, right-click a part of the model to be the reference point for the rotation. The graphic rotation controls are displayed in the lower-left corner of the model.
2. Select a control to change the view of the model:
 - Click the triangle to view a list of user positions, and then select a position from the list.
 - Click a square to make a 90-degree rotation around the reference point selected in step 1. For example, click the right square to make a 90-degree rotation around the vertical axis to the right. Click the same square again to turn around the model, keeping the same reference point. The back square displays the back view of the model.

Note: The squares are visible only when the mouse is over a model part when you right-click. The point at which you right-click is used as the center of the rotation.

Defining a target point for Fly through

In a complex assembly, defining a target point keeps the focus on that point as you move through the model. When defining the target point, the current position is used as the starting point. The final position is the target point.

1. In Fly Mode, right-click inside the model, and choose Target Point.
2. Select a point on the model.

Note: The red beam represents the distance between the two points.

3. Right-click in the model, and then click Play.
4. Press and hold the rotation keys A and E to move the model.

Note: Use the controls to define the speed and camera orientation. The camera follows the red beam, but the view direction can vary. When the target point is reached, the movement stops.

5. To stop the movement, do one of the following:
 - Right-click in the document pane, and then choose Reference > Free.
 - Define a new reference.
 - Define a new target point, and make an empty point selection (by clicking in an area of the window where no part is displayed).

Note: Because the starting point is fixed, you cannot change the height and the lateral position of the rotation. The Fly Mode direction keys Z, S, Q, D are not active.

Chapter 4: Analyzing product design

The tools accessible in the Design Review tab are dedicated to design-review work. Calculations for analysis can be precise, based on exact geometry defined by the original CAD file. Or, calculations can be approximate, based on the level of detail set for the display.

Calculating bounding boxes


A bounding box is a rectangular wireframe entity that encloses the selected components. The active coordinates system (Main CS) determines the orientation of the bounding box faces. Each segment of a bounding box is also an entity.

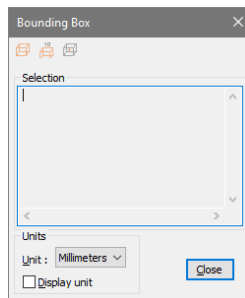
Bounding boxes are calculated from geometric elements. You can make bounding boxes of a part, multiple parts, assemblies, or subassemblies.

Bounding boxes appear under the Models branch of the Model Tree structure. You can hide or show bounding boxes or individual segments, as needed. You can also rename them.

Each segment of a bounding box is an entity. Tetra4D Reviewer gives default names to the bounding boxes and segments. You can change the names in the Model Tree after you create the bounding box.

Calculating a bounding box

1. Click  in DESIGN REVIEW tab, and the following dialog box appears:






2. Select options for displaying units with the bounding box wireframe:
 - Choose the measurement system in the Unit menu.
 - Select Display Unit to display the units of measurements with the dimensions, or deselect it to see the dimension values alone.

Note: The unit options apply only to bounding boxes created with the middle button, Create Bounding Box with Dimension.

3. Select the parts or assemblies that you want to enclose in the bounding box.

Note: The Selection area displays the volume and X, Y, and Z dimensions of the bounding box.

4. Click to create bounding boxes:

- Create bounding box , to create a wireframe bounding box.
- Create bounding box With Dimension , to create a wireframe bounding box that displays each dimension measurement.
- Create Gravity Center , to create a point entity for the center of gravity of the bounding box. Each click creates a new bounding box.



You can create bounding boxes for other parts without closing the dialog box: Select a different part, parts, or assemblies in the document pane or Model Tree. Then click buttons for the types of bounding boxes you want to create.

Accessing the bounding box information

1. In the Model Tree, select an existing bounding box or bounding box segment.

2. Click  in DESIGN REVIEW tab.

Note: Information about the selected element appears in the Selection area.

3. As needed, choose an option on the Unit menu to change the units of measurements for the Selection area.

4. Select another bounding box or segment to review it or close the Bounding Box controls window.


Note: You can select and copy information in the Selection area, but it can't be edited.

Measuring and dimensioning

About measurements and dimensions

Dimensions are annotations that represent distances, angles, or coordinates in the 3D window. These entities are saved as part of the session file. They appear in the Model Tree and in the document pane. You can select, hide, show, and view the properties for a dimension.

A measurement is like a dimension except that they appear only temporarily. When you click anywhere in the document pane or Model Tree, the measurement disappears.

Measuring and dimensioning involve selecting specific geometries rather than entire parts. Selection filters and the Accept/Reject  pointer help you select exactly the elements that you want to use as your start and end points.







Selection filters for geometric selections

The selection filters control the snap-to behaviors when you click a point in the 3D display area. When a specific selection filter is inactive, clicking ignores that type of geometry. When the same selection filter is active, clicking snaps to and selects the nearest geometry of that type. When all selection filters are active, clicking selects any type of geometry close to the location you click. For example, if you want to select only edges, you can disable the selection filters for points, faces, and solids.

Selection filters sometimes adjust automatically. For example, when the Radius measurement tool is active, the Points, Faces, and Solids filters are deselected. The Edges filter remains active so that you can select only geometries appropriate for a radius measurement.

Types of selection filters

1. In DESIGN REVIEW tab, activate one or several of the selection button:

-  **Points** : Snaps to vertices and points.
-  **Edges** : Snaps to edges, curves, and coordinates systems axes.
-  **Faces** : Snaps to faces from solids, surfaces, and planes.
-  **Solids** : Snaps to solids.
-  **Collapsed Tree Nodes** : Collapsed branches of the Model Tree can be selected. The full collapsed branch is selected if one part from that branch is selected in the display area.
-  **Highlight Selection** : Highlights entities as the pointer moves over the display, before you click to select.


Note: Selection filters don't affect selections in the Model Tree.

Automatic changes in selection filters

Selection filters sometimes adjust automatically as you perform specific tasks. These automatic adjustments leave on the filters that are appropriate for that task and turn off the other filters. For example, when the Radius measurement tool is active, the Points, Faces, and Solids filters are deselected. The Edges filter remains active so that you can select only geometries appropriate for a radius measurement.

The selectable entities can be explicit or implicit. Implicit entities are characteristic entities extracted from another entity. For example, the center point for a circular edge is an entity that is implicit for the edge.

Confirming geometric selections with the Accept/Reject pointer








When you select geometries in the display rather than parts, clicking can be ambiguous. For example, clicking close to the circular edge of a cylindrical solid can select various geometric objects. It can select a point, a face, the circumference, or an axis running vertically through the cylinder. In situations like that example, a Accept/Reject  pointer icon appears. This icon gives you the opportunity to use the left and right mouse buttons to accept or reject a series of possible selections. When you click the right mouse button, the first possibility is rejected and the selection changes to the next possibility. Clicking the right mouse button repeatedly cycles through all the possible selections and back to the first one.

When the element that you want to select is highlighted, clicking the left mouse button confirms the selection and restores the default pointer icon.


The active selection filters influence which types of geometries are shown as possibilities.

Measurement tools


The types of measurement correspond to the tools in the Dimension control window.

- **Distance**  measures on a straight-line between two elements that you select, such as points, lines, surfaces, and axes.
- **Radius**  measures the distance between a center point and an arc.
- **Diameter**  measures the distance across the center point of a circle.
- **Length**  measures the distance along an edge or curve. Length markups look like a standard distance dimension, with two lines attached to the extremities of the dimension entity. If the measured entity is not a line, only one attachment line appears, connecting the dimension with the middle of the selected entity.
- **Angle**  measures the angle between two geometric elements.
- **Orientation**  The angle between a selected edge, curve, axis, or face and a reference plane. The reference plane is perpendicular to the active axis of the current coordinates systems. For example, if X is active, YZ is the reference plane.
- **Coordinates**  Identifies the x, y, and z coordinates for a point.

Note: Coordinates are calculated according to the active coordinates system. The Lock option in the Dimension controls window creates an absolute reference. Locking disables the automatic update that occurs when you change the coordinates system.

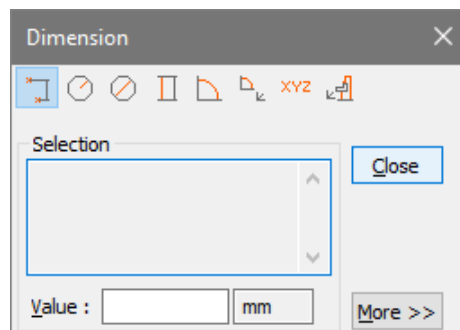
- **Minimum/Maximum distance**  The maximum distance is a measurement between an infinite planar reference and a part. The reference is the plane that is perpendicular to the active axis of the current Main CS.

Measuring a distance, an angle, or coordinates of a points





 As you select elements for a measurement, notice the information in the status bar. The status bar provides cues about each step of the selection process. These cues vary according to which measurement tool is active.



1. Click **Add Dimension** in DESIGN REVIEW tab



2. Select a measurement type.
3. Set the Selection Filters if required

-  **Points** , to snap to vertices and points on a part.
-  **Edges** : to snap to edges, curves, and axes.
-  **Faces** : to snap to faces
-  **Solids**: Snaps to surfaces solids.

Note: You can select multiple Selection Filters. Some filters are automatically deselected and unavailable for specific tools.

4. Click More, and select Dimension options, as needed.

Note: Selecting the Saved option creates a dimension annotation that is saved with the session file. Measurements you make with Saved deselected appear only temporarily.

5. In the document pane or Model Tree, select the first location or geometric element for the measurement.

Note: Some tools present an Accept/Reject  selection pointer that you use to toggle through selection possibilities.

6. If necessary, use the context menu to change the orientation of the 3D objects, and then select a second element for the measurement.
7. If Attach is selected, click the dimension annotation to set its location in the document pane. As needed, drag the dimension to reposition it.

The context menu holds four view commands you can use during the process of creating a dimension: Fit Visible, Undo Fit All, Redo, and Restore Config. View.

Modifying the Dimensioning options

By default, the Dimension controls window opens with only the tool buttons and Selection area shown. Clicking the More button expands the windows so that you can see more options, such as Saved and Attach.

- **Measurement tool buttons** Determines the type of measurement.
- **Selection** Displays information available about the selected element, such as the part name and geometry.
- **Value** Displays the measurement result.
- **Note:** You can edit the measurement result. For example, you can enter a comment in the dimension or force a specific value.
- **Saved** When selected, creates a permanent annotation of the measurement.
- **Attach** Creates lines in the document connecting the dimension markup to the locations being measured. You can drag an attached dimension within the display. When deselected, the dimension markup appears on the measurement.

- **Units** Defines the units of the dimension. The default units are those read from the CAD file data or set as session units in General Preferences.
- **Real** Creates the actual measurement.
- **Along X, Along Y, and Along Z** Creates measurements of the x, y, and z projections of the Real measurement. Labels in the document pane identify projection measurements as dx, dy, and dz components. When Saved is active during the measuring, the x, y, and z projections are saved as independent entities and appear in the Model Tree. Projections are calculated according to the active coordinates system.
- **Display Units** Shows the value and unit of measurement together in the markup.
- **Font** Opens the Font dialog box, for selecting font characteristics.

Editing a dimension

After you create a dimension, it appears in the Model Tree under the Dimensions portion of the Annotations structure. You can change the appearances and behaviors of some dimensions.

1. In either the Model Tree or document pane, right-click the dimension and choose Edit.
2. Edit the dimension:
 - To change the value displayed for the dimension, type new text or numbers in Value.
 - To move the dimension markup, select Attach (or deselect and then select again). Click to set the new position for the markup.
 - (For Coordinates only.) Select or deselect Along X, Along Y, and Along Z for the projection of the dimension along the Main CS.
 - To tie a Coordinates dimension to the current Main CS, select Lock. To automatically update the dimension when another Main CS is activated, deselect Lock.
 - To change the measurements units, select from the Units menu.
 - To show the units as part of the annotation, select Insert. To hide the units, deselect Insert.
 - To change the font characteristics for the annotation, click Font and select options.



You can quickly change the text of a dimension by clicking its text in the Model Tree and typing.

Modifying the position of a dimension

1. In either the Model Tree or document pane, select the dimension.
2. Right-click the dimension, and choose Move.
3. Move the pointer to reposition the dimension annotation, and click to set the new position.

Showing and hiding annotations

1. In either the Model Tree or document pane, right-click the dimension or markup and choose Show/Hide.

About visible and hidden dimensions

Visibility of dimensions also depends on the visibility of the parts being measured. These changes occur automatically when you hide and show 3D objects.

- Dimensions associated with elements that are all visible are also visible.
- Dimensions associated with one or more hidden elements are also hidden.
- Dimensions associated with a deleted part are deleted. If you restore the associated part, you can then restore the dimension, too. When you move a part, the dimensions associated with that part also change:
- Internal measurements of the part remain intact and move in the same way that the part moves.
- Measurements between points on that part and points outside the part are hidden. A red x on the dimension icon in the Model Tree indicates that it is no longer valid.

Calculating physical properties

The Physical Properties dialog box provides features to calculate areas, volume, mass, and the center of gravity for parts and assemblies. All these calculations are performed simultaneously.

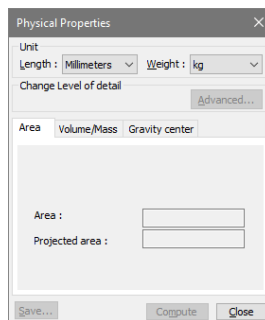
It's important to remember the distinction between Properties and Physical Properties. Properties are persistent characteristics for individual entities. Physical Properties temporarily displays calculations. These calculations are based on Properties values and the geometry of the selected parts. You can save Physical Properties calculation results by exporting them.

Calculating physical properties

You can save calculations only by exporting results to a text file. Otherwise, the Physical Properties information clears itself when you close or click outside Physical Properties.

1. Select the entities in the document pane or Model Tree.

2. Click  in DESIGN REVIEW tab



3. Click Advanced, and change the precision level, as needed.
4. Click Compute.
5. Review the results on the Area, Volume/Mass, and Gravity Center tabs.
6. (Optional) Click Save and choose a location and filename for the text file of the calculations.



You can calculate physical properties for one entity after another without closing Physical Properties. After you finish one calculation, select a different part in the Model Tree or document pane, and click Calculate again.

Physical Properties calculation results

When you calculate physical properties, the results appear on the three tabs that run across the lower area of the Physical Properties.

Area tab

- **Area** Shows the surface area of physical parts (the area you could cover with paint).
- **Projected Area** Projected areas are calculated according to the active axis of the current coordinates system.

Volume/Mass tab

- **Volume** Physical volume of the selected parts.
- **Density** Value of the Density property for the selected parts, as specified in the Properties. If the parts have different densities, the Density physical property appears as Various.
- **Mass** Calculation result based on the volume and density values.

Gravity Center tab

- X, Y, and Z Coordinates of the center of gravity for the selected parts.

Advanced options for Physical Properties calculations

The Advanced button in Physical Properties opens a dialog box with precision options. Settings you select in the Advanced dialog box apply only to the next calculation. If you close the Physical Properties or select different parts, the detail level reverts to a default setting.

Note: Level of Detail options are also available on the Graphics preferences. As preference settings, the options you choose apply globally.

User Defined

When selected, these options are available:

- **Precision list** Includes Extra Low, Low, Medium, High, and Extra High. When you select one of these levels, the Chord Height Ratio and Wireframe Chord Angle change to preset values for the selected level.
- **Chord Height Ratio** Specifies the percentage of bounding box used to compute chord height.
- **Wireframe Chord Angle** Specifies the maximum angle between two contiguous segments of wire edges for every face. The value must be from 10 through 40.

Note: If you change your selection in the precision list after you adjust the Chord Height Ratio or Wireframe Chord Angle, those changes are lost.

Controlled Precision

When selected, these options are available:

- **Wireframe Chord Angle** (As described under User Defined.)
- **Maximum Chord Height** Specifies the maximum distance between surface and tessellation.
- **Minimal Triangle Angle** Specifies the angle between two contiguous segments of wire edges for every face. Allowable values range from 10 through 30.



Managing custom coordinates systems

By default, all session files inherit a global coordinates system from the original CAD source file. This coordinates system is listed under Coordinates in the Model Tree as CAD CS. When you open a session file, the default coordinates system is always active.

You can't delete, move, edit, or rename these default global coordinates. You can define and save additional coordinates systems with the orientation and origin that you specify.

Coordinates systems are especially important for sectioning, dimensioning, displacement of parts, and controlling standard views.

Creating a coordinates system

1. Click  in DESIGN REVIEW tab.
2. Select Create Coordinates System .
3. Select a method option and set any required geometry.

Note: To undo geometry selections, click Reset and select different geometry.

The custom coordinate system appears as Axis X under Coordinates Systems in the Model Tree.

Renaming a coordinates system

1. In the Model Tree, right-click the custom coordinates system and choose Rename.
2. Type a name.

Activating a coordinates system and a specific axis

Two roles apply to coordinates systems: Main CS and View CS. You can assign both roles to the same coordinates system or assign the roles to different coordinates systems.

1. Expand the Coordinate Systems structure in the Model Tree so that you can see the available coordinates systems.
2. Right-click the coordinates system and choose its role:
 - **Main CS** Serves as the reference environment for measurements and calculations. The Main CS is the active coordinates system.
 - **View CS** Serves as the reference for Default Views. Solid arrow shapes in the lower left corner of the document pane represent the current View CS.

3. (Optional) Right-click the Main CS coordinate system and choose Direction > and X, Y, or Z to change the main axis. Assigning a role to a custom coordinate system temporarily changes its name in the Model Tree to Main CS (Z) or View CS. If you activate a custom coordinates system in both ways, its name becomes View & Main CS(Z). The custom coordinates system reverts to its original name when you assign the roles to a different coordinates system.

When you assign the Main CS or View CS role to the CAD CS, the name doesn't change.

Note: Simply selecting a coordinates system in the Model Tree highlights the three lines indicating the axes in the document pane. However, selecting or double-clicking doesn't activate that coordinates system in any role.

Changing the visibility of a coordinates system

Coordinates system axes is represented in the 3D area by triedron located at its origin point.

1. In the Model Tree, expand the Coordinates Systems so that you can see the CAD CS and all custom coordinates systems.
2. Right-click the coordinates system that you want to view, and choose an item from the context menu:
 - **Isolate** Hides all entities except the coordinates system.
 - **Hide/Show** Toggles the visibility of the axes.
 - **Focus** Zooms in on the origin point.
 - **Show Me** Draws a circular highlight around the origin point.

Deleting a coordinates system

1. In the Model Tree, right-click the coordinates system and choose Delete.

Calculating sections

Types of sections

Two methods of cutting through parts are Sectioning and Clipping.

Sections cut through either selected parts or all parts. Sectioning can cut with multiple evenly spaced, parallel planes, or with a single plane. You can save sections, which appear as entities in a Sections #x or Multiple Sections #x folder under Models in the Models Tree. Sections can be exported as vector images.

Clipping affects the view but doesn't create an entity. Clipping cuts through all visible parts with one or two planes that can be parallel or not parallel to each other. You can associate only one clipped view with a configuration, but each configuration can have a different clipped view. The Clipping command on the View menu toggles the clipped view off and on.

If you simply want to view inside an assembly, sectioning is easier than clipping. If you want to save the view and work on the displayed parts, clipping is the better choice.

Calculating a section

3D objects are cut by a section plane. You can restrict sectioning to selected parts or section the entire assembly. Cross-sections are saved as entities under Models in the Model Tree. Like other entities, you can select, hide/show, and review properties for a cross-section.

Tetra4D Reviewer gives you freedom to define how section are created. The options that control where, how, and through what parts the cross-sectioning happens are in the Sections dialog.



A Section plane entity appears in the Model Tree under each Section group. You can activate the section by right-clicking an existing section plane and choosing Activate Position. The display shows the original orientation and position of the section plane. It doesn't change aspects of the view, such as zoom level or hidden and shown parts.

The Section by Direction method uses the active coordinates system (Main CS). This method sets the cutting plane perpendicular to the active axis of that coordinates system.

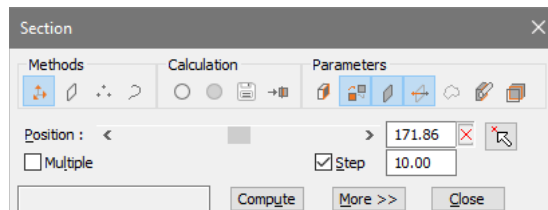
Changes to most of the Section position and display options immediately alter the 3D display. This change is temporary until you click Compute.



You can make and save snapshots of a temporary section display without actually computing the section.



1. Click **Section** in DESIGN REVIEW tab, the following dialog box appears:



2. Under Methods, select Section by Direction.
 3. Under Parameters, select Show Cutting Plane, and leave other display options deselected.
 4. Leave the Position options, Multiple, and Step settings at the default values, and click Compute.
 5. Click More, and drag the sliders or type values to change the X and Y orientations of the cutting plane.
 6. Click Compute to create a section entity.
- After you create a section, the result appears in the Model Tree under Models. The default name is "Section [x]," where x indicates the order in which sections are created. You can select, hide, rename, and apply other entity options by right-clicking section entities in the Model Tree.





Note: If you close the Section dialog without clicking Compute, the section is not created and the display reverts to its previous condition. If you leave Section open after clicking Compute, you can select different options and create additional sections.

Options for sections

The options in the Section dialog provide flexibility for the creation of cross-sections. You can specify a different cutting plane and rotate that plane in three-dimensional space. You can customize the way a section appears in the document pane. You can cut with multiple parallel planes. You can limit the parts that are sectioned and export sectioning results as vector image files.

Methods



By default, the method to define the section orientation is set to “Section by direction”. Three of the Methods options in Section controls create planes based on geometric elements that you select.

- **Section by direction**  cutting plane is normal to the active axis of the Main CS.
Note: You can reassign the Main CS, active axis, and define new coordinates systems while the Section controls window is open.
- **Section by plane**  aligns the cutting plane along one of the selected planes in the geometry.
- **Section by 3 points**  defines the cutting plane by three points that you click in the 3D display.
- **Section Along a curve** : Aligns the cutting plane along a curve that you select in the geometry.











Look for instructions in the text area to the left of the Compute button. Text appears here when the method you select requires further actions.

Calculation



- **Geometrical Section** ☐ Calculates based on exact geometry, if read from the CAD file. When not selected, sections are calculated using display information (tessellation).
- **Create Face** ☐ (Available only when Geometrical Section is selected) Calculates the face corresponding to the section. The results are used to calculate the section area in Physical Properties.
- **DXF Export**  Exports the section to a vector image file that can be used for documentation.
Note: When Multiple is selected, each calculated section is saved in a separate file. The filenames include the text typed in File Name and a number corresponding to the number of the section.
- **Add in configuration**  Makes it possible to reinstate all conditions that existed when you created the section. For example, this option retrieves the orientation, part positions, default view, and so forth. You can add sections to locked or unlocked configurations.

Parameters

- **Show capping faces**  Displays surfaces covering the cut sides of parts. Capping faces appear in Shading mode with no edge, using the part color.
- **Use part color for curves color**  Toggles the coloring of the intersection between parts and the cutting plane. When deselected, these intersection lines appear in yellow.
- **Show Cutting Plane**  Displays an outline of the cutting plane with a solid arrow indicating the axis normal to the plane.
 When the cutting plane is shown, it appears with an arrow and dots at the end of the axes lines. You can drag the arrow or dots to move and rotate the cutting plane.
- **Flip Cut Side**  Reverses the visibility of sectioned parts from one side of the cutting plane to the other.
- **Show Only Section Curves**  Hides the parts and outlines the intersections between the cutting plane and the sectioned parts.
- **Cut Only Selected Parts**  Restricts the sectioning to cut only the parts you select in the document pane or Model Tree.
- **Show Parts Completely**  When selected, the entire parts remain visible, and highlight lines show where the plane cuts through them. When deselected, the section appears in a tear-away view (with everything on one side of the cutting plane hidden).

Position

Position slider and value box Moves the cutting plane.

- **Reset Value**  Restores the cutting plane position to the default value.
- **Select Point To Put Section Plane**  Moves the designated section plane parallel to itself to the position of the selected point.
- **Multiple** Creates sections using multiple parallel cutting planes.
Note: You can see the sections after you click Compute. The Multiple option doesn't include a preview.
- **Nb Sections option and value (Appears only when Multiple is selected.)** Activates the box that you can edit to set the number of cutting planes. When not selected, the number appears in the value box but can't be edited.
Step value Has two effects. First, it adjusts the effect of clicking the Position slider arrows. Larger Step values move the slider more. Second, when Multiple is selected, Step determines the distance between the cutting planes.



More options

Clicking the More button expands the Section controls window. The button is not available for the Section Along a Curve method.

- **Tilt Y slider and value box** Rotates the cutting plane around the Y axis of the Main CS. Tilt Z slider and value box Rotates the cutting plane around the Z axis of the Main CS.

Align the camera to a section

The set camera feature arranges the 3D display according to your choice of elements. You can use this feature to adjust the view in relationship to section entities.

1. In the Model Tree, select the Section #x folder.
2. Click  in HOME tab and then  Set Camera and select From Selection.

Clipping

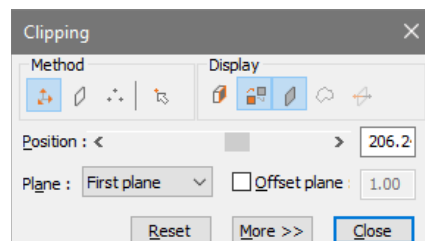
Clipping (or double-clipping) is a view created by two planes cutting across parts. The clipped portions lying between the two planes are visible and other portions are hidden. Clipping doesn't create entities, so nothing new appears in the Model Tree after a clipping operation.

Clipped views are associated with the active configuration at the time they are created. You can toggle the clipped view off and on when its associated configuration is active.

This mode helps and facilitates the understanding and the manipulation of complex assemblies.

Creating a clipped view

1. Click  in VISUALIZATION tab







2. Select the method for orienting the first plane, and select geometries, as needed.
3. Drag the Position slider or type a value for the plane displacement.
4. In Plane, choose Second Plane, and select a method and position. Select from the same options and controls as for the First Plane.
5. Choose Display options, as needed.
6. (Optional) Select Offset Plane and enter a value for the offset.
7. (Optional) Click More and drag the orientation sliders to rotate the clipping plane around the Y and Z axes.






Clipping Methods and Display options

Clipping and Sections include similar options for defining and positioning cutting planes. The Display options for the two features are also similar.

Methods options

- **Clipping by direction**  Aligns the cutting plane normal to the active axis of the Main CS.
Note: You can reassign the Main CS, active axis, and define new coordinates systems while the Section controls window is open.
- **Clipping by plane**  Aligns the cutting plane along one of the existing planes in the geometry.
- **Clipping by 3 points**  Defines the cutting plane by three points that you click in the 3D display.
- **Select point to put plane**  Moves a defined section plane at the selected point, parallel to its original position.

Display options

- **Show capping faces**  Displays surfaces covering the cut sides of parts. Capping faces appear in Shading mode with no edge, using the part color.
- **Use default curves color or use part color**  Toggles the coloring of the intersection between parts and the cutting plane. When deselected, these lines appear in yellow.
- **Display clipping plane**  Displays an outline of the cutting plane with a solid arrow indicating the axis normal to the plane.
- **Display curves**  Outlines the intersections between the cutting plane and the clipped parts.
- **Flip clip side**  Reverses the visibility of clipped parts from one side of the cutting plane to the other.

Show, hide, or edit a clipped view

1. Click  in VISUALIZATION tab.

Note: When you reopen a clipped view, the Clipping controls window opens. You can close it to leave the clipping display the way it is. Or, you can change and apply different clipping options.


Comparing and splitting parts

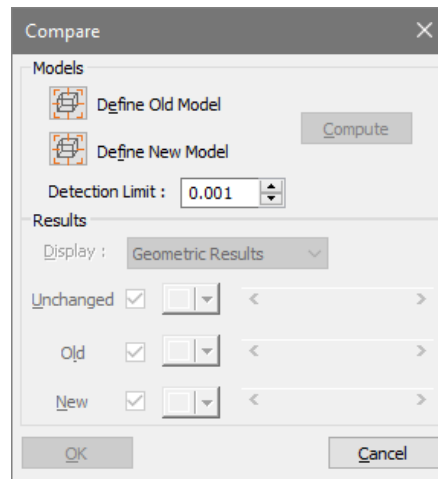
Comparing parts

The Compare feature identifies geometric differences between two parts within a document. You can compare two different parts or two versions of the same part.

The results appear in the document pane and in the Model Tree.

Note: This feature works when B-rep is selected as the Reading Mode preference.

1. Click  in DESIGN REVIEW tab, the following dialog box appears:



2. Select Define Old Model, and then select a part in the Model Tree or document pane.
3. Select Define New Model, and select a second part.
4. As needed, change the value for Detection Limit, and then click Compute.

Note: The Detection Limit sets the tolerance threshold for the comparison. Differences under the specified value aren't identified as modifications of the part.

5. Select a Display menu option and colors, and drag transparency sliders for the display of results, as needed. When you click OK, a new entity, Compare #x, appears under the Models structure of the Models Tree.

Note: The actual results of a Compare Parts computation are based on precise geometry. The Representation Results also calculates based on exact geometry, but they are shown in tessellated form. Viewing Representation Results is useful because you can easily identify which areas changed and how they changed.

Results options for part comparisons

- **Display** indicates how the calculated results are shown.
 - **Geometric Results** Calculated from exact geometry.
 - **Representation Results** Calculated from the currently active level of detail for the display.

- **Unchanged** Area common to both parts.
- **Old** Area unique to the first component part.
- **New T** Area unique to the second component part.
- **Check boxes** Set the visibility the Unchanged, Old, and New spaces. Selected items are shown. Deselected items are hidden.
- **Color swatches** Open color pickers for designating the Unchanged, Old, and New spaces.
- **Sliders** Change the transparency of the Unchanged, Old, and New spaces.

Splitting parts

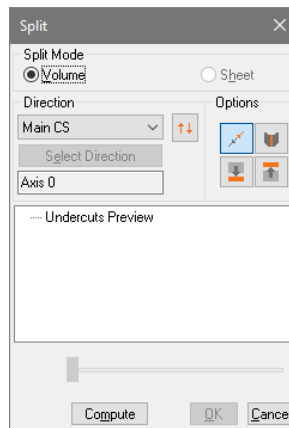
Splitting parts is a function dedicated to mold preparation for manufacturing parts. In Tetra4D Reviewer, you can split parts in your choice of directions and analyze draft angles for a specific split. You can make sequential splits in various directions.



After splitting, different colors apply to parts on either side of a split. Each split is an entity that appears as a Split #x folder under Models in the Model Tree.

Splitting is available only for files read with B-rep mode, selected in the reading mode preferences.

Splitting a part

1. Click  in DESIGN REVIEW tab




2. Select Volume or Sheet as the Split Mode.
3. Choose a direction for the splitting axis, and then select the reference items in the display.
Note: An Accept/Reject  pointer icon appears when the position you click could mean various things, such as a point, edge, or surface. Use the right and left mouse buttons to step through the possible selections.
4. As needed, select Inverse Direction  and any buttons in the Options area that you want to apply.
Note: Before you finalize the split, you can check your settings by right-clicking undercut preview and choosing draft angle analysis.
5. Click Compute.
6. (Optional) Change Split options, and click Compute again to create more splits.
7. Click OK to create the splits, or click Cancel to close the controls without creating the split.

Split options

Parting Mode





- **Volume** The default setting.
- **Sheet Processes** parting in only one direction. Optimizes results for splitting parts without any thickness. (Available only for shell-based surfaces.)

Direction

- **Main CS** Aligns the splitting axis to the active coordinates system. (No selection required.)
- **2 Points** Aligns the splitting axis to points that you designate. **Edge** Aligns the splitting axis to an edge of a designated part. **Plane** Aligns the splitting axis to a designated flat surface.
- **Inverse Direction**  Reverses the direction of the splitting axis. For two-headed arrow lines, a brightness gradient on the axis line indicates direction.
- **Select Direction** Restarts the process. Click Select Direction when you want to change your selection of two points, an edge, or a plane.

Note: The text under Select Direction describes the currently set splitting axis.

Options

- **Part in active and opposite directions**  Determines two groups of faces instead of just one group.
- **Split faces with their silhouette curves**  When available, splits any non-planar face into two faces according to the splitting direction. Also, useful for some non-planar faces that are partially visible from both sides of the splitting direction. The determined subfaces are automatically linked to the appropriate group.
- **Add vertical faces to bottom group**  Attaches non-assigned faces to the bottom group. Available only when Inverse Direction is deselected.
- **Add vertical faces to upper group**  Attaches non-assigned faces to the upper group. Available only when inverse Direction is selected.

Slider

- **Move groups along directions** (Visual aid, available after a split computation.) Moves the split entities from their original positions, following the direction of the splitting axis.

Reading Split results

The results of a split appear immediately in the Split controls window and in the Model. If you close the Split window and reopen it later, the results of the original split don't appear there. The results remain in the Model Tree. The organization of information is similar in both places.

Note: Be careful to leave the Split controls window open if you want to see a separation display. The slider in the Draft Angle Analysis dialog box creates this view.

- **Unassigned Faces** remaining stores all faces that were undetermined during the split operation.
- **Split #[x]** (In the Model Tree only.) Entity representing the splitting session. Groups, Unassigned Faces, and Parted Faces listings are nested under the Split.
- **Group #[x]** The splitting session listings. If you compute multiple splits simultaneously, then multiple Group # items appear, each with a unique value for x.
- **Parted Faces #[x]** The faces that are assigned during a splitting operation. A single split produce either one or two Parted Faces groups, corresponding to the number of directions selected. Parted Faces are nested under the Group #.
- **Position slider** Moves the display of separated parts along the splitting axis.

Note: Right-clicking an item in the results area opens a context menu. The options available depend on the item clicked.

Note: You can hide or show individual groups from a split like any other entity. The Hide and Show commands appear on the context menu that opens when you right-click the group in either the Split window or Model Tree.

Analyzing draft angles

You can analyze draft angles during the splitting process, either before or after you compute the split. Draft angle analysis is useful for testing directions before you finalize a split.

Note: After you close the Split controls window, draft angle analysis is not available.

1. In the Split controls window, choose Draft Angle Analysis on a context menu:
 - Before making a split, right-click Undercuts Preview in the results area.
 - After making a split, right-click Group #x in the results area.
2. Adjust settings and selections, as needed:
 - Click the color swatches to open color palettes, and select preset or custom colors.
 - Change the values for the angles taken according to the split directions. The angle values become threshold values for the colors display.
3. View the effects of your changes:
 - Click Show Undercuts.
 - Drag the slider to move the separated parts along the splitting axis. (Available only for computed splits, not for Undercuts Preview.)

Note: If you make further adjustments to the colors and values, click Show undercuts to toggle back to the original view. Then click Show undercuts again to update the view.

4. Close the Draft Angle Analysis dialog box:
 - Click back to split to return to the split controls window to finalize the split or change the settings for the split.
 - Click OK or Cancel to close the dialog box and return to the document window without finalizing the split.

Reassign split faces

You can change the assignment of faces only immediately after the split, while the Split controls window is still open.



Before you begin, drag the slider in the Split controls window to visually separate the various unassigned and assigned group displays.

1. With Split controls still open, select faces:
 - To change an entire group of faces, select the group in the results area.
 - To change individual faces, click or Ctrl-click in the document pane to select faces in the groups.
2. Choose from the context menu:
 - Right-click a group in the Split controls window and choose Part With.
 - Right-click the selection in the document pane and choose Part With.
3. In the document pane, click any part in the group to which you want to assign the selection.

Unassign individual split faces

1. In document pane, click or Ctrl-click to select individual assigned faces.
2. Right-click the selected faces and choose Unassign.

Chapter 5: Preparing technical documents

Tetra4D Reviewer can be used to prepare screen captures, diagrams, section views and other graphics to incorporate in technical documents.

It is also possible to prepare exploded views to include in exported PDF files. Users working in Acrobat or Reader® can view and manipulate the 3D data exported as PDFs.

3D markups

About 3D markups

3D markups are entities that appear in the Model Tree, under Annotations, and in the document pane. Like any other entity, you can hide, show, delete, and see properties for individual markups. You can select and edit a markup.

Note: Balloon markups serve different purposes than 3D markups and appear in a different location in the Model Tree. Balloon markups assign numbers to individual parts. These numbers are useful for identifying parts for exploded-view callout captions and the bill of materials.

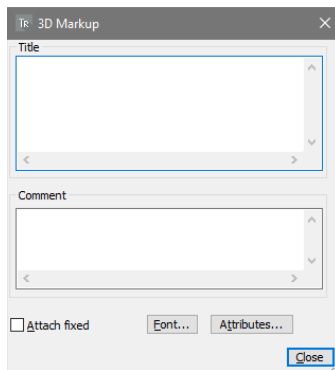
Creating a 3D markup

You can attach markups to all types of geometries, such as points, edges, axes, surfaces, and solids.



If you want to limit your markup to a specific type of entity, change the Selection Filters.

1. Click  in DOCUMENTATION tab



2. Under Title, type the text to appear as the markup label, or leave the Title blank to use the part name as the markup text.
3. Set other options, as needed:

- Type in the Comment area.

Note: Comment text appears only as a tool tip for the markup in the Model Tree and in the markup Properties.

- Click Font and select options for font characteristics.
- Click Attributes and select property information to include in the markup.

Important: Be careful to leave the 3D Markup dialog box open. If you close the dialog box before placing the markup in the document pane, no markup is created and your selections are lost.

4. Select the reference entity in the Model Tree or document pane.
5. Without clicking, move the mouse over the document pane to move a preview version of the markup.



Note: As you move the mouse, the reference point snaps to various positions on the selected entity. The preview text position and connector line also change.

6. When the preview 3D markup is in the position you want, click to set it in place.
7. Continue working:
 - To create another 3Dmarkup, change the text and options in 3DMarkups, as needed, and select a reference entity.
 - To work on a different task, close 3D Markups.

Note: After you click to set the markup position, the markup appears in both the Model Tree and the document pane.

Hide and show 3D markups

Change the visibility of 3Dmarkups:

- To hide and show individual markups, select them, right-click the selection, and choose Hide/Show.
- To hide all 3D markups, Click  Hide Session Markups in VISUALIZATION tab.
- To show 3Dmarkups, Click  Hide Session Markups in VISUALIZATION tab to deselect it. If any markups remain hidden, select them in the Model Tree and choose Hide/Show from the context menu.

Advanced 3D markups

You can add advanced options and features to markups either as you create them or later, by editing existing markups.

Attach Fixed

The Attach Fixed establishes an uneditable reference point for a markup. As a reminder, the reference end of the connector line appears as an arrow instead of a dot. You can select the Attach Fixed option when you create a markup or later, by editing.

When you create an Attach Fixed markup, your first click sets the reference point. When Attach Fixed is not selected, a preview markup snaps to different positions as you move the mouse over the document pane. When you click, the current preview sets the markup in place.

You can apply the Attach Fixed feature when you create a markup or by editing it later.

Multiple connector lines

You can attach one 3D markup to multiple parts or locations. You can add lines immediately after you create a markup by right-clicking anywhere in the document pane and choosing Add Geometry. Or, you can add connector lines later, by editing.

Bent connector lines

Connector lines can be segmented so that they bend around other displayed items. You can bend lines before you set the final location of the markup by right-clicking anywhere in the document pane and choosing Add Bend Point. Your next click sets the position of the bend point. You can also bend lines later, by editing.

You can add multiple bend points to a connector line.

Comments

Notes you add here appear as a tool tip in the Model Tree and in Properties for the markup. Comments aren't part of the markup label.

Font options

You can set font options for new markups in two ways:

- Globally, by changing settings on the Preferences Markup tab.
- Individually, by opening the Fonts dialog box from within 3D Markup as you create the markup. You can change font options for existing markups by editing.

Attributes


The Attributes dialog box provides an opportunity to add certain CAD data to the 3D markup label. For example, you can include information about the Modeler or density of a component part. This attribute information is added to Title text you type when you create or edit the markup.

Editing a markup

You can edit the text, the text display, the connector lines, and the position of an existing markup. Editing always begins by selecting the markup.

You can select multiple markups and edit them simultaneously. Simultaneous editing is useful for certain kinds of changes, such as moving markups in the same direction or changing font characteristics.

You can select markups in three ways:

- Expanding the Annotations and Markups structures in the Model Tree and clicking and Ctrl-clicking individual markups.
- Clicking and Ctrl-clicking the connector line or text of markups in the document pane.
- Click  in VISUALIZATION tab, and button Markup.

Note: Simultaneously editing the text of all markups isn't recommended. The Edit menu has no Undo command. If you change your mind and want to revert to the original text, it would require editing each markup one at a time.

Moving a markup

1. Right-click the selected markups and choose Move.
2. Move the mouse until the markup preview appears in the position you want, and then click to complete the move.

Note: When Attach Fixed is active for the markup, you can move the markup text and change the connector line, but not reference point. To make the reference point movable, too, right-click the markup, choose Edit..., and deselect Attach Fixed.

Editing 3D markup text

1. Right-click the markup and choose Edit....
2. Type text edits in Title and Comment.
3. When finished, click the Close [x] button in the upper right corner.

Note: You can edit the title by double-clicking the markup name in the Model Tree and typing.

Changing font settings for a 3D markup

- 1 Right-click the markup and choose Edit Markup Font.
- 2 Change the font options as needed.

Note: You can also open the Font dialog box by clicking Font in the Edit Markup dialog box when you are editing markup text.

Adding a bend point to a markup leader

Bend points create a segmented line so that it can bend around other display items. You can create multiple bend points in a connector line.

1. Click the markup text or line in the document pane to select the markup, or select it in the Model Tree.
2. Choose Tools > EditMarkup.
3. In the document pane, right-click the markup and choose Add Bend Point.
4. Click a position in the document pane to set the bend point position, and then click again to set the position of the markup.

You can create multiple bends in a line by choosing Add Bend Point again in the context menu.

Adding markup leaders referencing other geometries

Adding geometry means creating multiple connector lines between a markup and the component parts.

1. Click the markup text or line in the document pane to select the markup, or select it in the Model Tree.
2. Choose Tools > EditMarkup.
3. In the document pane, right-click the markup and choose Add Geometry.
4. Click to set the position of the new connector line.

You can create more lines by choosing Add Geometries again in the context menu.

Note: You can also bend the lines as you add geometries, using the right-click Add Bend Point command.

Removing one of multiple markup leaders

1. Select the markup.
2. Choose Tools > EditMarkup.
3. In the document pane, right-click the markup and choose Remove Link.

Balloon notes


About balloon notes

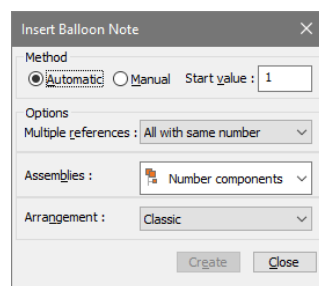
Balloon notes assign numbered balloons to parts. These useful references identify individual parts for the bill of material and in illustrations.

Balloons appear as “Balloon definition” in the Model Tree. Balloons fit under the Configurations structure, as part of the configuration that was active when you generated the balloons.

You can create balloons for locked or unlocked configurations.

Creating balloons

1. Click  in DOCUMENTATION tab, the following dialog box appears:



2. Select a Method area, select Automatic or Manual, and type a Start Value for the numbering.
3. If appropriate, choose a Multiple References option to handle cases of multiple instance soft he same part.
4. If appropriate, choose an Assemblies option to handle cases in which your selection includes a branch assembly.
5. In Arrangement area, choose how to organize the balloon positions in the document: Classic (each balloon close to the part it numbers), Square, Circle, Top, Bottom, Left, or Right.

Note: You can change the Arrangement later by right-clicking the Balloon listing in the Model Tree.

6. In the document pane or Model Tree, select the parts or assemblies that you want to number.
7. Click Create.

Balloon note options

The Insert Balloon Note dialog organizes related options together.

Automatic and Manual number assignments

You can apply manual and automatic numbering either alone or together.

Manual numbering: Manual numbering is useful when you want to reserve a number or series of numbers for specific parts. In these cases, apply the Manual numbering first. Then apply Automatic numbering to add balloon labels to the unnumbered selected parts.

Automatic numbering: the default starting value is 1, but you can start with any positive integer or zero. Regardless of what value starts the numbering, the increment is always 1.

Multiple References options

- **Only The First Numbers** only one instance of the part.
- **All With Same Number** Assigns the same number to all instances of the part.
- **All With Different Number** Assigns a unique number to each instance of the part.

Assemblies options

- **Number head group** Creates balloons for the selected branches themselves but not for the components within those branches.
- **Number Components** Creates balloons for the components within a branch but not for the items located within those components.
- **Number Items** Creates balloons for all components and for the items nested within the components.

Arrangement options

- **Classic** Sets balloons according to the available space so that they are close to parts but don't block the view of those parts.
- **Square or Circle** Sets balloons on a plane along the edges of an imaginary square or circle.
- **Top, Bottom, Right, or Left** Sets the balloons in a straight line across the top, bottom, left or right of the display.

Editing balloon number

You can change the numbering of individual balloon markups or replace balloon numbers with text.

In the Model Tree, double-click the number and type replacement text or numbers.

The size of the balloon adjusts so that it fits around the replacement text or number. Multi-word text doesn't wrap within the balloon but appears on a single line. The Balloon column in any existing Bill Of Materials updates to show the revised balloon text.


Bill of material

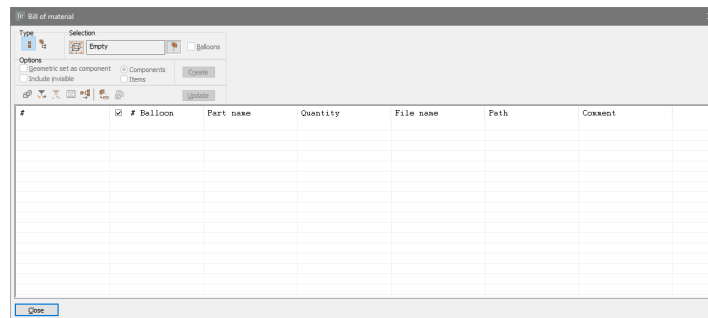
About bills of material


A bill of material (BOM) collects data about selected parts and organizes it in table form. A bill of material is especially useful for checking assemblies and producing technical documents. You determine which parts appear in each bill of material you create.

You can create multiple BOM entities. These entities are named BOM #x, where x is a number representing the order in which you created it. Each new bill of material appears in the Model Tree under the currently active configuration.

Creating a bill of material

1. In the Model Tree, activate the configuration in which you want to create the BOM.
2. Click  in DOCUMENTATION tab



3. Under Selection, click the selection button  to enable the manual selection process.
4. In the document pane or Model Tree, click and Ctrl-click to select elements that you want to include in the bill of material.
5. You can select individual items (parts), components, and assemblies.
6. In the Bill Of Material window, click Create.



Note: The information appears in the table and a BOM #x entity appears under the active configuration in the Model Tree.

Note: You can reopen any bill of material (BOM) entity by double-clicking it in the Model Tree.





Advanced options for bill of material

Settings for creating a bill of material are organized in three areas in the Bill Of Material window.

“Type” options

- **Part List Bill Of Material**  Includes the selected parts at the item level only, and ignores the assembly hierarchy.
- **Hierarchical Bill Of Material**  (Requires a selection in the Model Tree, not in the document pane.) Lists according to the Models structure and the selections in the Options area. If Items is selected, small indents in the Part Name column reflect the relationships between assemblies, subassemblies, and components.

“Selection” options

- **Selection button**  Enables you to manually select entities that you want to include in the bill of material.
- **Information area** Shows the current number of selected elements and identifies them as Groups or Items.
- **Detail Level button** Determines what elements are included when you select a group. Repeated clicks cycle through three selection levels:
- **Group**  Lists a selected assembly, subassembly, or component without listing subordinate elements within that group.
- **Component**  Lists all product definitions (components) in the selected group but does not list the group itself.
- **Item**  Lists only the individual parts (geometries) in the selected group.
- **Balloons** Automatically selects all entities with balloon markups. If the active configuration contains no balloon markups, the Balloons option is not available. When Balloons is selected, you can manually edit the parts selection by Ctrl-clicking, as needed.


“Options”

Additional options are available when Hierarchical is selected in the Type area:

- **Geometric Set As Component** Interprets any geometric sets as hierarchical nodes. (Geometric sets are available as a feature in only some CAD formats.)
- **Include Invisible** Lists all parts in the selection, regardless of whether they are hidden or shown. When deselected, hidden parts don’t appear on the list.
- **Components** Lists structures down to the component level on separate rows. Rows include references to geometric parts.
- **Items** Lists each individual geometry (item) and product definition (component) on a separate row.

Opening a bill of material

The information for parts is organized in columns. By default, attributes columns are hidden.

1. If necessary, activate the configuration in which the bill of material appears.
2. Click  in DOCUMENTATION tab.



You can also open a bill of material by double-clicking it in the Model Tree. BOMs appear under individual Configurations listings in the tree structure.

Information provided in a bill of material

The displayed columns appear in the order shown here and with these names:

- **Number** Value corresponding to the order of the row on the table.
- **# Balloon** Balloon value for the listed part (if available).
Note: A check box appears next to the # Balloon column name when balloons exist for any of the parts. Deselecting the check box hides the balloon numbers in the column but leaves the column open.
- **Part name** Name of the listed element.
- **Quantity** Number of instances of the listed part.
- **File Name** Name of the CAD file corresponding to the listed part.
- **File** Path Location of the CAD file for the listed part.
- **Comments** A placeholder column for text notes you add to the bill of material.
- **Attribute columns** By default, columns for part attributes, such as density, don't appear in the bill of material.

Adjusting layout of bill of material

The data and arrangements in a bill of material can be changed like other types of tables.

Change column width

1. Drag the bar next to column name.



Double-clicking a cell expands the column width so that you can see all the text in that cell.

Rearrange columns

1. Drag the column headings left or right.

Sort table by the data in a column

Sorting arranges the table rows in alphabetic or numeric order for the selected column.

1. Click a column header once to sort the table in ascending order.
2. Click the column header again to sort in descending order.

Show and Hide individual columns

1. Right-click any column heading, and choose Show/Hide Columns.
2. Under Standard, deselect only the columns that you want to hide.
3. Under Attributes, select only the additional columns that you want to include in the bill of material.



Adding notes in a bill of material

1. Double-click inside a Comments column cell, and type any text you want to add.

Note: Your notes are saved with the bill of material. You can edit Comments text by double-clicking a cell and typing.

Updating a bill of material







Only the Comments text can be directly edited in a bill of material. However, you can update the list with new selections and options.

1. If necessary, activate the configuration with the BOM, and button  in DOCUMENTATION tab.
2. Change settings for the BOM type and hierarchy, as needed.
3. Select the selection button  to activate part selection.
4. Select and deselect groups and parts, as needed.
5. Click Update.

Note: Updating removes any text in the Comments column. To keep a record of the comments, choose Create instead of Update. Or, export a reference copy of the BOM before updating.


Interactions between a bill of material and document views

The Bill Of Material includes a toolbar. You can use these tools for interactions between the Bill Of Material window and the parts display.

- **Highlight Selected Parts**  When selected, parts that you select in the Bill Of Material are also selected and highlighted in the document pane.
- **Display Only Selected Part**  Hides all parts except those parts that are currently selected in the Bill Of Material.
- **Restore State**  Reverts to the document pane view in effect when you created or opened the bill of material.
- **Always Fit All**  Adjusts the document pane zoom to the maximum level that still shows all selected parts. When you select a different part, the zoom adjusts again. (Available only when Display Only Selected Part is selected.)
- **Always Display All Instances**  Displays all instances of a selected part instead of displaying only that instance of the part. (Relevant only when Always Display Only Selected Parts is also selected.)
- **Select In View To Find Item In Current Bill Of Material**  When selected, parts that you select in the document are also selected and highlighted in the bill of material.

Exporting a bill of material

When you export a bill of material, the resulting file includes all information in the visible columns. Data in hidden columns isn't included.

1. With the bill of material open, click Export Bill Of Material  to open the Save As dialog box.
2. Select a location and filename.
3. In Save As Type, choose a file format: Text Files (*.txt) or HTML Files (*.html).
4. Select additional options for the selected format:
 - For a text file, choose separator characters in the Field Separator and Text Separator menus, and click Save.
 - For an HTML file, select Automatic Snapshot to include snapshots of all listed parts. Then select an image file format (.bmp or .jpg), and click Save.





Note: The Automatic Snapshot option adds an Automatic Snapshot folder to the location selected for the exported HTML file.

Snapshots

About snapshots

Tetra4D Reviewer provides features to create screen-shots of the 3D area. You determine how much of the image to capture, the resolution, and the format of the saved snapshot file. Snapshots don't become part of the session file. You save snapshots externally as independent image files or by pasting them into a different application. You control the resolution of snapshots through the Read/Write Image preferences.

Creating and saving a snapshot



1. Adjust the view as needed, such as changing the orientation, zoom level, background color, and so forth.
 2. Click  in DOCUMENTATION tab and select the type of snapshot:
 - For BMP, JPEG, and TIFF formats, Click  Window to Raster or click  Region to Raster.
 - For 2D DXF or EMF formats, click .
 3. (Only for the Region To Raster method.) Adjust the black outline that appears to enclose the area that you want to capture:
 - Drag the corners and midpoints to move the image borders.
 - Drag within the outline to move it.
 - Click Snap in the Snapshot dialog box when you finish changing the outline.
- Note:** While you are defining a region, you can zoom with the mouse wheel or by choosing View > 3D Controls > Fit Visible. Other 3D Controls tools aren't available.
4. In the Export dialog box, select the file format, filename, and location for the snapshot.

Elements shown and ignored in vector exports

Vector snapshots always include all geometric entities and balloons currently appearing in the document pane. The background color is ignored and replaced with white. Vector snapshots capture geometries in real size.


Vector snapshots ignore any grid displays. In DXF format, dimensions and markups are ignored. In WMF format, dimensions and markups are included.

Copying snapshot to clipboard

1. Adjust the view as needed.
2. Click  in DOCUMENTATION tab and click  Window to Clipboard.
3. Paste the image into a document in another application.

Creating a “Quick Snap” image

Quick Snap creates snapshots using the settings for a previous snapshot in the same work session. Consequently, Quick Snap is not available before you export at least one snapshot or immediately after you capture an image to the clipboard.


1. Create and export a snapshot by another method, using the parameters you want to apply to your Quick Snap images.
2. Adjust the view of the 3D model, as appropriate for the next screen capture.
3. Click  in DOCUMENTATION tab.
4. (Only if you used Region To Raster for the original snapshot.) Adjust the size and position of the cropping outline, and click OK in Snapshot.

Note: The Quick Snap capture is automatically named and saved without any notification.

You can continue changing views and taking Quick Snaps. Quick Snap remains available until you close the document.

The Quick Snap capture is saved in the same location as the previous snapshot. Quick Snap applies the same filename as the previous snapshot with an identifying number. For example, if you took a raster snapshot and named it engine.bmp, the subsequent Quick Snap images would be named engine_00.bmp, engine_01.bmp, and so forth.

Set Snapshot resolution

1. Click  Preferences in WORKSPACE tab, and click the Read/Write Image tab.
2. Click Resolution.
3. Select a resolution:
 - Select Window to use the current resolution of the document window.
 - Select Desktop to use the screen resolution set on the Settings tab of the Display properties Control Panel.
 - Select the numeric option and choose a resolution from the menu of resolutions supported by your computer graphics board.
 - Select Custom and enter Width, Height, and DPI values.

Note: When DPI is selected, a second set of Width and Height options are available. The pixel values define the dimensions of the snapshot image.

Note: Settings changed in the Preferences remain in effect until you change them again. If you select a high resolution and want to go back to a default resolution setting, use the Read/Write Image preferences to reset the resolution.

Advanced resolution options

The pixel values for Height and Width determine the actual size for image output. If DPI is selected, the DPI and pixel values are used to calculate the print size of the snapshot. The Height (mm) and Width (mm) describe that print size.

If you change the DPI or a height or width dimension, other values are automatically recalculated. These changes appear after you press Tab or click to advance to another option in the Resolution dialog box.

For example, changing the Width (pixels) value automatically changes the Width (mm) value.

Changing the DPI value automatically changes both the Width (pixels) and Height (pixels) values.

However, changes to a Width value don't affect Height values.

Exploded views and Animations

About exploded views and animations

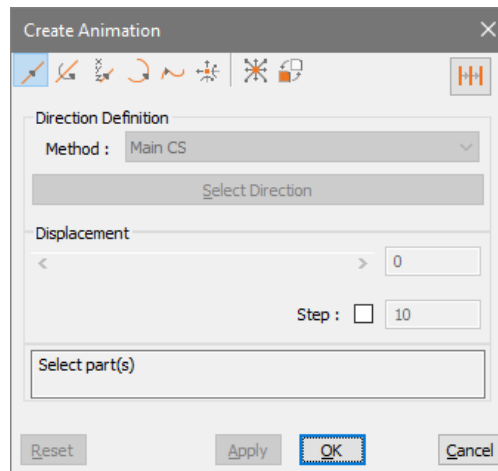
Animations show individual parts or groups of parts in action. Animations can be simple or complex, multi-step actions. Animations can show all or selected parts in motion and changes in visibility. Animations are associated with the active configuration at the time of their creation. You can create animation only when the active configuration is unlocked.


Built-in features create exploded views in a minimal number of steps. You can create other simple or complex custom animations one action at a time. Animations are saved with the session file and can be exported in AVI format or within a PDF.

Creating an exploded view

An exploded view shows the component parts separated from each other around a central point. Exploding moves each part in a separate action. You can rearrange the action sequence and add other actions by editing the animation after you create it.

1. Click  in DOCUMENTATION tab:



2. Select Explode  in the Create Animation controls window.
3. In the Model Tree or document pane, select the assembly or specific parts that you want to separate in space.
4. Select options, as needed:
 - Hierarchical Moves assemblies and parts in the order that they appear when the Models structure is fully expanded. The first assembly on the list moves first, followed by the first subassembly under that assembly. Next, any tiers under the first subassembly move, and so forth.
 - By Level Moves assemblies and parts by tier levels. All assemblies at the top tier explode first, starting at the top of the Model Tree structure. Next, all second-tier subassemblies move, in the order that they appear in the Model Tree. Then all third-tier structures move, one at a time, and so forth.

- Project Path Keeps actions parallel to the Main CS. Displacement slider Controls the amount of movement for parts. Level Limits the number of structural tiers that move.
5. Finish the animation:
- Click Apply to set the first action of the animation and continue adding more actions.
 - Click OK to save the animation with just the exploded view action.

Expanding and collapsing exploded views

You can show or reverse an exploded view as a single-step change.

1. In the Model Tree, activate and open the configuration containing the exploded view.
2. Right-click the exploded view and choose the available command:
 - Compact To collapse the exploded view.
 - Expand To expand the exploded view.

Hiding or showing paths


By default, blue lines appear in the document pane to show the travel distance and direction of animated parts. You can hide or show these lines.

In the Model Tree, right-click the animation and choose Hide Path or Show Path.

Creating animations with custom actions









Actions are individual movements or visibility changes in an animation. Several methods of defining movements are familiar techniques used to move parts in the document pane.

1. In the Model Tree, activate the configuration in which you want to create the animation.

Note: You can create new animations only in unlocked configurations. If you lock the configuration after creating an animation, you can edit the animation without unlocking the configuration.
2. Click  in DOCUMENTATION tab.
3. Select an action type for the first action, using the buttons across the top of the window.
4. In the Model Tree or document pane, select the parts that you want the first action to move.
5. Select the appropriate options for the action type, and review the preview results in the document pane.
6. Click Apply to define the final state for the action.
7. Repeat the process to create more actions, using your choice of methods, options, and parts.






After you create an animation, you can set cameras in the Play Animation controls window. Cameras change the point of view as the animation plays.

Types of action

- **Move Along Axis**  Moves parts in a straight line that you specify in Direction Definition.
- **Rotate Around Axis**  Turns parts around a line that you specify in Direction Definition.
- **Move With Coordinates**  Displaces parts by specific values for the X, Y, and Z axes.
- **Move On Helix**  Combines movements along and around a line that you specify in Direction Definition.
- **Move On Curve**  Moves parts along an exact-geometry curve that you select in the Model Tree. Or, moves along a selection in the document pane that is either the edge of a surface or a face of a solid.
- **Moving Box**  Moves with the Drag & Drop controls. Multiple changes made with different controls can apply to a single action.
- **Explode**  Separates parts in space around a central point.
- **Hide/Show**  Toggles the visibility of selected parts.


Options for custom actions

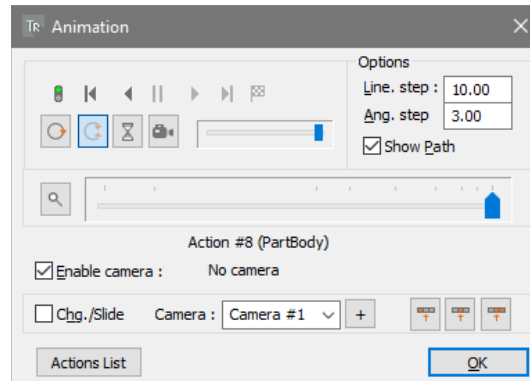
The options that appear vary according to the selected type of action. Options are available only after you select the parts for the action.










- **Stack Up**  Stack Up creates a separate action for each part you select. The parts move by successive multiples of the displacement setting.
Note: The order in which you select parts determines how far and in what order they move. For example, if the first part moves by x amount, the second part selected moves 2x, the third part selected moves 3x, and so on.
The actions occur in the reverse order that you selected the parts. For example, if you create a Stack Up animation with four selected parts, part #4 moves first. Part #3 moves next, and so on.
Note: Stack Up is available only for Move Along , Rotate Around Axis , and Move With Coordinates  action types.
- **Displacement sliders** Move selected parts, either by linear displacement or rotation. Two methods have multiple sliders:
 - **Move On Helix** The upper slider sets the translation (linear displacement). The lower slider sets the rotation angle.
 - **Move With Coordinates** Each of the three sliders sets the displacement along a specific axis of the Main CS.
- **Direction Definition**
 - **Main CS** Moves along the active axis of the active coordinates system.
 - **2 Points** Sets the direction by two points that you select. The Accept/Reject pointer  appears as you select points so that you can confirm the position or toggle to another possible location.

- **Edge** Sets the direction parallel to an edge that you select in the document.
- **Plane** Sets the direction perpendicular to a plane that you select in the document. (Not available for Move On Helix actions.)
- **Select Direction** When highlighted, shows that the selection process is active. You can click Select Direction to deactivate and restart the selection process. (Not available when Main CS is selected.)
- **Reset** Sets all options back to the default values.
- **Step value** The Step value controls the number of intermediate positions shown when you play the animation. Smaller step values show more intermediate positions.
- **Reverse Path** (Available for Move On Curve.) Changes the direction of movement along the curve.
- **Keep Parallel** (Available for Move On Helix and Move On Curve.)
Note: When Keep Parallel is selected, the orientation of the parts is always parallel to their original orientation. This type of rotation compares to a human hand sliding in circles on a flat surface. The hand always points in the same direction.
When Keep Parallel is deselected, the selected parts stay oriented to the center of rotation as they move. This type of rotation compares to a passenger airplane circling around an airport at constant altitude. The plane constantly changes its orientation to align with the circumference of the circle.
- **Hide and Show** (Available for Hide/Show.) Sets the visibility of the selected parts for the current action. The visibility state applies to all intermediate actions until you change it again in a later action.

Playing an animation


1. In the Model Tree, apply and expand the configuration with the animation.
2. Click  in DOCUMENTATION tab



3. Set the starting point for the animation:
 - Drag the long action slider in the middle of the Animation controls window.
 - Click Go To Start  or Go To End .
 - Click Previous Action  or Next Action  to progress through actions one step at a time.
4. Drag the smaller slider to set the animation speed.
5. Select automatic pause options:
 - To play in continuous loops without pauses, select Repeat Simulation .
 - To play forward and then backward in continuous loops without pauses, select Auto Reverse . (Available only when Repeat Simulation is selected.)
 - To pause at the end of each action, select Auto Pause .
6. Set the camera option:
 - To play entirely from the default point of view, deselect Enable Camera.
 - To show assigned changes in the point of view, select Enable Camera.
7. Play the animation, either by dragging the action slider or by selecting Play Forward  or Play Backward .

Note: The Chg./Slide option determines whether or not the camera changes appear when you drag the animation slider.



Options for playing animations

- **Line. Step** Controls the default speed at which linear displacements play. Smaller values move the selected parts by smaller fractions of the total displacement, which slows the movement during playback. The linear step value applies only to actions without custom Step values.
- **Ang. Step** Controls the default speed at which rotations play. Smaller steps rotate the selected parts by smaller angles, slowing the movement during playback. The angular step value applies only to actions without custom Step values.
- **Show Path** Toggles the visibility of a line and dots indicating the travel positions of parts for each action.
- **Show Current** Action Slider button  Toggles what the main slider shows: global actions (the entire animation sequence) or individual actions (the current action only).

Defining cameras for an animation






A *camera* sets the view at a specific orientation and zoom level. You can define cameras that change the point of view as an animation plays. Camera changes in an animation make smooth transitions as they shift from one point of view to another.

After you define custom cameras in the Play Animation window, you can apply them to the actions in the animation. You can edit the camera assignments after you make them. You can play animations with the camera disabled, so the action appears from a constant point of view.




1. In the Model Tree, activate the configuration with the animation, and click  in DOCUMENTATION tab.
2. Select Enable Camera.
3. Using the 3D Controls tools, set the orientation and zoom level for the camera.
4. Click Add Camera .
5. Repeat the process, as needed, to add more cameras.

The camera is automatically named Camera #x, where x represents the order in which you created it. Cameras can't be edited, renamed, or removed from the menu.

Applying cameras to actions

1. In the Model Tree, activate the configuration with the animation, and click  in DOCUMENTATION tab.
2. Select Enable Camera, and click Action List.
3. Activate an action in the Action List in any of these ways:
 - Drag the action slider.
 - Click Go To Start  or Go To End .
 - Click Previous Action  or Next Action  to progress through actions one step at a time.

Note: Be careful to use these controls to navigate to the action you want to see. Merely highlighting the action in the Action List does not activate the selected action.

4. Choose a camera from the Camera menu, and apply that camera:
 - To apply the camera to only the currently selected action, click Define Camera For Current Action .
 - To apply the camera to all actions before the selected action, click Define Camera For Previous Actions .
 - To apply the camera to the selected action and all later actions, click Define Camera For Current Action And Next Actions .
5. Continue activating actions and applying cameras, as needed.

Note: You can apply a specific camera to several actions or apply a different camera to each action. You can also apply a mixture of No Camera and custom cameras.


Note: The Chg./Slide option enables camera changes when you drag the action slider. By default, Chg./Slide is deselected, which prevents changes in the point of view when you drag the action slider.

Editing an animation

Animations fall under configurations in the Model Tree. To edit an animation, it's important that you activate the appropriate configuration first.

Adding actions to an existing animation







Adding actions simply continues the process you used to create the original animation. To review those steps and options.

1. Click  in DOCUMENTATION tab.
2. Select an action type, parts, and options, and click Apply.

Note: The new actions are added to the end of the animation sequence, as shown in the Model Tree structure of the animation.

Changing the sequence of actions

Changing the sequence can be especially useful for Explode animations. You can change the order in which parts move out from the center. You can also intersperse other actions, such as changing the visibility of individual parts.

1. Click  in DOCUMENTATION tab.
2. Select Action List to expand the Animation controls window.
3. Select an action in the list and select changes that you want to apply:
 - Move Up  and Move Down  buttons Changes the sequence of a selected action in the Action List.
 - Group Actions  Groups the selected actions so that they play simultaneously.
 - Ungroup Actions  Removes the selected actions from any groups in which they were included.
 - Remove Action  Deletes the selected action from the animation.

Note: Although you can directly rearrange or delete actions in the Model Tree, using the Action List is the preferred method. The advantage of using the Action List is that you can immediately check results by playing the animation with the slider or play controls.

Editing options for individual actions

1. Right-click the action in the Model Tree or the action path in the document pane, and choose Edit Action.

Note: The paths of fall actions appear in the document pane. Yellow arrows, green endpoints, and step numbers appear for steps that can be lengthened or shortened.

2. Make changes available for the action, as needed:

- Type a new name for the action.
- Drag the slider or type a value to change the displacement.
- Type a new Step value.
- Select or deselect Keep Parallel.

Note: You can also drag the yellow arrows, endpoints, or X, Y, and Z arrows for an action to make changes in the document pane. The displacement value in Edit Action updates as you drag.

3. Click Apply.
4. Select another action in the animation:
 - Select the action in the Model Tree.
 - Select the yellow arrow or path of the action in the document pane.
5. Change the action, as needed, and click Apply.
6. Continue selecting, changing, and applying new options to other actions, as needed.

Note: If you select another action without clicking Apply, any changes you made to the previously selected action revert to their previous settings.

Adding or removing parts to an action

1. Right-click the action in the Model Tree or the action path in the document pane, and choose Edit Action.
2. In the document pane or Model Tree, select the parts you want to add.
3. Right-click the selection and choose Add Geometry or Remove Geometry.
4. Click APPLY.

Grouping and ungrouping actions

Grouped actions occur simultaneously. Actions can be grouped only when they have no common part.

1. Select the actions in the Model Tree.
2. Right-click the selection and choose a command:
 - To group the selected actions, choose Group Actions.
 - To remove the selected actions from an existing group, choose Ungroup Actions.


Chapter 6: Exporting and printing

When you are ready to share the results, you've created in Tetra4D Reviewer, you can export and print.


An important feature of Tetra4D Reviewer is the ability to export 3D models to PDF. Users who open the PDF in Acrobat or Reader can collaborate in reviewing the design. For example, you can apply one of the prepared form templates to create a PDF form that other users fill in and return. Or, you can use the Review features in Acrobat to distribute the PDF and collect feedback from team members and clients.

Exporting session

Exporting an assembly

1. Click  in application button, and select a filename and location for the exported file.
2. Choose a format in Save As Type.
3. Click Options to open the Read/Write CAD for the selected format, and make changes, as needed.

Exporting only some parts of an assembly

1. Click **File Tree** in Workspace tab, if necessary.
2. Expand the File Tree format for the parts you want to export, and select those parts.
3. Right-click the selection and choose Create New Product.
4. Load, unload, import, delete, and copy-and-paste to include just the parts you want to export, as needed.
5. Click  in application button, select a filename, location, format, and format options, as needed.

Exporting to 3D PDF


Acrobat and Reader users can view and manipulate the 3D models that you export to PDF from Tetra4D Reviewer. For example, users can activate a 3D model in those applications. Users can then manipulate the model, such as rotating it, adjusting the lighting, viewing markups, and adding 3D comments. Acrobat and Reader users can also open and expand a view of the hierarchical structure of the model.

Several templates for exporting to PDF are included with Tetra4D Reviewer. Some templates are PDF forms. These forms include various fields that are typically needed for product design review and other manufacturing processes.

In Acrobat, you can also create and save custom templates. These templates are typically based on documents that you prepare in various business applications.

For information about creating and using PDF templates and PDF forms, see Tetra4DConverter Help.

Exporting to 3D PDF using a PDF template


1. Click  in application button.
2. Choose PDF Files (*.pdf) on the Save As Type menu.
3. Click Options.
4. Under Write Options, click Options.
5. Under Template PDF, select Use Template, and then click the Browse button.
6. Select the provided or custom template that you want to apply.
7. Select any other export options that you want to apply, and then designate a file name and location for the exported PDF. Once you select a template, the choice is used for all subsequent PDF exports. To stop applying the template, you can deselect Use Template. You can reopen the PDF Export Options either during another export or by opening the Read/Write CAD preferences for PDFs and clicking Options.



PDF templates provided in Tetra4D Reviewer


The available PDF templates for Tetra4D Reviewer are:

- 3DTemplate_Tetra4D_ACR
Opens the model as a form with all the elements in a3drtemplate_02 plus boxes for Comments, Configurations, Functions, Results, and Details. Created and editable with Acrobat X Pro forms features.
- 3DTemplate_Tetra4D_LC
Identical to a3drtemplate_03, but created and editable with LiveCycle® Designer ES2. The LiveCycle Designer application is installed with 3D PDF Converter Premium and Acrobat X Pro.
The PDF templates for Tetra4D Reviewer can be found in the 3DPDFConverter\templates folder that is installed in the plug-ins folder with Acrobat

Exporting an animation to AVI

1. In the Model Tree, apply and expand the configuration with the animation.
2. Click  in DOCUMENTATION tab.
3. Adjust the playing options for the animation.

Note: Exporting with Auto Pause  selected creates a separate AVI file for each action. The AVI filenames include a number corresponding to the action. Exporting ignores the Repeat Simulation  setting.

4. Click Save AVI , select a location and filename, and click Save.
5. In the Video Compression dialog box, select Compressor and Compression Quality Settings. If appropriate, click Configure and select settings for your compressor utility.

Printing

You can print the current view in the document pane on a color printer or black ink printer. The printed image includes all visible entities, including balloons, dimensions, markups, and axes for the coordinates systems.





The Printer Setup Preferences determine the text, alignment, and font options for the header and footer for the pages. The same Preferences tab contains an Options link to additional settings for scale, grids, and other advanced features.

You can print two images in two ways: raster (pixelated) and vector. Raster images show parts as they appear in the document pane. For example, if Solid is selected, then the printed image includes shadings and surfaces. Vector images ignore shadings and print only edges, wireframes, markups, and dimensions.

Printing involves the standard dialog boxes and options that are common to most Windows applications. For simple print jobs, the only difference is the presence of Scale options in the lower left corner of those dialog boxes.

Set the scale for a print job

You can include or exclude the background color and the View CS indicator in the printed image.


1. Click  in application button and then button  **Print Setup** or button  **Print Preview** and then .
2. Select Scale and the amount of scaling that you want to apply:
 - **Scale value** Choose a scale coefficient, or type to enter a decimal number.
 - **Automatic Scale** Calculates the scaling for you so that the image fits on one sheet of the selected paper size.

Note: You can set the scale value and advanced Scale features in Preferences. Use the Options button on the Printer Setup tab to see these settings.

Scale values that you change in the Preferences, Print, or Print Setup dialog boxes update all three places.

Define headers and footers for print jobs








Printed documents can include a header, a footer, or both. A header or footer can contain multiple items of typed and automatic entries.

1. Button  **Preferences** in WORKSPACE tab, and select Printer Setup.
2. Select Header, and type to add text.
3. Select an Align option for the header to set the horizontal position.
4. In Add Option, choose an automatic entry for the header, if appropriate:
 - **File Name** Adds the name of the read file.
 - **User Name** Adds the name you registered when you installed Tetra4D Reviewer.
 - **Printing Date** Adds the current date when you print the document.
 - **Scale** Adds a note about the scale setting.

5. Select Footer, and specify the text, alignment, and any of the automatic entries, as needed.
6. Select options in the Font area or click Select to set these options in the Font dialog box. The header and footer Preferences apply to future print jobs until you change them.

Print multiple views on pages



You can print several views of the components on the same page.

1. Click  Preferences in WORKSPACE tab, select the Printer Setup tab.
2. Select Options, choose frame and sheets options under Multiple Views, and close the Preferences.
3. Click  Split in DESIGN REVIEW tab.
4. Change the orientation, zoom, and visibility settings for each segment of the document pane, as needed.
5. Start printing:
 - Click  in application button and click  Print Preview and then .
 - Click  in application button and then click  Print Setup, to print without preview

The Preferences apply to future print jobs until you change them. If you drag the borders to collapse view panes, collapsed panes aren't printed. Dragging the borders to resize the view panes has no effect on the printed views.



Showing the grid on printed pages

Set up the printer for raster or vector printing

1. Click  in application button and then click  **Print Setup**.
2. Set the Scale option:
 - To print raster images, deselect Scale.
 - To print vector images, select Scale.

Set options for printing grids


After you set up the printer for raster or vector printing, you can include the grid in the printed images. Some advanced options are available for vector images.

1. Button  **Grid** in VISUALIZATION tab.
Note: For raster printing, no further steps are necessary. For vector printing, you can continue to set advanced options.
2. Button  **Preferences** in WORKSPACE tab, and select the Printer Setup tab.
3. Click Options.
4. If necessary, select Scale, and then select grid options:
 - Select one or more Step On [axis] options, to specify which axes you want to print.
 - Set values for the grids along each selected axis.
 - Select Extend To View to print the grid over the entire page. Or, leave it deselected to print the grid only over the area that contains parts.
5. Click OK to apply and close the Preferences.
6. Choose File > Print Preview to make sure that the grid appears the way you want it to.
 - If you are satisfied with the appearance of the grid, click Print.
 - If you want to change the grid settings, click Cancel, and go back to the Grid preferences and select different settings.

Note: Preferences apply to future print jobs until you change them. If the grid is hidden at the time of printing, it doesn't print.

Chapter 7: Preferences

The Preferences dialog box determines global settings for Tetra4D Reviewer. After you change Preference options and click either OK or Apply, you must close and reopen the document for the changes to take effect.

The button  **Preferences** command opens the Preferences dialog box so that all tabs are available.

General preferences

- **Language** Sets the language for text in the application. Relevant only if multiple languages are installed on your computer.
- **Confirm Deletion Of Unused Files** Determines whether an alert appears when you close documents.
- **Show Reading Options When Necessary** Controls the display of reading options when opening CAD files by dragging. Select this option if you want to be able to access the reading options regardless of the technique used to open files.
- **Drag & Drop: Reset Import/Open Dialog Box** Cancels options previously selected when opening a file by dragging the file icon. (Available only if Always Do The Same is selected in the Drag And Drop To Open/Import Files dialog box.)
- **Highlight Entity When Mouse Pointer Is On It** Highlights selectable entities as the pointer hovers over those entities. The Highlight Selection option on the Selection Filter toolbar must be selected.
- **Highlight Tree Item When Select Entity** Expands the Model Tree structure to the entities selected in the document pane and highlights those entities. When deselected, the Model Tree structure doesn't expand or collapse, and highlights the collapsed elements that contain the selected entities.
- **Set Working Unit To** Overrides the CAD file units of measurement for new documents, replacing the original units with the selected units.
- **Precision** Number of digits displayed to the right of the decimal. Applies to values in Physical Properties and for dimensions.

2D preferences

This tab contains parameters dedicated to native 2D drawings display.

- **Default Background Color** Opens a color picker for the background color.
- **Simplify Text Size** Sets the threshold size for greeking text that is too small to read at the current zoom level. Greeking replaces text with non-letter forms that act as placeholders in the document display.

3D preferences

This tab contains several areas, related to display and rendering parameters.

Rendering options

- **Rendering buttons** Set the default display mode: Solid, Wireframe, Illustration, or Perspective. Solid and Wireframe can be selected together or separately. Perspective can apply to any of the other three selections.
- **Transparency** Enables display of reduced transparency levels specified in the native CAD files and in Properties for selected parts. Deselecting Transparency is recommended for systems without graphic boards that support transparency.
- **Material color** Enables display of reflective shine on the surfaces of parts.

Default definition options

You can apply customized 3D default preferences at your choice of levels: at the current document level and at the application level. The selected level determines whether the changes apply to the currently active document or to future documents.

- **Default** When selected, the changes to the 3D preference options become the default settings at the application level. These settings apply to future documents but don't affect the current document. The Preview shows an example of how the settings affect the 3D image of a part.
Note: The Apply button is available after you change one or more options. Clicking Apply while Default is selected doesn't change the current document. Clicking Apply does save the changed application-level default settings so that you can continue working in the Preferences.
- **Current Document** When selected, changes in the 3D preference options apply to the active document but don't affect future documents. When you switch from Default to Current Document, the preview images show the 3D options currently applied to the active document.
- **Restore Default** Reapplies default settings according to which level is selected:
 - When Default is selected, restores the application-level default settings installed with Tetra4D Reviewer.
 - When Current Document is selected, restores the currently saved application-level defaults as the default settings for the active document.
- **Save As Default** (Available only when Current Document is selected) Sets the currently selected options as the application-level defaults.

Light settings

- **Lights menu** Specifies the light source to which changes apply.
- **Note:** You can change the position of the lights by dragging them in the Preview display. If Current Document is selected, clicking Apply updates the display in the document pane.
- **Enable** Turns off or turns on the light selected in the Lights menu.
- **Local** Activates options in the Attenuation area for the selected light.
- **Attenuation slider** Increases or decreases the intensity of the selected light according to the selected method:
- **Constant** Changes the intensity by a constant factor.

- **Linear** Changes the intensity in proportion to the distance between the screen and the part.
- **Quadratic** Changes the intensity in proportion to the square of the distance between the screen and the part.
Note: Results appear in the Preview as you change methods and drag the slider.
- **Colors slider** Increases or decreases the qualities of the selected light:
- **Ambient** Changes the level of the overall light in the scene.
- **Diffuse** Changes the level of focused light, like a spotlight.
- **Specular** Changes the degree of reflecting shine off the part. (Applies only when Material Colors is selected in the Rendering area.)

Background options

- **Scaled Background option and slider** Displays the background color as a vertical gradient using different values of the selected color. Dragging the slider to the left lightens the upper end of the gradient.
Dragging the slider to the right darkens the upper end of the gradient.
- **Color picker** Sets the background color.
Note: The background color can vary between the configurations inside one document.

Advanced dialog box

Provides information about the graphic board of the computer and options for parameters linked to OpenGL capabilities of the graphic board. The available options and information depend on the type of graphics board in use.

- **Use Hardware Acceleration** Activates hardware acceleration for display.
- **Use Hardware Antialiasing** Enables high-quality rendering for images and text.
- **Force Redraw** Disables management of display refreshes by the graphic board.
- **Enable Double-sided Rendering** When deselected, improves performance by not rendering back triangles.
- **Reverse Face Rendering** Inverts the display of parts.
- **Clipping Plane Position** (For Perspective display mode) Controls the separation displayed between the clipping plane (OpenGL) and from the camera.

Fly Through preferences

Keyboard

The list of keyboard shortcuts that apply only when Fly mode is active. You can edit the list, substituting keys that are easy for you to remember.

- **Front** Moves the camera forward, toward the target position.
- **Back** Moves the camera backward, away from the target position.
- **Left** Moves the target position to the left.
- **Right** Moves the target position to the right.
- **Up** Moves the target position up.
- **Down** Moves the target position down.
- **Roll Left** Tilts the camera to the left.
- **Roll Right** Tilts the camera to the right.
- **Stop** Pauses the movement.
- **Speed Up** Increases the movement speed.
- **Speed Down** Slows down the movement speed.
- **Wheel Back** Inverts the movement without changing the speed.

Other options

- **Mouse** Drag the slider to the right to increase the sensitivity of the mouse or to the left to decrease sensitivity.
- **Speed** Drag the slider to the right to increase the Fly mode speed or to the left to decrease speed.
- **Height** Sets the value and units for the altitude of the camera above the reference plane.
- **Units** Lists available printers', metric, and English units of length.
- **Display Plane** When selected, shows the reference plane as a transparent plane.
- **Target** Displays cross hairs indicating the direction in which the camera travels.
- **Indicators** Displays a speed indicator bar at the bottom of the document pane and a horizon indicator on the right side of the document pane.
- **Restore Default Keys** Replaces any custom-defined keys for Fly mode with the default control keys for Tetra4D Reviewer.

Graphics preferences

- **Level Of Detail** Sets the precision for the screen display. Options range from Extra Low to Extra High. The Advanced option makes the Advanced button available.
- **Default** Applies the current settings as defaults for new documents and loaded parts.
- **Current Document** Applies the current settings to only the active document.
- **Advanced** (Available when Advanced is selected for Level Of Detail) Opens the Advanced dialog box for setting user- defined precision levels.
- **Edges color options** Sets the color of wireframe edges.
- **Set To Default Color** When deselected, edges appear in the same color as the faces to which they belong. When selected, the color picker is available.
- **Color** Specifies the color in which edges appear.
- **Section color picker** Sets the highlight color for the edges created by the cutting plane for a section.
- **Items color picker** Sets the color for parts or faces of parts that have no defined color.
- **Framerate Threshold slider** Sets the number of frames per second maintained during dynamic manipulations. When the complexity of the model causes the frame rate to fall below the specified value, the model is optimized during the manipulation. Optimization temporarily removes faces in the display while the model is in motion.
- **View** Uses smooth transitioning when you apply a different view, such as a different configuration or a default view.
- **Zoom/Fit** Uses smooth transitioning for changes in zoom level, including Fit Visible.
- **Temporary Hide shortcuts** Options that set the keyboard shortcuts for temporarily hiding and showing selected entities.
- **Display Thickness** Shows lines with a thickness for entities in wireframe display mode.

Grid preferences

- **Main Color** Color picker for the major lines in the grid. Select Automatic to enable Tetra4D Reviewer to adapt the grid color according to the background color for each configuration. Select a specific color to prevent the line color from adapting to different background colors.
- **Sub Color** Color picker for the finer lines in the grid. Select Automatic to adapt to different background colors. Select a specific color to prevent the line color from adapting to different background colors.
- **Style** Graphic menu of line styles, such as a solid line, dotted line, and so forth.
Note: The final option on the menu is blank. When selected, the grid lines are transparent but numbers for reference numbers along the sides of the document pane remain visible.
- **Text Color picker** Sets the color of the text that indicates reference numbers along the sides of the document pane. When Automatic is selected, Tetra4D Reviewer selects and updates the text color according to changes in background color.
- **Font** Designates the font family for grid reference numbers. When Automatic is selected, Tetra4D Reviewer manages font characteristics.
- **Size** Sets the font size for grid reference numbers. (Not available if Automatic is selected.)
- **Select** Opens the Font dialog box for setting more font options. (Not available if Automatic is selected.)
- **Position options** Selects the edges of the document pane where the grid reference numbers appear: Left, Right, Top, and Bottom. You can select any or all of these options. When none are selected, the reference numbers don't appear.
- **Lock Rotation** After selected and applied, disables the Rotation and Spin tools in future activations of Grid view. When you hide the grid, rotating and spinning are available again. When Lock Rotation is deselected, rotating and spinning are always available.
Note: If the grid is shown when you select and apply Lock Rotation, rotating and spinning remain available. Rotating movements are locked after you hide the grid and then show it again.

Markups preferences

- **Always Pre-select Default Font** For session markups, applies the selected font preference settings to new 3D markups, dimensions, and balloons. When deselected, applies the most recently used markup or dimension font properties to new markups, dimensions, and balloons.
- **Font options** Specify the default characteristics for markup text. You can select the font family, size, and style.

Note: The Frame style option displays the markup text within a wireframe box.

- **Color pickers** Designates colors used for markups:
- **Text** Applies the selected color to the text
- **Attach** Applies the selected color to the lines between the markup and the part. The same color applies to the wireframe box around the text that appears when Frame is selected in the Style area.
- **Marker** Applies to the dot at the markup end of a connector line.
- **Always** Substitute Always uses the Substitution Font selection to replace markup fonts for CAD markups read from the CAD files.
- **Substitution Font** Designates the font family used to substitute for CAD markup fonts.
- **Preferred 3D PMI Rendering Mode** Defines how PMIs are processed. The selected setting affects all displayed PMIs.
- **Use Content's Setting** PMIs behave as defined in native CAD files: either buffered or always in front.
- **Always Render 3D PMI In Front Of Model PMIs** always appear superimposed on visible parts so that PMIs are always visible.
- **Always Render 3D PMI In Using Z-Buffer** PMIs appear according to buffer-defined positions. When selected, a PMI or part can block other PMIs or parts in some views.
- **Preferred 3D PMI Text Orientation** Defines how the CAD-defined initial orientations of PMIs behave in the display. (Applies only to PMIs created with a flip attribute in the CAD application.
- **Use Content's Setting** PMIs behave as defined in native CAD files: either always flipped or never flipped.
- **Always Flip PMIs** PMI text is always readable, regardless of changes in the orientation of parts.
- **Never Flip PMIs** PMI text always appears with the same orientation to the related parts. For example, when you turn the part upside down, the text is also upside down. If you rotate the part vertically, the text rotates, too, so at 180 degrees the text appears as the mirror image or the unrotated view.

Printer Setup preferences

All text appearing in the Header and Footer text boxes is included when you print. When Header is selected, Footer is unavailable, but it is not disabled. If you want to remove a header or footer, select that option and delete the text.

Headers and footers characteristics are set separately.

- **Header** Activates the Header box where you type text that you want to appear across the top of printed pages. When selected, all other options apply only to headers.
- **Footer** Activates the Footer box where you type text that you want to appear across the bottom of printed pages. When selected, all other options apply only to footers.
- **Align options** Aligns the header or footer text: Left, Center, or Right, along the top or bottom of printed pages.
- **Add Option** Automatically appends the designated item into the header or footer text. For example, if you typed "Project A," in Header and then selected Printing Date (&t), the Header shows "Project A (&t)." When you print, the current date replaces the "(&t)" symbol. You can add multiple Add Option selections to a header or footer.
- **Name** Designates the font family for the header or footer text.
- **Size** Designates the font size for the header or footer text.
- **Select** Opens the Font dialog box, which has additional options for fonts.
- **Style** Specifies the selected typeface styles for the header or footer text.
- **Options button** Opens the Options dialog box for setting scaling, grid, and views options.


Options dialog box

- **Scale** Enables options for scaling, grid, and views options. When deselected, prints raster images. When selected, prints vector images against a white background.
- **Scale Coefficient** Sets the scaling coefficient for the printed image.
- **Automatic** Adjusts the scaling to fit the printed image on the currently selected paper size.
- **Black And White** Printed lines are converted to black during Printing. Available only when Scale is selected.
- **White Background** Changes the background to white for printing raster images. Available only when Scale is not selected.
- **Main Frame** Draws a frame around the page, considering the format and the margins.
- **Border** Draws lines in the corners, indicating the limits of the printed page size.
- **Step On X** Designates the spacing of grid lines along the x-axis. If deselected, the x-axis lines are not printed. **Step On Y** Designates the spacing of grid lines along the y-axis. If deselected, the y-axis lines are not printed. **Step On Z** Designates the spacing of grid lines along the z-axis. If deselected, the z-axis lines are not printed.
- **Extend To View** Prints the grid for the entire background. If deselected, prints the grid only in areas where parts appear.
- **Frame Around Views** Draws a frame around each printed view.
- **Views menu** For printing a document with multiple views in different panes of a split

window. Specifies which views are printed:

- All Views Prints all currently displayed views on one sheet of paper. If the document pane is split unequally among the views, the relative proportions of the panes also appear on the printed page.
- One View Per Sheet Prints each displayed view on a separate page.
- Active View Only Prints the currently active view.

Reading Mode preferences

- **rep (Solid)** Boundary representation, by which solids are read as solids with topology. B-rep reads all exact geometry available in the CAD files and includes that geometry in the session.
- **Tessellation (Faceted)** Reads a representation of the solids but not to the standard of exact geometry. The tessellation can be read from any tessellation data in the CAD file. Or, the tessellation can be calculated from the exact geometry in the CAD file.
- **Note:** After a file is opened in B-rep reading mode, you can alternate between working in tessellated and exact-geometry representations of the file. If the file is read in Tessellation reading mode, exact geometry is not available.
- **Enable Disk Cache** An optimization mechanism used for reading large data files in B-rep reading mode. The cache stores some geometry and tessellation information on the local hard drive. The cache reduces the amount of data stored in the session. The result is that Tetra4D Reviewer can load a larger amount of data.
Note: Caching is not recommended if the amount of data is compatible with the physical memory of the computer. As an alternative, consider incremental loading.
- **More** Opens the Enable Cache dialog box. The on Item with More Than option determines the minimum number of faces an entity must have before caching applies.
- **Browse**  Sets the location of the folder in which the cache is placed.
- **Collapsed** When selected, builds only a simplified version of the Model Tree. (It is not built when the data are read). Only one icon appears in the geometry folder, representing the read file (the one that was selected for the reading operation).
- **Component** the Model Tree structure builds only to the component part level.
- **Expanded** (Default) The tree is fully built during the reading operation, including the item parts under the component level.



When caching files in a specific format, deselect Hidden Objects on the Read/Write CAD preferences for that format.

Read/Write CAD preferences

- **File Formats** Lists the CAD and standard formats that Tetra4D Reviewer is currently set up to read or write. You can add and remove formats.

The options under Read Options and Write Options are format specific. Some formats have no associated option, including 3D Studio, HP-GL, Collada, KMZ, and WaveFront. Because options vary according to format, the order in which they appear also varies.

Read options

The Read Options area includes options that filter the types of information that are read when CAD files open in Tetra4D Reviewer. Options are format specific, so the available options change when you select different formats in the File Formats list. Some formats have no reading options.

- **Note:** Options influence the reading time. Depending on the nature and the organization of the read files, the final number of read entities can vary substantially. [Filtering options](#)
- **3D PMI (Text, Notes, GD&T)** Reads PMI markups and notes.
- **Active Filter** Reads only data stored in the active filter or on layers corresponding to the active filter.
- **Attributes** Reads non-geometric information linked to entities and files.
Note: Attributes linked to solids and entities, such as Material and Density, are always read, even if Attributes is deselected.
- **Construction & References** Includes construction entities, such as planes and axes.
- **Filtering of entities** (reading of Wireframe, Surfaces, Solid entities) If unchecked, the corresponding entities are ignored during reading.
- **Hidden Objects** Reading includes any objects that are present but currently hidden in the CAD file.
- **Solids** Reads solid parts.
- **Surfaces** Reads surfaces of parts as entities. **Wireframes** Reads all wireframe entities in the CAD file.
- **3D/Drawing** Sets whether data is read to create a 3D model or a 2D drawing.
- **Datum** Includes construction entities, such as planes and axes, for Pro/ENGINEER format. Relates to the Display Visible Datum option.
- **Default Units** Reads the CAD units as the default for the imported document.
Display Visible Datum (Relates to category-level show/hide options in Pro/ENGINEER files). Reads or ignores items that are visible in the original CAD file.
- **Geometry** Reads available geometry data in JT format files.
Note: JT is a visualization format. By default, Tetra4D Reviewer reads JT files as tessellated data and ignores the ReadingMode setting in the ReadingMode preferences.
Note Texts with Symbol Specifies how note text from IDEAS files appears. When selected, any symbols in the text appear in the read PMI notes. When deselected, definitions substitute for symbols in the text.
- **Read Embedded PRW** Reads any Tetra4D Reviewer documents that are embedded in the PDF. When selected, any session information from the PRW file is read.
- **Tessellation with Setup Criteria (For PDF files with B-rep information)** Specifies whether tessellation uses the current settings in Tetra4D Reviewer or the settings used to create the PDF. When selected, the Tetra4D Reviewer settings are used.











Write options

- **Create Log File** Generates a log of the exporting process.
- **Write Hidden Objects** Includes any hidden entities in the exported file.
- **Version (For AutoCAD DXF format)** Specifies which format version of DXF is written.
- **Write in** For AutoCAD DXF format
- **Paper space** Writes entities to a 2D environment.
- **Model space** Writes entities to a 3D environment.
- **Save Solid to** Specifies how solids are written:
- **Surface** Solids are written as independent faces.
- **Save Faceted to** Entities are faceted or exported as wireframes.
- **Save Analytics to** Determines the format for saving analytic surfaces, such as cylinders, cones, planes, and so forth. NURBS converts all analytics to NURBS surfaces. Analytics keeps analytics as analytics.
- **Black and White** Ignores color characteristics when exporting to PostScript®.
- **Automatic Conversion to PDF (For PostScript)** Automatically writes an PDF from the exported PostScript file.
- **Use Short Names** Shortens entity names to reduce the file size.
- **STEP Standard** for STEP exports, determines whether files are AP203 or AP214.
- **Level of Detail (For STL)** Applies when B-rep sessions are written to STL. Determines the quality used to generate the STL mesh: Extra Low, Low, Medium, High, Extra High, Advanced.
- **Binary File (For STL)** When selected, compresses the STL file to reduce file size.
- **Formats (For Universal 3D exports)** Sets the exported file format for U3D ECMA 1 (Reader 7.0 Compatible) or U3D ECMA 3.
Mesh Quality and Mesh Quality Value (For Universal 3D with U3D ECMA 3 selected)
Determines the quality used to generate the Universal 3D mesh.
- **Attributes** Includes non-geometric information linked to entities and files.

Assembly dialog box

For some formats, an Assembly button appears in the Read Options area. Options in the Assembly dialog box identify the locations that Tetra4D Reviewer searches to load subassemblies and parts for the assembly document.

Note: Options vary according to the selected file format.

- **Recursive Search** Extends the search recursively.
- **Use Root File Directory** Searches subparts in the same directory as the head of the assembly. If not found, they are searched for in the additional search directories. The Browse button  opens a dialog box with additional options:
- **Additional Search Directories** Specifies other areas for the search. Buttons add New , delete , selected directories, and move directories Up  or Down  the search order.
- **Path Definition Files** Specifies the location of text files that define additional paths. Buttons  add New files, delete  selected files from the list, and move the order in which files are listed Up  or Down .
- **Load Product Structure Only** Loads only the Assembly structure (not the parts). Used for incremental loading.
- **Cache Activation (CATIA 5)** Enables caching and activates the path and Browse button for setting the location of the caching folder.
- **Allow Search in Other DDNAME (CATIA V4)** Uses the options set by clicking the Browse button  and setting the DDNAME.
- **Preferred Reference Sets (Unigraphics format only.)** Determines how references sets are determined and in what priority.
- **Apply to All Levels** (Unigraphics format only.)

Syntax for a path-definition file

When you set Assembly option preferences, use the following syntax to create an entry in the Path Definition Files list.

syntax: "path name", [recursive =] 1/0

The keyword "recursive" can be omitted. Values: 1 for recursive mode, 0 for non-recursive.

Example of a path definition file: "c:\data\project\", recursive=1 "f:\parts", recursive=0 "d:\special ",1

Read/Write Image preferences

Preferences on the Read/Write Image tab apply to two-dimensional raster snapshots.

Note: Changes you make to Write options apply only to the selected format and only after you click Apply. If you change several formats before clicking Apply, only the changes to the currently selected format are saved.

- **File Formats** Lists the formats that Tetra4D Reviewer can read and write. When highlighted, any options for that format appear under Write Options. Check boxes determine which formats appear in the Files of Type menu in the Open dialog box.
- **Quality slider (Appears for JPEG only)** Sets the quality for writing JPEG images: Extra Low, Low, Medium, High, Extra High.
- **Extension** Editable menu of the extensions for available formats. You can select an extension from the menu or type a new extension in the box and click Add.
- **Add** Adds an extension that you typed into the extension box.
- **Remove** Deletes the currently shown extension from the extension menu.
- **White Background** Removes any background color in snapshot images. (This option doesn't affect the actual background in the document pane view.)
- **Display Trihedron** Includes the coordinates trihedron in the snapshot.
- **Resolution** Opens the Resolution dialog box, which contains additional options for raster images.

Note: Generating high-resolution images takes time. It can also require large amounts of memory and storage space. For best results, use the minimum resolution that matches the quality requirements for your project.

- **Window** Matches the size of the Tetra4D Reviewer document pane window.
- **Desktop** Matches the resolution of the desktop on your local computer.
- **Menu** Lists various preset resolution settings, including the desktop resolution.
- **Custom** Makes the remaining options available. These options are useful for creating high-quality images at custom resolutions.
- **Width (In Pixels) and Height (In Pixels)** Determine the resolution of the image output.
- **DPI (Dots per inch)** When selected, makes the Width (mm), Height (mm), and DPI value box available. The value sets the resolution for the printed image. Changes to the DPI value remain in effect after you deselect the DPI option.
- **Width (In mm) and Height (In mm)** Dimensions of the print size, calculated according to the selected DPI and the pixel settings for width and height.

Pixel dimensions, millimeter dimensions, and the DPI values are interrelated. Changes you make to one value cause recalculations of the other values. The affected values update when you press Tab or click to shift the focus to another option.

Changes to DPI also change the pixel dimensions.

Changes to a millimeter dimension also change the corresponding pixel dimension.


Changes to a pixel dimension also change the corresponding millimeter dimension. These updates occur even when DPI is not selected.

The user specifies the DPI value, and is never updated when other values are changed.


Changes in dimension values don't affect the DPI value.

Views Management preferences

Split Window options

- **Split Window Buttons** Selecting a button activates the view and grid options for the corresponding split panes.
- **Single Window** Activates menu and grid options for only the first pane of a window split.
- **2-Way Horizontal** Sets the default behavior to create two panes, one above the other. The upper pane is the active view. When selected, view and grid preferences options are available for those two panes.
- **2-Way Vertical** Sets the default behavior to create two side-by-side panes. The left pane is the active view. When selected, view and grid preferences options are available for those two panes.
- **4-Way Split** Sets the default behavior to split the document pane horizontally and vertically, creating four panes. The upper-left pane becomes the active view. When selected, view and grid preferences options are available for all four panes.
- **Select View Type Menus** Sets the default views for each pane in a split. The menu includes the default views and an eye icon , which represents the view in the document pane at the time of the split.
- **Grid buttons** When selected, displays the grid in the selected pane.
- **Restore All** Resets the view menu and grid settings for all four panes. By default, grids are hidden. The upper-left pane shows ZX view (Front). The upper-right pane shows XZ view (Left). The lower-left pane shows YX view (Top). The lower-right pane shows the current view in the document pane at the time of the split.

Standard Views options

- **Select Iso View Menu** Designates which axis is vertical in isometric view. Choose: OZ, OX, or OY. The vertical axis is used as the main axis for isometric view.
- **View Orientation** Sets the orientation for Iso View. The menu choices appear as graphical representations of the relative positions of the x, y, and z axes for the selected Iso View.
Note: The Select Iso View options affect the six default views listed in the lower area of the Views Management preferences. The View Orientation setting affects only isometric view.
- **View Name** Assigns a custom name to the Iso view (optional). The custom name appears as a tool tip for the Iso button  in the Default Views toolbar. If the text field is blank, the default tool tip appears.
V Axis value (Available only when the two-axes option is selected in View Orientation) Sets the angle between Iso View and the vertical axis.
H Axis value (Available only when the two-axes option is selected in View Orientation) Sets the angle between Iso View and the horizontal axis.
- **Individual View Orientation Menus** Define the axis orientation for each set of axes: YX, XY, ZY, ZX, XZ, and YZ axes.
- **Individual View Name Options** Define custom names for the individual view orientations. The custom names appear as a tool tips for the Default Views toolbar buttons.

Chapter 8: Shortcuts

Keyboard shortcuts are combinations that you type to apply various commands. Many shortcuts appear on the main menus, opposite the commands to which they apply.

Shortcuts for Tetra4D Reviewer™

Result	Shortcut
New	Ctrl + N
Open	Ctrl + O
Import	Ctrl + I
Save	Ctrl + S
Save As	Shift + Ctrl + S
Export	Ctrl + E
Close window	Ctrl + W
Close all window s	Shift + Ctrl + W
Print	Ctrl + P
Print Setup	Shift + Ctrl + P
Exit	Ctrl + Q
Copy selected items	Ctrl + C
Paste copied items	Ctrl + V
Delete	Del
Select All	Ctrl + A
Deselect All	Shift + Ctrl + A
Toggle selection	Ctrl + T
Find	Ctrl + F
Preferences	Ctrl + K
Show or hide the grid	Ctrl + U
Activate previous configuration	F2
Activate next configuration	F3
Quick Snap	F8
Show or hide the Model Tree	F4
Restore default toolbars	Alt + F8
Full Screen	Ctrl + L
Cascade windows	Shift + Ctrl + J
Tile windows horizontally	Shift + Ctrl + K

Tile windows vertically	Shift + Ctrl + L
Focus on selected item	F5
Fit Visible	F6
Set camera from selection	F7
Flip	Shift + F7
Undo View operation	Alt + Z
Redo View operation	Alt + Y

Shortcuts for Fly mode

For Fly mode, you can use the default shortcuts or define custom key combinations for the same items.

Result	Default shortcut
Front	R
Back	F
Left	Q
Right	D
Up	Z
Down	S
Roll left	A
Roll right	E
Stop	spacebar
Speed up	Num + [plus, on numeric keypad]
Speed down	Num - [minus, on numeric keypad]
Wheel back	Ctrl

Assign custom shortcuts for Fly mode

1. Choose Edit > Preferences, and select the Fly Through tab.
2. Type to replace a keyboard shortcut with a different key.

Note: When the new key is pressed, its value is directly assigned to the active field. Each key assignment is unique. If a key is already used for another function, no change occurs.

Restore default keyboard shortcuts for Fly mode

1. On the Fly Through preferences tab, click Restore Default Keys.

Shortcuts for Help

Result	Shortcut
Open Help window	F1
Close Help window	Ctrl + W
Move back to previously opened topic	Alt + Left Arrow
Move forward to next topic	Alt + Right Arrow
Move to next pane	Ctrl + Tab
Move to previous pane	Shift + Ctrl + Tab
Move focus to next link within a pane	Tab
Move focus to previous link within a pane	Shift + Tab
Activate highlighted link	Enter
Print Help topic	Ctrl + P