یکپارچه سازی مدیریت زمان و هزینه با استفاده از نرم افزارهای Navisworks & Synchro

Kambiz Kordani PMP,MSc
Linkedin: @kordani-kambiz
Instagram: @acemi_social

۱. مفاهیم کلی و مقدمات

www.dralavipour.com ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

Problems owners face

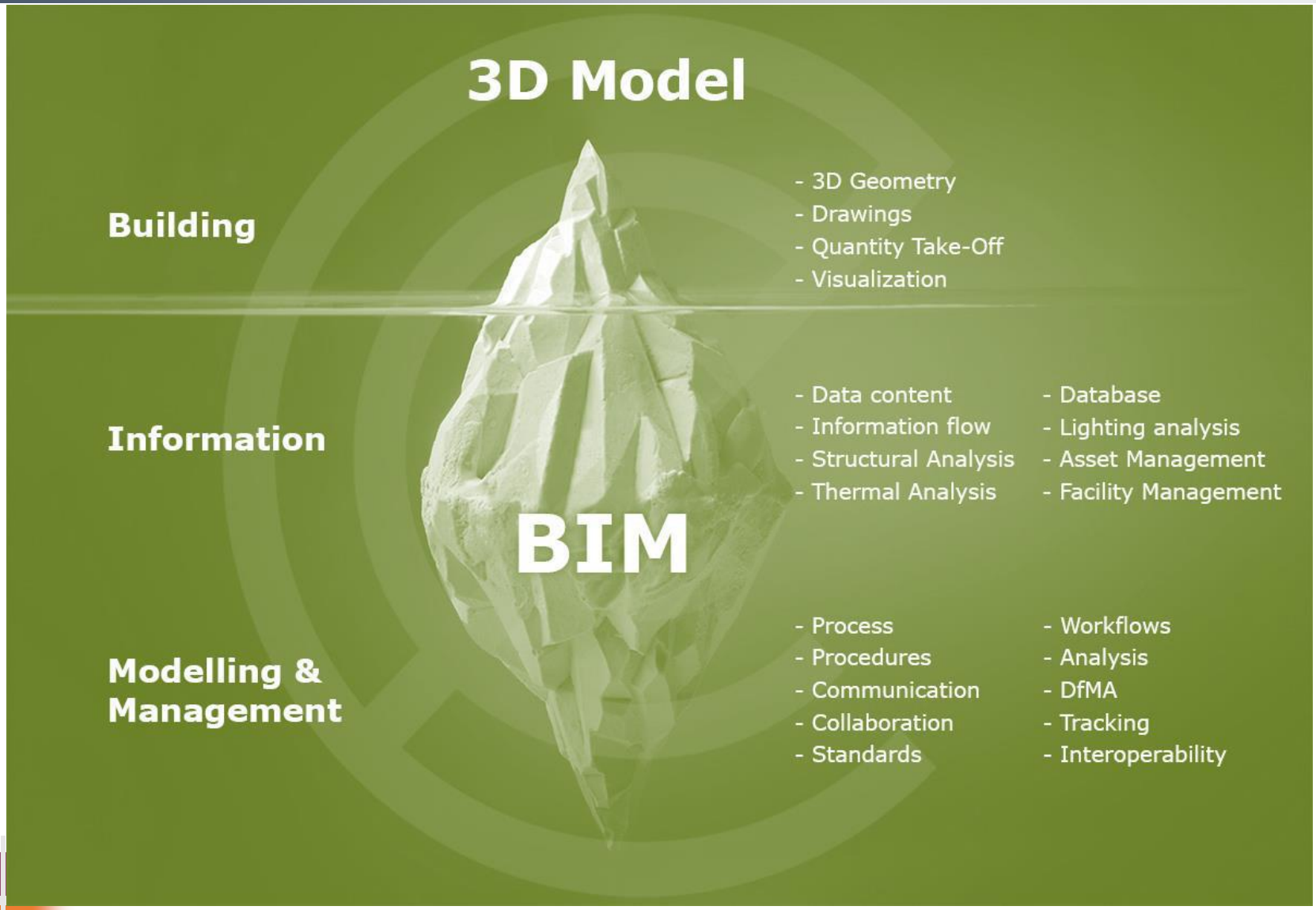
Over **60%** of major capital programs fail to meet cost and schedule targets

30% of construction cost is rework

55% of maintenance remains reactive

Property of Kambiz Kordani PMP, MSc
Do Not Distribute

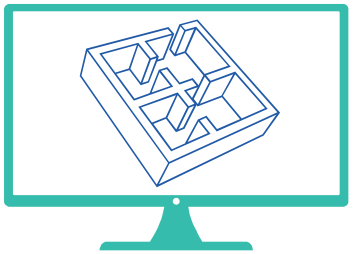
Difference Between 3D Model & BIM



ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

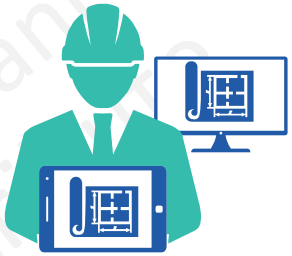
BIM - Building Information Modeling

- A foundational, intelligent model-based process for business and industry transformation



Uses 3D models to capture, explore, and maintain consistent and coordinated planning, design, construction, and operational data

Provides greater project insight for cost, schedule, and constructability



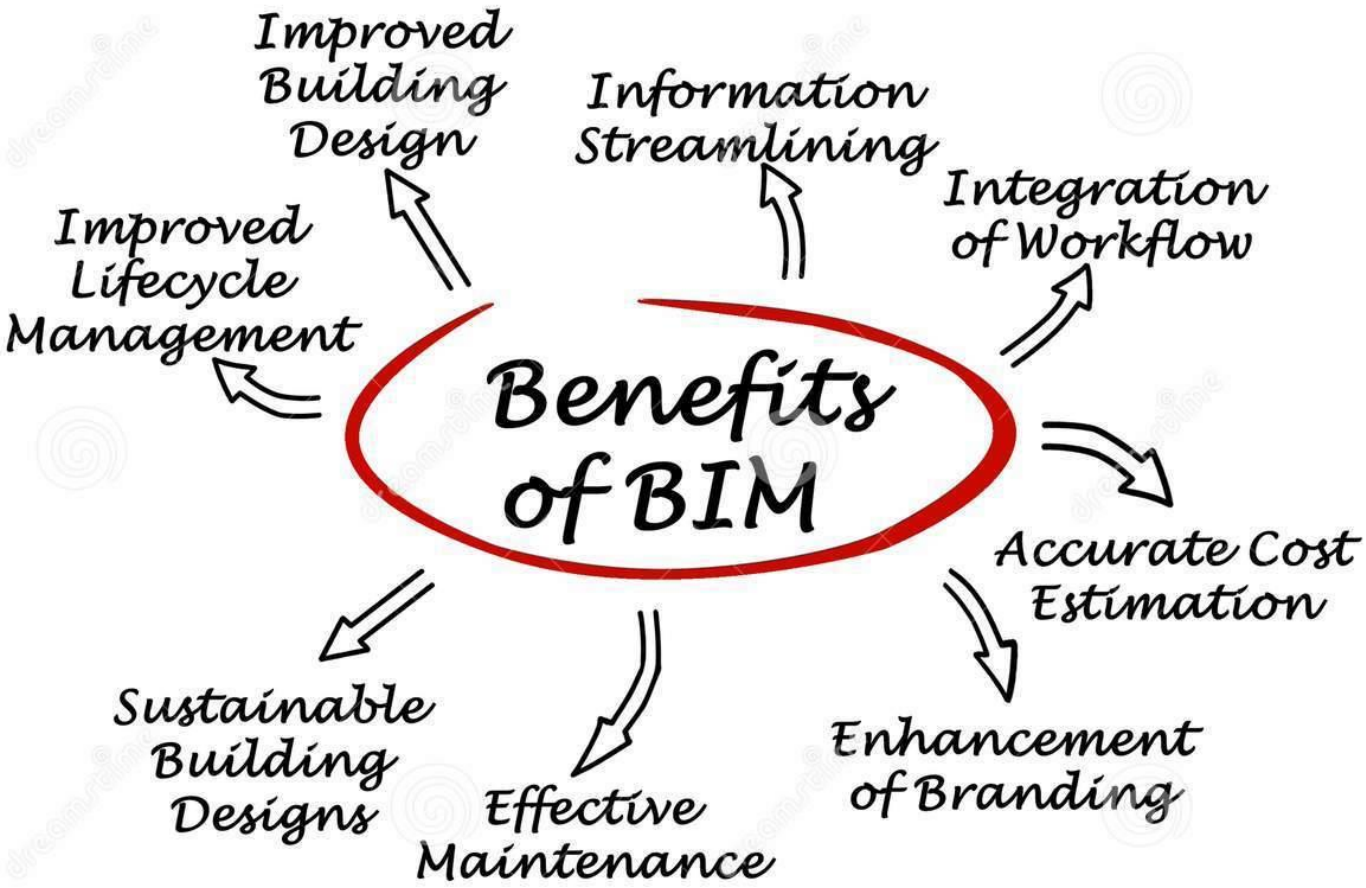
Uses and shares the same consistent data whether you're at your desk or in the field

Enables prompt response to change with processes that are smarter and faster

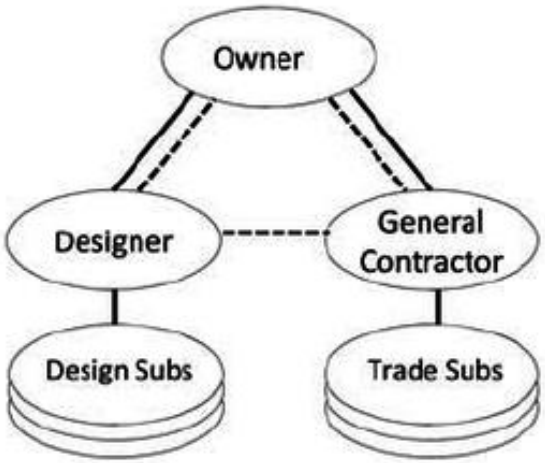
Prepared By: K.Kordani PMP, MSc
Do Not Distribute

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

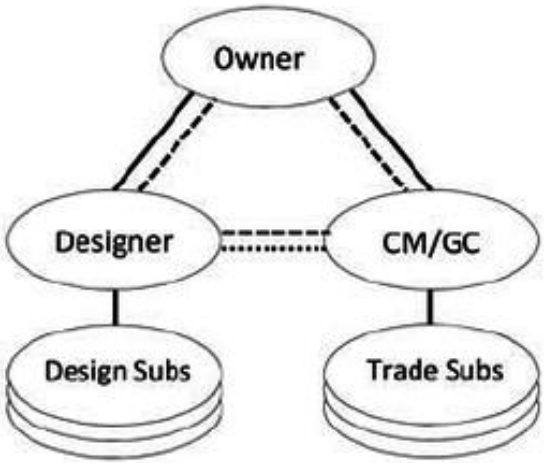
BIM - Advantages



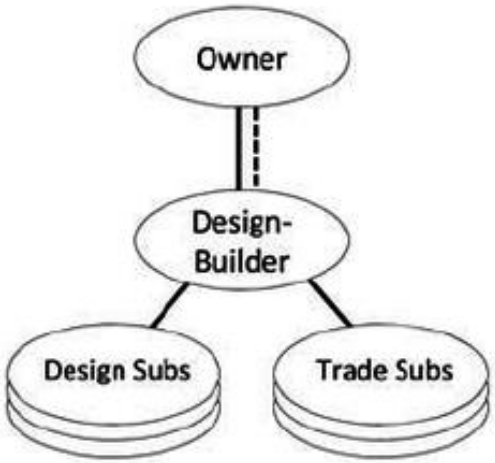
Project Delivery System



Design-Bid-Build



Construction Manager/
General Contractor



Design-Build

Contracts
Communications
Contractually required coordination

Project Delivery System

Three types of clashes that can occur in the construction process:

- In the construction process which requires collaboration between various disciplines namely architecture, structure and MEP there will be some clashes.
- It may be an air-conditioning duct running through a load-bearing wall.
- the lack of clearance around an important component or even a work scheduling conflict.
- During the building process the breath of information that any project member must account for is substantial and very much vulnerable to human error.

Prepared By Kambiz Kordani PMP, MSc
Do Not Distribute

1. Hard Clash:

- Hard Clash is a type of conflict which involves geometrical issues.
- A hard clash occurs when two objects pass through each other or are taking up the same place.
- In Architectural BIM Services they can also be due to modeling errors like bad connection between the partition and the wall or placing the toilet too close to the door.
- A hard clash is easy to spot because of the logical arrangement of elements and their null dynamics.

Prepared By Kambiz Kordani PMP, MSc
Do Not Distribute

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2. Soft Clash:

- Soft clashes are defined by clearances and it occurs when any object either encroaches into geometric tolerances set for other objects.
- Any building object has spatial or geometrical tolerance when that buffer zone is breached a soft clash is detected.
- Navisworks identifies soft clash as per applicable regulations and standards.

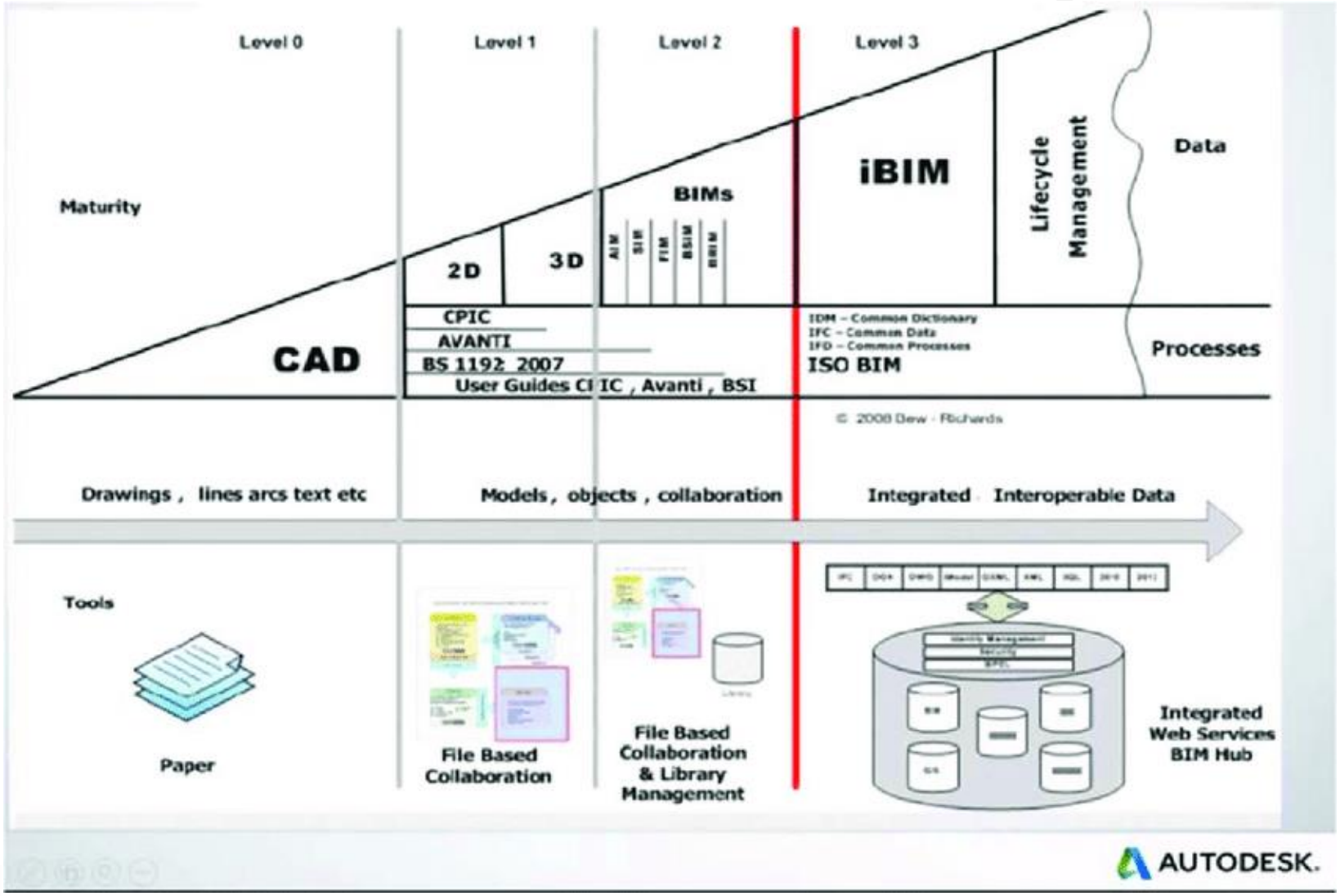
Prepared By Kambiz Kordani PMP, MSc
Do Not Distribute

3. Work-Flow Clash:

- Work-flow clash or 4D clash involves clashes related to contractor scheduling, equipment and material delivery and general workflow timeline conflicts.
- Even though these problems are not directly connected with design but they can significantly affect the ongoing work timeline.
- A workflow clash not only disturbs the project efficiency but also can have a domino effect
- Navisworks allows to develop a **4D construction** sequencing timeline which points out the start and end date of each activity and thus helps to schedule each activity.

Prepared By Kambiz Kordani PMP, MSc
Do Not Distribute

BIM maturity level



ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

4D BIM

- 4D BIM is adding scheduling information to model construction sequences. Adding a dimension of time allows the project team to better visualize how the construction will be sequenced.

- From a contractor point of view, this is vital. 4D BIM was a huge step forward for the industry when first made possible through the use of new modelling tools – it demonstrated collaboration between the design and construction team through coordination and sharing of 3D models.

Prepared By Kambiz Kordani PMP MSc
Do Not Distribute

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

Why you should consider 4D BIM Technology?

- Builders and contractors have several construction sequencing options to choose from, including the simple and conventional 4D scheduling techniques, which can help them, keep construction projects on schedule.
- It provides several prominent alternatives for addressing the project requirements and to overcome any challenges. With various capabilities, 4D BIM supports furthermore the accurate information collection process.

Prepared By Kambiz Kordani PMP, MSc
Do Not Distribute

Why you should consider 4D BIM Technology?

- In more detail, 4D BIM can do the following for your organization:
  - Clash-free Construction Scheduling
  - 4D Graphical Visualization
  - Optimal Resource Utilization
  - Better Interdisciplinary Coordination

Prepared By Kambiz Kordani PMP, MSc
Do Not Distribute

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

How a UK Based Architectural Firm has Leveraged 4D BIM Technology

- A UK based architectural firm had challenges of inter-woven building plans at five different levels and modeling exact elements was a huge challenge. The technical instructions were in foreign language and MEP drawings lacked dimensions for pipe and systems.

- Hitech CADD Services developed a federated/coordinated 4D BIM model for construction scheduling using Revit and Navisworks respectively. Interdisciplinary clashes were detected and eliminated to develop a coordinated model.

- A clash free federated 4D Revit BIM model was delivered to client accompanied with a project introduction video having photo realistically rendered models, which helped the client gain design intent, save costs and rework.

Prepared By Kambiz Kordani PMP, MSc
Do Not Distribute

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

5D BIM

- 5D BIM is generally considered to be adding cost information to a model. If discussing 5D BIM, it is advised to clearly set out these specific requirements.
- For example, is the team expected to be providing capital or operational costs? Are these costs expected to be pre-tender estimates or a record of as-built costs? Who is responsible for adding this information? What method of measurement is to be used?

Prepared By Kambiz Kordani PMP, MSc
Do Not Distribute

Why should you consider 5D BIM Technology?

- 5D BIM, is the next step up from 4D (time scheduling), but furthermore integrates design with estimates such as for and cost, and includes the generation of quantities (BOMs/BOQs), development of rates and overall cost.
  - Enhanced Cost Forecast and Estimation
  - More Accurate and Faster Quantity Take offs
  - Automated Quantity take offs as per Design Modifications
  - Easy Selection of available Options
  - Multi-Platform Access

Prepared By Kambiz Kordani PMP MSc
Do Not Distribute

Other benefits of 5D BIM includes:

- Visual and data modeling capabilities open up new avenues for predictability resulting in a more efficient project scope. It drives costs based on specific designs, materials, site conditions, phasing and various other parameters responsible for producing accurate information within project timelines.

- It helps the entire project team to develop a clear understanding of the proposed design, assumptions and cost drivers. Before the project begins, the entire team would have a concise and accurate description of scope, cost, and schedule.

- Relevant data when updated, automatically changes the construction costs in real-time. It helps in eliminating last minute surprises that directly or indirectly impact funding decisions.

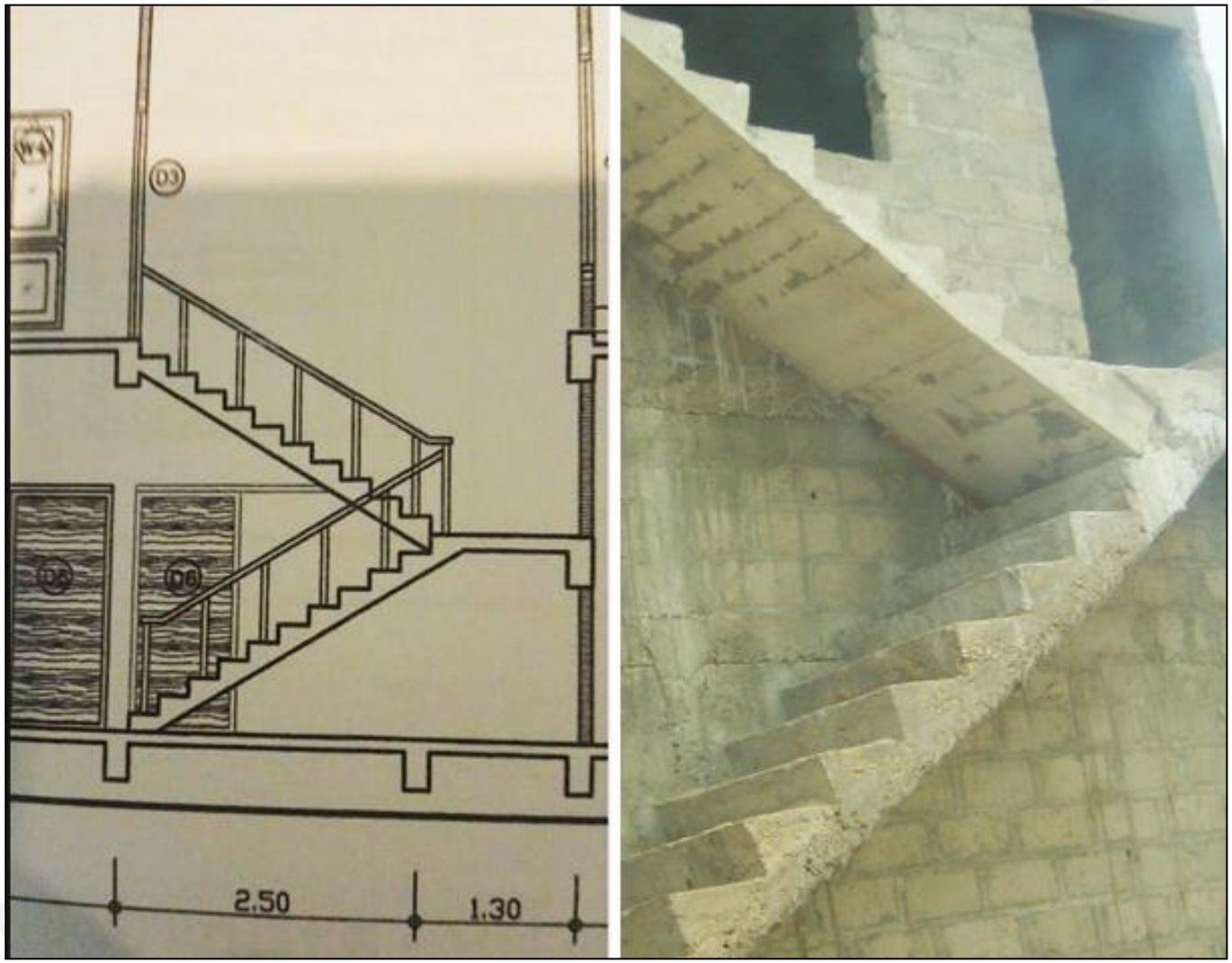
Prepared By Kambiz Kordani PMP, MSc
Do Not Distribute

Other benefits of 5D BIM includes:

- Relevant data is shared on real time basis, as and when the model either develops further or any changes are made. Exploring alternative concepts becomes possibly easier, whereas the project team and owners get to see the cost and other data updates in real-time.
- Design and estimation cycle time reduces significantly, as compared to conventional pre-construction and early design stages.

Prepared By Kambiz Kordani PMP, MSc
Do Not Distribute

Funny Example of Not Collaborating Effectively



Funny Example of Not Collaborating Effectively

Prepared By

Funny Example of Not Collaborating Effectively

Challenges in Effective Collaboration

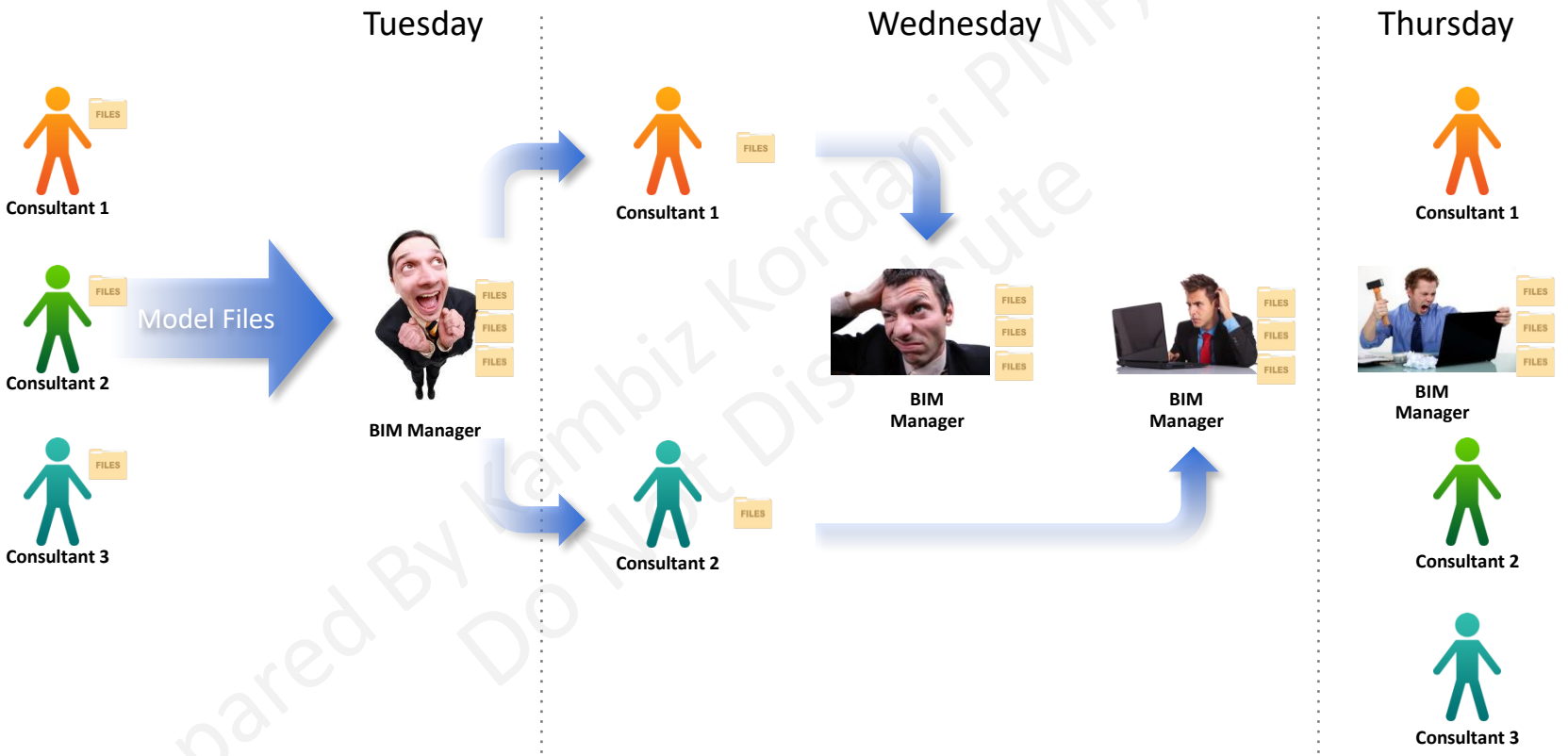
ACEMI

In the industry, Confusion about

Autodesk NAVISWORKS

Prepared By: Kambiz Kordani PMP, MSc
Do Not Distribute

A Typical Week in a Coordination Process



Prepared By Kambiz Kordani PMP, MSc
Do Not Distribute

یکپارچه سازی مدیریت زمان و هزینه با استفاده از نرم افزارهای

Navisworks &

Synchro

Kambiz Kordani PMP,MSc

Linkedin: @kordani-kambiz

Instagram: @acemi_social

۲. مدلسازی اطلاعات ساخت با استفاده از نرم افزار Navisworks

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است. www.dralavipour.com

آموزش نصب و فعال سازی نرم افزار

- نرم افزار را نصب کنید.
- فایل فعال ساز را دانلود و از حالت فشرده خارج کنید.
- فایل Autodesk License Patcher.exeرا اجرا کنید و چند ثانیه صبر کنید تا مراحل انجام شود و پنجره فعال سازی بسته شود.
- نرم افزار را اجرا کنید و در پنجره فعال سازی گزینه Use A Network Licenseو بعد Single License Serverرا انتخاب کنید و در کادر خالی 127.0.0.1 را وارد کنید.
- برای نصب سایر محصولات Autodeskنیاز نیست مجدد مراحل بالا را انجام دهید. کافی است نرم افزار را نصب و اجرا کنید و در پنجره فعال سازی مرحله بالا (4) را انجام دهید.
- این روش ساده ترین راه برای فعال سازی محصولات نرم افزاری Autodeskمیباشد.

Prepared By Komeiz Kordani PMP, MSc
Do Not Distribute

۲. مدلسازی اطلاعات ساخت با استفاده از نرم افزار Navisworks

Kambiz Kordani PMP,MSc
Linkedin: @kordani-kambiz
Instagram: @acemi_social

2.1 Introduction to Autodesk Navisworks

- Introduce you to Autodesk Navisworks.
- Familiarize you with the Autodesk Navisworks interface.
- Explain various Autodesk Navisworks file types.
- Teach you various means of collaborating designs in Autodesk Navisworks.
- Teach you to publish a password-protected file that can be opened in the free Autodesk Navisworks viewer.
- Teach you various navigation tools available in Autodesk Navisworks.

Kambiz Kordani PMP MSc Do Not Distribute

www.dralavipour.com ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.1 Introduction to Autodesk Navisworks

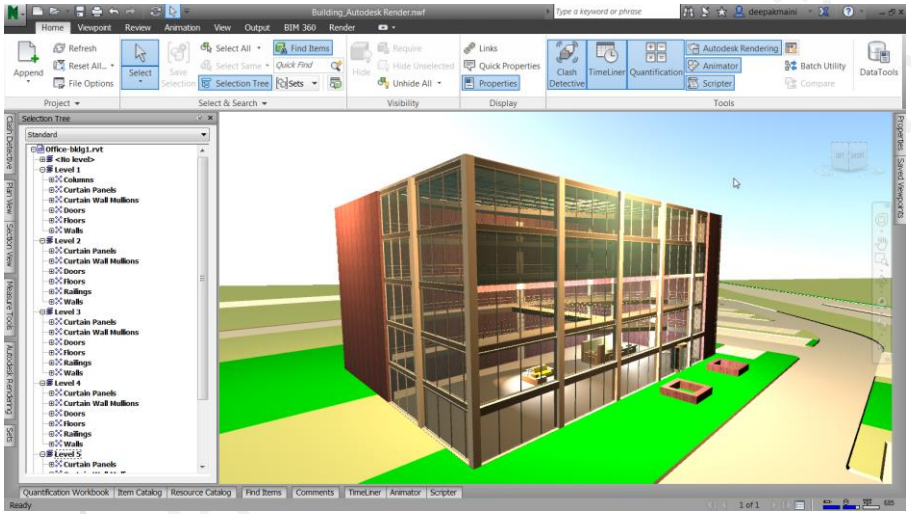
- Introduce you to Autodesk Navisworks.
- Familiarize you with the Autodesk Navisworks interface.
- Explain various Autodesk Navisworks file types.
- Teach you various means of collaborating designs in Autodesk Navisworks.
- Teach you to publish a password-protected file that can be opened in the free Autodesk Navisworks viewer.
- Teach you various navigation tools available in Autodesk Navisworks.

Prepared By Kambiz Kordani PMP, MSc
Do Not Distribute

2.1 Introduction to Autodesk Navisworks

AUTODESK NAVISWORKS

Autodesk Navisworks is a 3D design collaboration tool that lets you import models from various design platforms to create an aggregated design for visualization and review. The resulting design is a virtual model in which you can navigate in real-time and create walkthroughs that allow better understanding of the design intent. This virtual model can be shared with various teams to find and fix problems, thus allowing for a more effective management and coordination of the project.



2.1 Introduction to Autodesk Navisworks

AUTODESK NAVISWORKS PRODUCTS

Autodesk Navisworks has been bundled in various Autodesk Suites in recent years. But in terms of product offerings, Autodesk Navisworks offers the following three flavors:

Autodesk Navisworks Simulate

This is the base level of Autodesk Navisworks products. It includes the following modules:

- **Publisher**: *To publish NWD files that can be viewed in a free viewer called Autodesk Navisworks Freedom.*
- **Autodesk Rendering**: *To create high quality photorealistic visualizations of the Autodesk Navisworks scene.*
- **TimeLiner**: *To create simulation of construction schedules.*
- **Animator and Scripter**: *To create movement of objects in the scene.*
- **Quantification**: *To perform quantity takeoff from the Autodesk Navisworks scene. This module was introduced in Release 2014.*

2.1 Introduction to Autodesk Navisworks

Autodesk Navisworks Manage

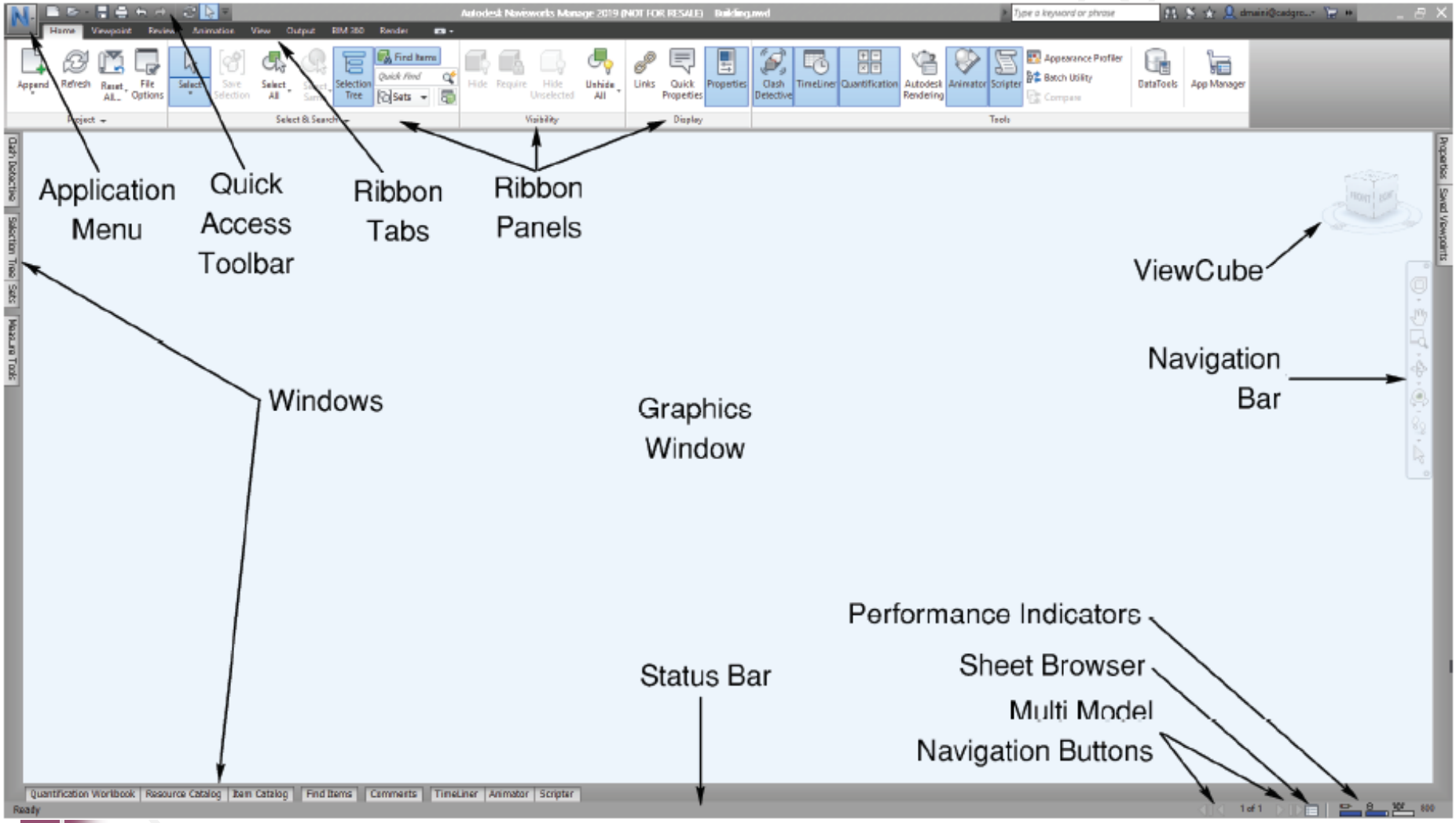
This is the highest level of Autodesk Navisworks products. It includes all the modules that Autodesk Navisworks Simulate has and also the following module:

- **Clash Detective**: *To perform and report interferences between different components of the Autodesk Navisworks scene.*

Autodesk Navisworks Freedom

This is the free viewer that can be used to view the published Autodesk Navisworks model. This tool is generally used by various stakeholders to analyze, interrogate, and communicate the design intent at various stages of the design process. It can be downloaded from the Autodesk Navisworks Web site http://www.autodesk.com/products/navisworks/autodesk-navisworks-freedom.

2.1 Introduction to Autodesk Navisworks



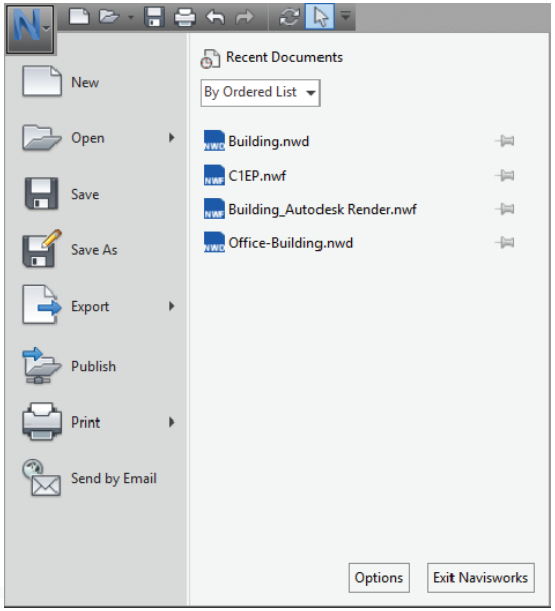
2.1 Introduction to Autodesk Navisworks

Autodesk Navisworks Interface Components

The following are the various components of Autodesk Navisworks interface:

Application Menu

This button is used to display the **Application Menu**, which is divided into two areas, as shown in Figure 4. The area on the left shows standard tools such as **Save**, **Save As** etc. The area on the right shows the recent files that you opened in Autodesk Navisworks. The buttons on the bottom right of this area can be used to change Autodesk Navisworks global options and to close this program.



Prepared By Kambiz
Do Not Distribute

2.1 Introduction to Autodesk Navisworks

Ribbon Panels

These are logical groups of Autodesk Navisworks tool buttons. Some ribbon panels have a down arrow at the bottom that is used to expand the panel and show additional tool buttons.

Graphics Window

This is the area where the design is displayed and manipulated.

ViewCube

This is a visualization tool used to display the static views of the model from various directions. You can click on one of the faces or the edges of the ViewCube to display the model from that particular direction. You can also click on one of the corners of the ViewCube to display the 3D view of the model from that direction. This visualization tool is discussed in detail later in this chapter.

Navigation Bar

Various drawing display tools available in Autodesk Navisworks are located on this toolbar.

Status Bar

This bar runs through the bottom of the Autodesk Navisworks window. The left side of this bar shows a prompt sequence related to some tools in Autodesk Navisworks, The right side of this bar has performance indicators that are discussed below.

2.1 Introduction to Autodesk Navisworks

Sheet Browser

This button is used to turn on the **Sheet Browser** palette.

Multi Model Navigation Buttons

If you have loaded multiple models in the **Project Browser**, you can use these buttons to switch between them.

Performance Indicators

The right side of the status bar hosts performance indicators, which are discussed next.

Pencil Indicator Bar

Generally, when you are zooming, panning or walking in a large design, depending on your machine performance, the entre view is not loaded for every frame. This bar shows how much of the view is currently loaded. In other words, it indicates that there are objects dropped from the current display.

Disk Indicator Bar

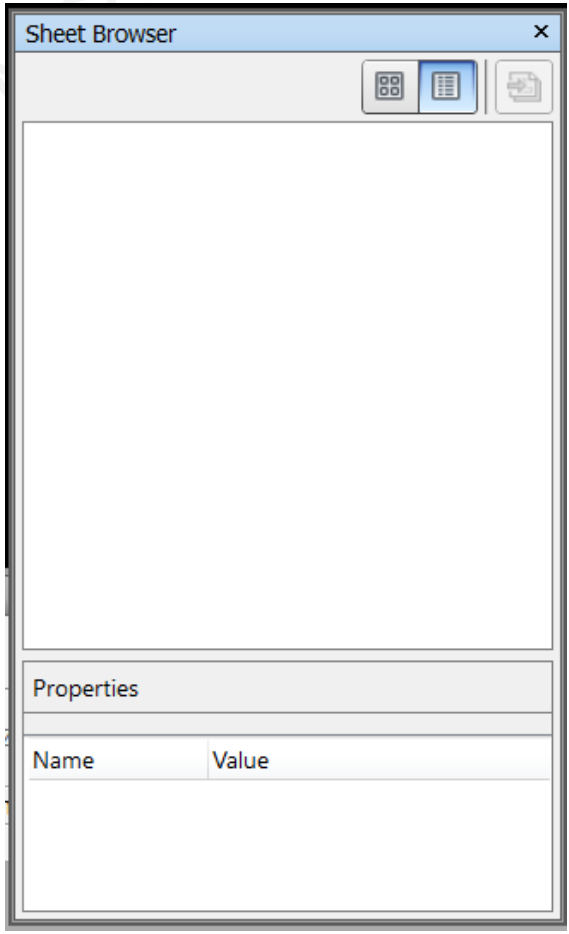
This is the second indicator bar and is used to show how much of the current model is loaded from the disc into the memory.

Web Server Indicator Bar

This indictor bar is used only when you are opening a model from a web server, a URL or BIM 360 Glue. It indicates how much of the model is downloaded from the Web server into the memory. If the model is not opened from a web server, this indicator bar will remain white.

Memory Indicator

This indictor shows how much memory, in Megabytes, is being currently used by Autodesk Navisworks.



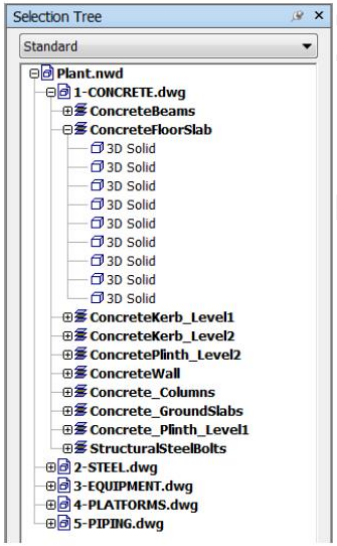
2.1 Introduction to Autodesk Navisworks

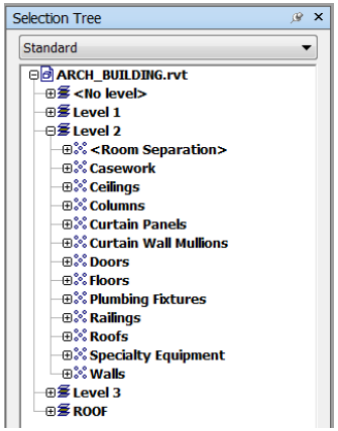
Autodesk Navisworks Windows

For the ease of performing various tasks, Autodesk Navisworks provides a number of windows that can be turned on or off, depending on their requirements. The following are some important windows that you will need while working with Autodesk Navisworks.

Selection Tree

The **Selection Tree** can be turned on or off using its button on the **Select & Search** panel of the **Home** ribbon tab. This tree is used to select the components in the current design. There are three displays of this tree; **Standard**, **Compact**, and **Properties**.



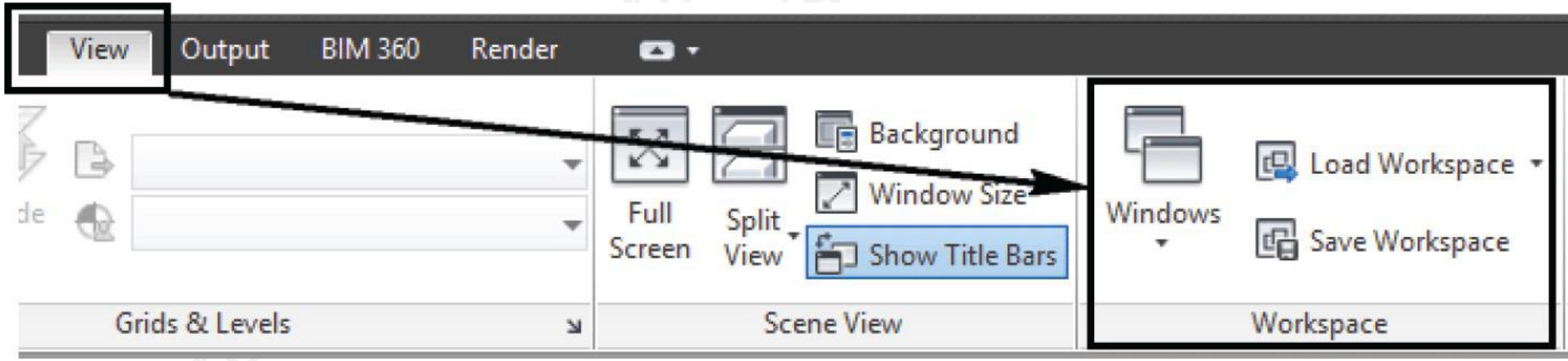


ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.1 Introduction to Autodesk Navisworks

- Turning various windows on and off regularly is a tedious job. As a result, Autodesk Navisworks has a really smart concept of workspaces, which controls the visibility and location of various windows on the screen.

- You can turn on the visibility of the windows that you often use and then save the screen display as a workspace.



2.1 Introduction to Autodesk Navisworks

Safe Mode

This workspace loads the **Selection Tree** and **Plan View** windows docked on the left of the screen, the **Comments** window docked at the bottom of the screen, and the **Tilt Bar**, **Properties**, and **Saved Viewpoints** windows docked on the right of the screen.

Navisworks Minimal

This is the default workspace and has no window loaded on the screen.

Navisworks Extended

This workspace is similar to the **Safe Mode** workspace but has additional windows such as **Clash Detective**, **Sets**, and **Measure Tool** windows docked to the left in the auto-hide mode. In the auto-hide mode, the window slides out when you move the cursor over it. In addition, this workspace also has the **Find Items**, **Comments**, **TimeLiner**, **Animator**, and **Scripter** windows docked at the bottom of the screen in the auto-hide mode. Figure 9 shows the Autodesk Navisworks screen in this workspace.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

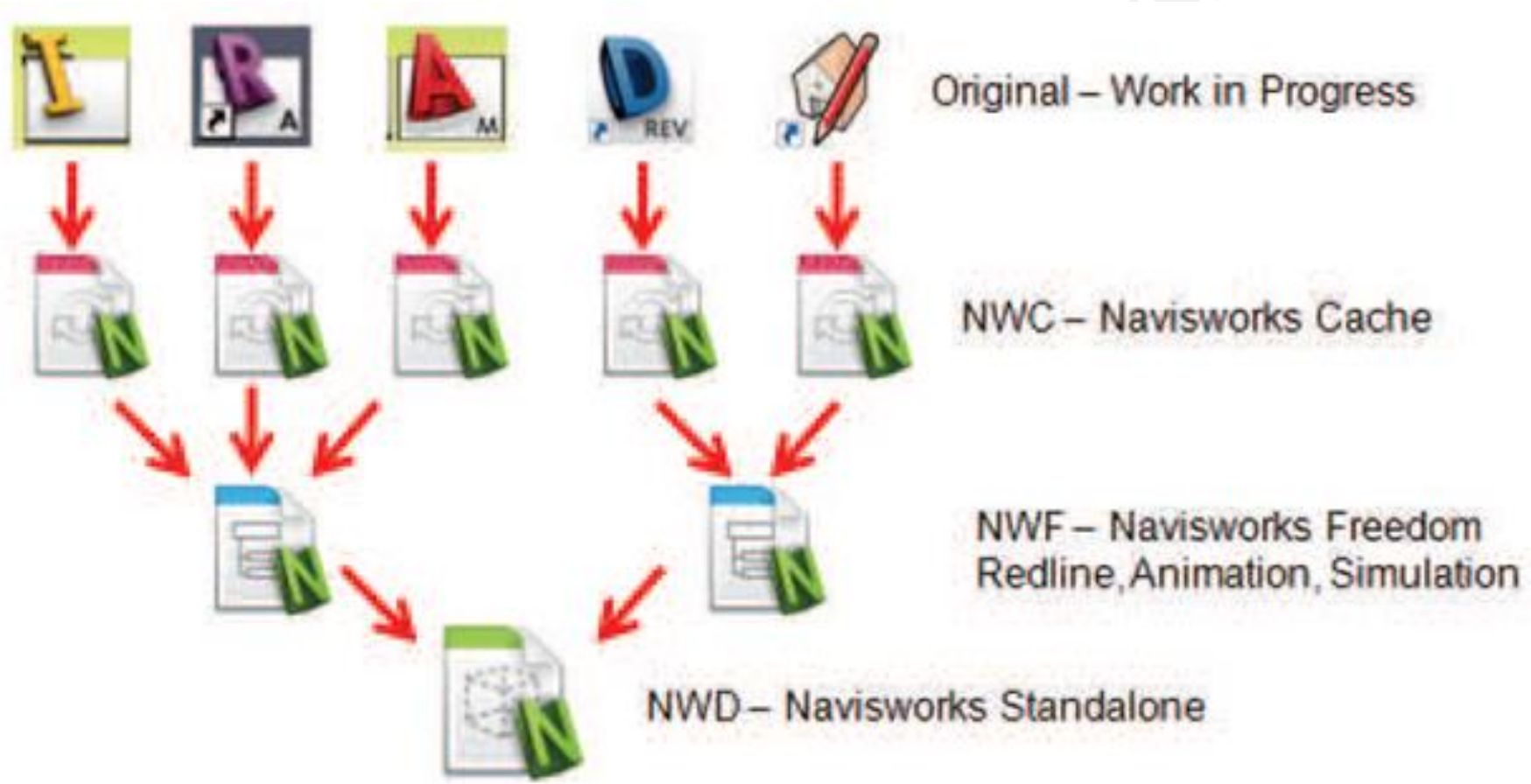
2.1 Introduction to Autodesk Navisworks

Creating a User-Defined Workspace

To create your own workspace, simply turn on the visibility of the windows that you need from the **Home** ribbon tab or from the **View** ribbon tab > **Workspace** ribbon panel > **Windows** flyout. Alternatively, you can activate the **Navisworks Extended** workspace and then turn off the windows you do not need. Once the Autodesk Navisworks interface looks like what you need, click on the **View** ribbon tab > **Workspace** ribbon panel > **Save Workspace** button; the **Save Workspace** dialog box will be displayed. By default, the workspace is saved inside your roaming profile folder. Specify the workspace name and then click on the **Save** button; the workspace will be saved as a .XML file. This workspace will now be listed in the **Load Workspace** flyout.

Prepared By Kambiz
Do Not Distribute

2.1 Introduction to Autodesk Navisworks

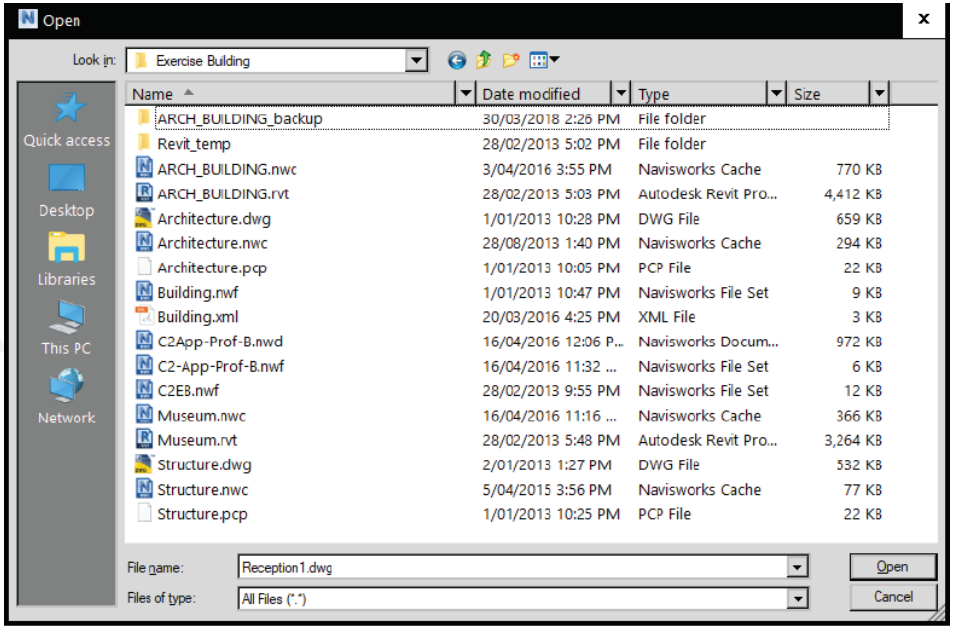


2.1 Introduction to Autodesk Navisworks

When starting the collaboration process from scratch, the first file is opened using the **Open** tool and the remaining files are aggregated using the **Append** tool. Both these tools are discussed below.

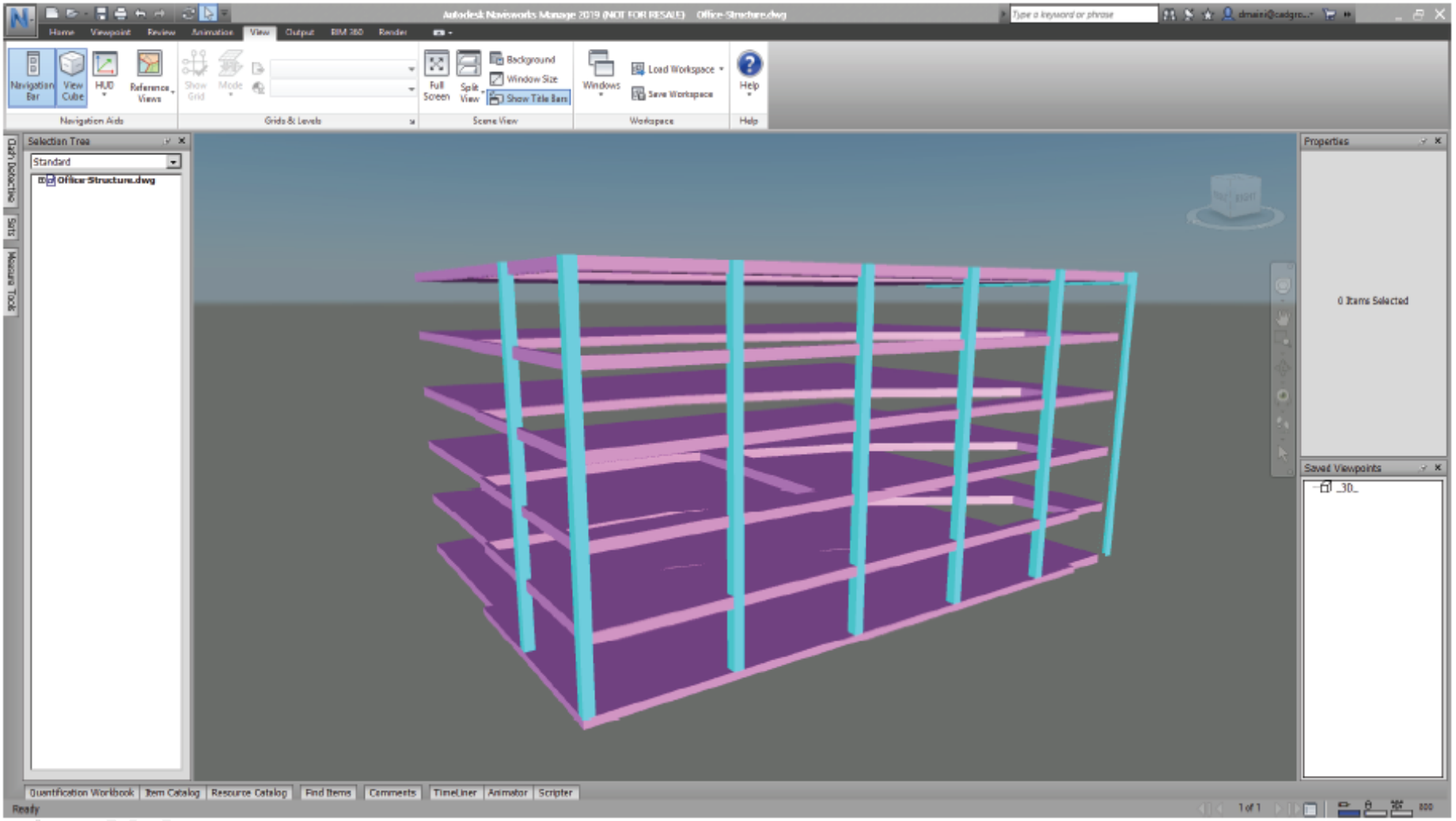
Opening Files

Quick Access Toolbar > Open
Application Menu > Open



Prepared By Kambiz
Do Not

2.1 Introduction to Autodesk Navisworks



www.dralavipour.com ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.1 Introduction to Autodesk Navisworks

Opening from Web Server

Application Menu > Open Flyout > Open URL

Appending Files

Quick Access Toolbar > Open Flyout > Append
Home Ribbon Tab > Project Ribbon Panel > Append

Merging Files

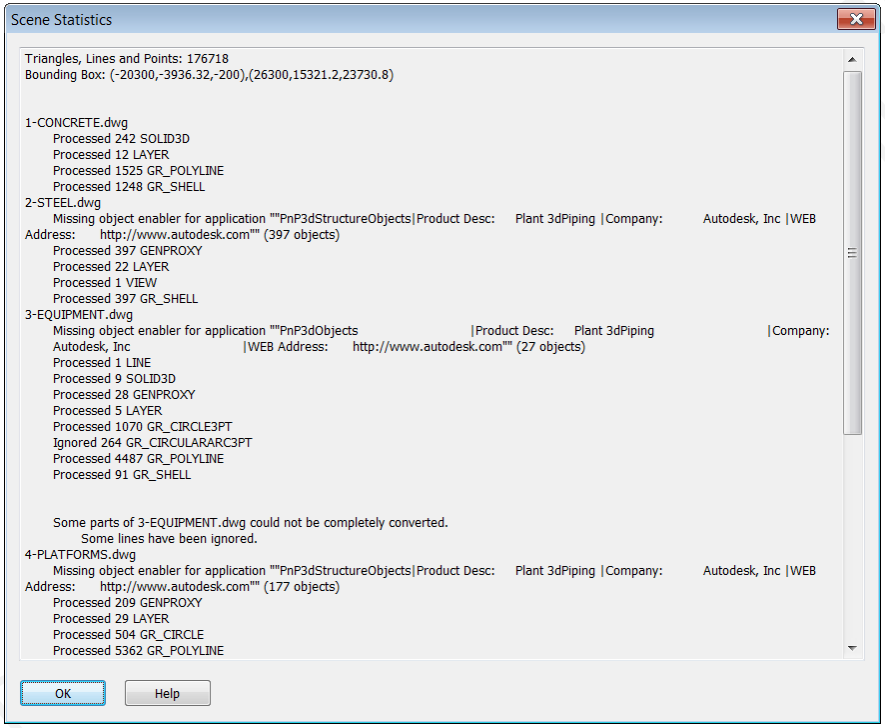
Quick Access Toolbar > Open Flyout > Merge
Home Ribbon Tab > Project Ribbon Panel > Append Flyout > Merge

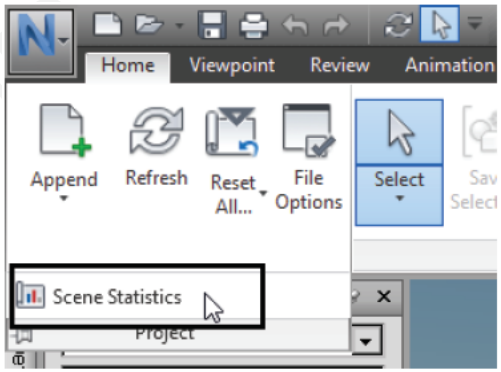
ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.1 Introduction to Autodesk Navisworks

SCENE STATISTICS

Home Ribbon Tab > Project Ribbon Panel > Scene Statistic

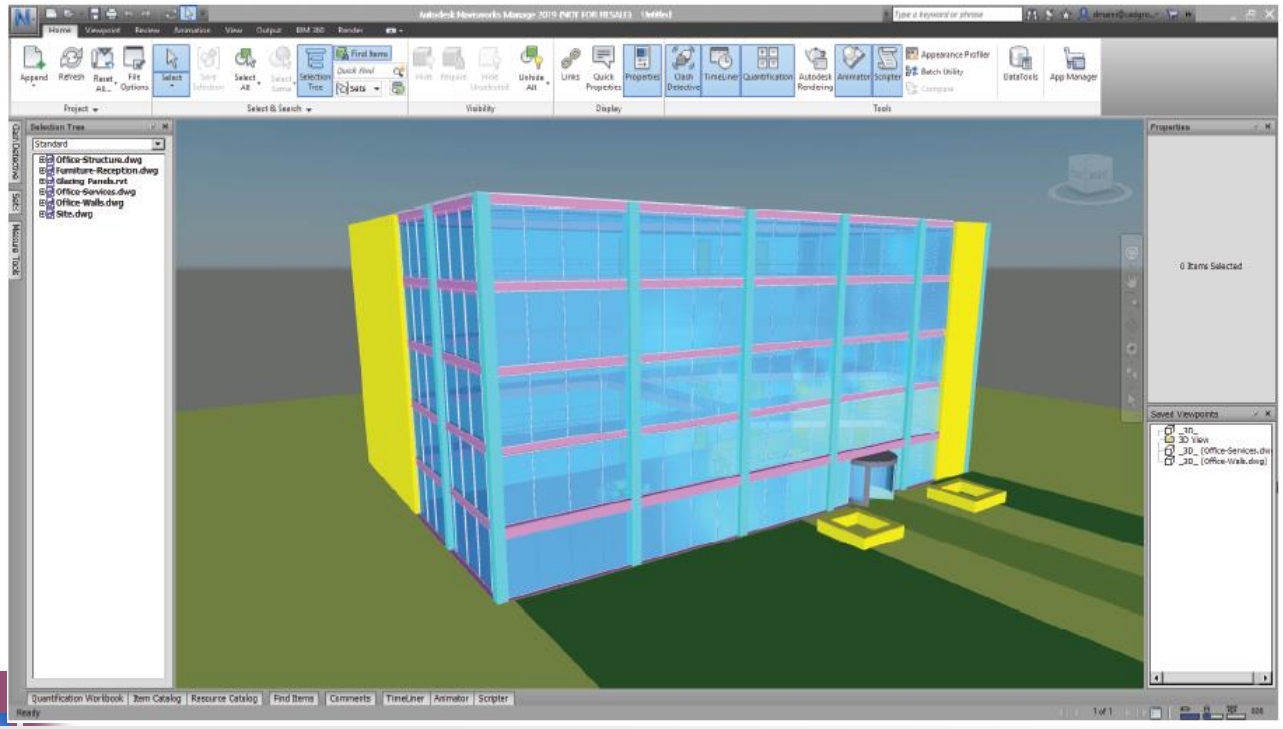


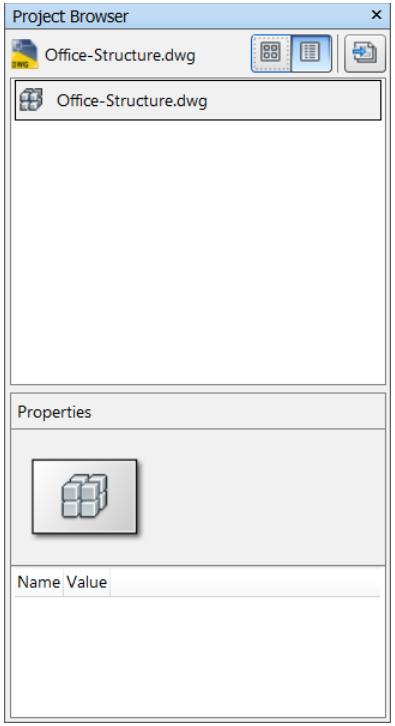


2.1 Introduction to Autodesk Navisworks

SHEET BROWSER

Status Bar> Sheet Browser
View Ribbon Bar > Workspace Ribbon Panel > Windows Flyout





2.1 Introduction to Autodesk Navisworks

SAVING YOUR WORK

It is important to regularly save your work to avoid any unexpected loss of data because of machine problems. Autodesk Navisworks provides you three tools for manually saving your work and also an option to auto-save the files. All these are discussed next.

Save Tool

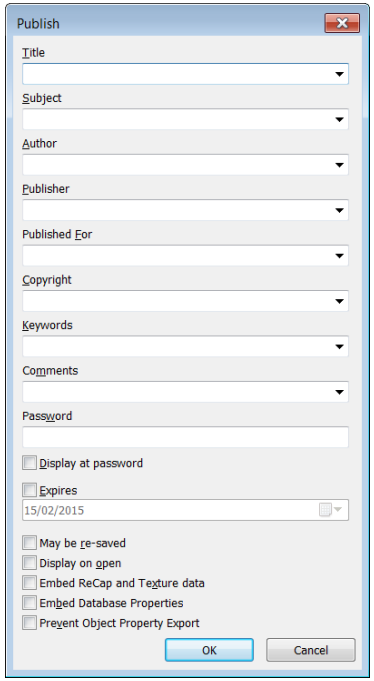
Quick Access Toolbar > Save
Application Menu > Save

Save As Tool

Application Menu > Save As

Publish Tool

Application Menu > Publish
Output Ribbon Tab > Publish Ribbon Panel > NWD

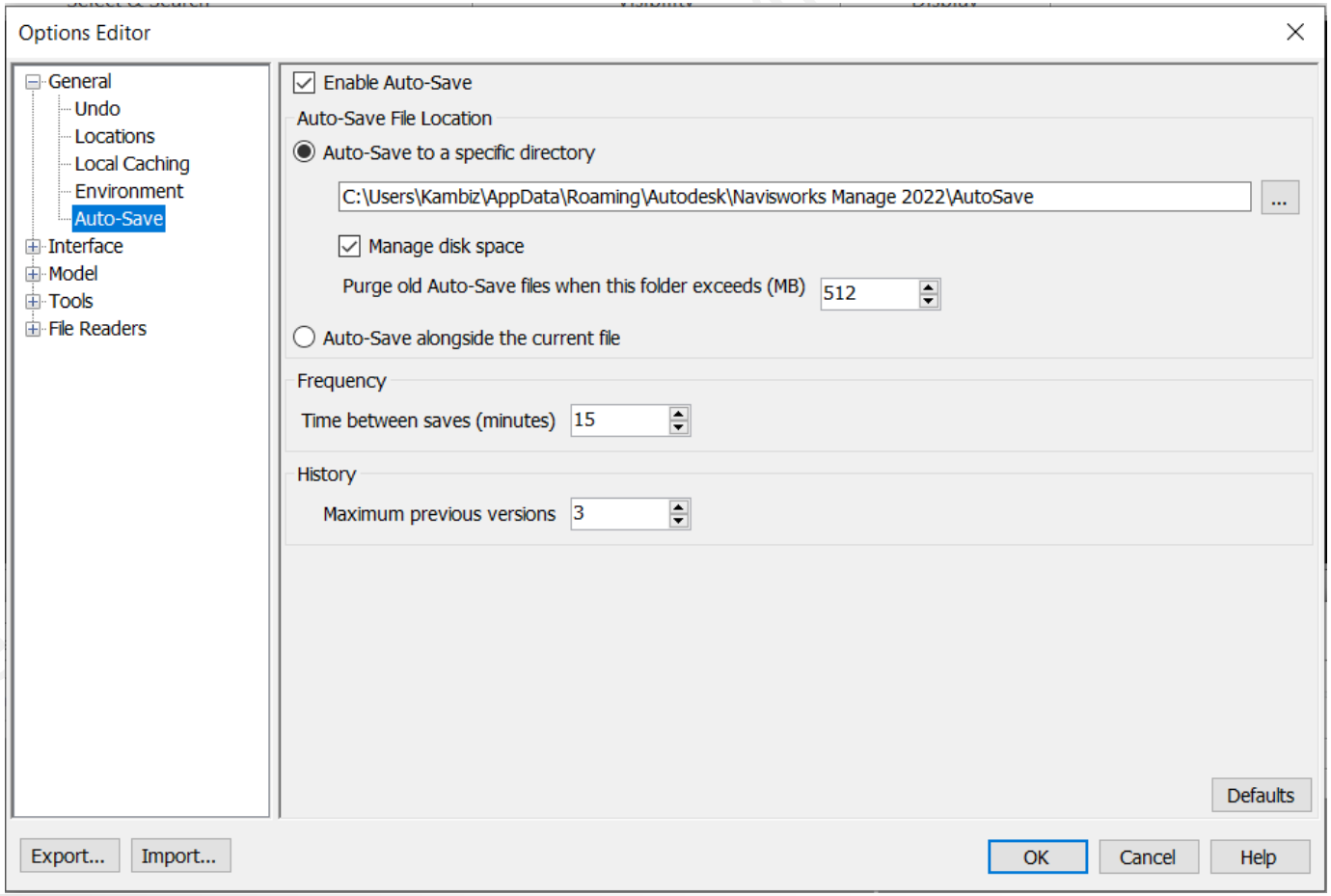


www.dralavipour.com ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.1 Introduction to Autodesk Navisworks

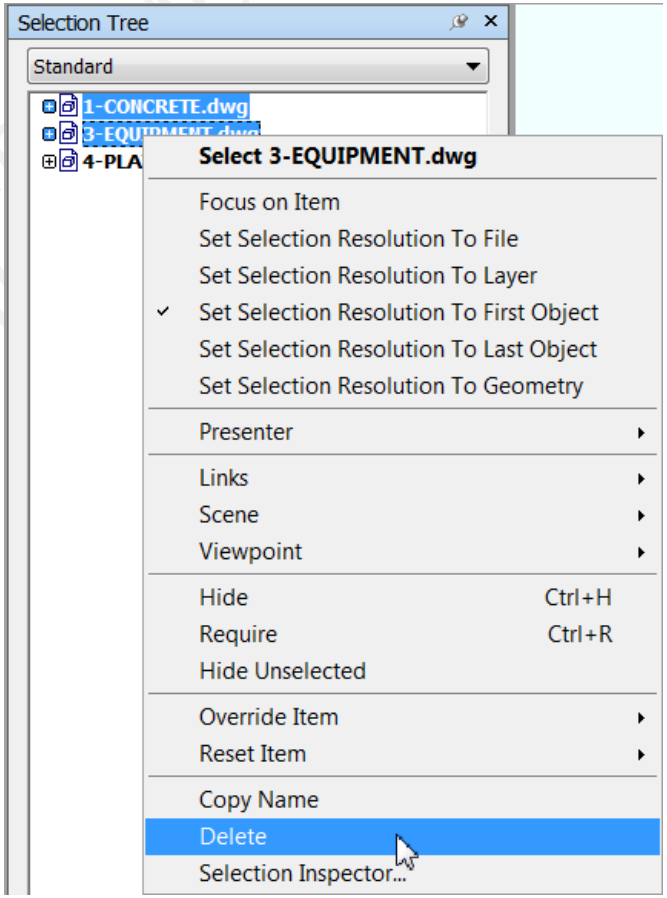
Auto-Save

Shortcut Key F12
Right-click Shortcut Menu > Global Options



2.1 Introduction to Autodesk Navisworks

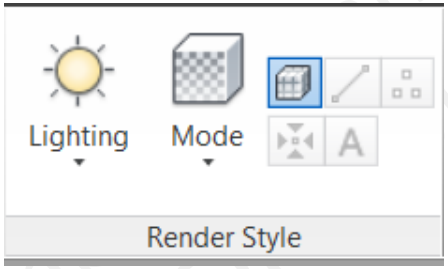
Autodesk Navisworks allows you to delete one or more files from the current scene if the file is not yet saved or it has been saved as an NWF file. Note that the files cannot be deleted from the NWD files. To delete a file, right click on its name in the **Selection Tree** and click **Delete** from the shortcut menu,



Prepared By Kambiz Kordani PMP, MSc
Do Not Distribute

2.1 Introduction to Autodesk Navisworks

CHANGING THE RENDER STYLE OPTIONS: CHANGING THE RENDER STYLE OPTIONS The **Viewpoint** ribbon tab > **Render Style** ribbon panel provides you with the options of changing the lighting and render mode of the scene. There are also the options to specify what type of objects are displayed on the screen,



Lighting Flyout

Viewpoint Ribbon Tab > Render Style Ribbon Panel > Lighting Flyout

Render Mode Flyout

Viewpoint Ribbon Tab > Render Style Ribbon Panel > Render Mode Flyout

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.1 Introduction to Autodesk Navisworks

Surfaces

Viewpoint Ribbon Tab > Render Style Ribbon Panel > Surfaces Button

Lines

Viewpoint Ribbon Tab > Render Style Ribbon Panel > Lines Button

Points

Viewpoint Ribbon Tab > Render Style Ribbon Panel > Points Button

Snap Points

Viewpoint Ribbon Tab > Render Style Ribbon Panel > Snap Points Button

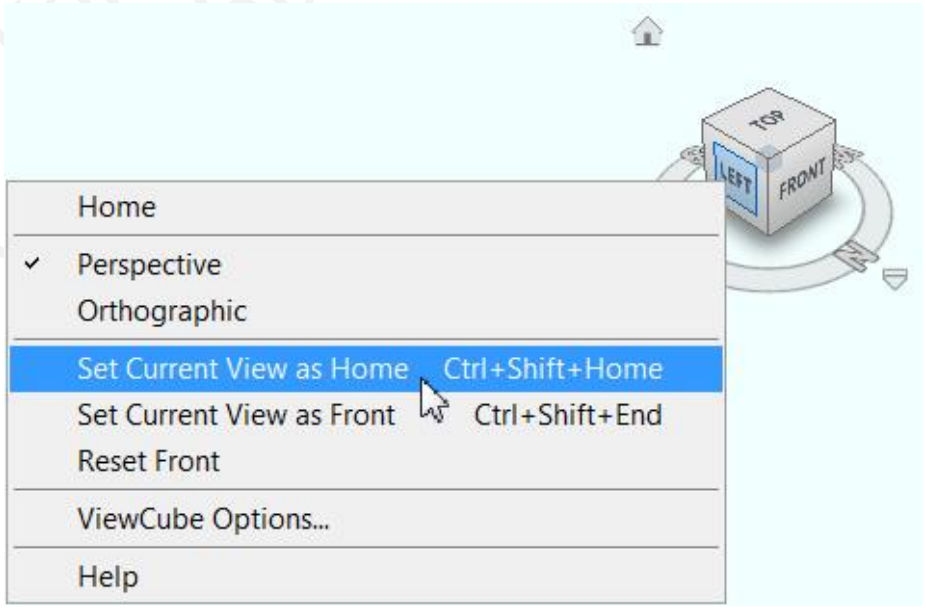
Text

Viewpoint Ribbon Tab > Render Style Ribbon Panel > Snap Points Button

2.1 Introduction to Autodesk Navisworks

VIEWCUBE

The ViewCube is displayed on the top right corner of the graphics window. As mentioned earlier, it is a visualization tool used to display the static views of the model from various directions. By default, it appears faded, which is the inactive state. But when you move the cursor over it, the ViewCube becomes active, highlighting the face, edge, or corner on which you move your cursor.

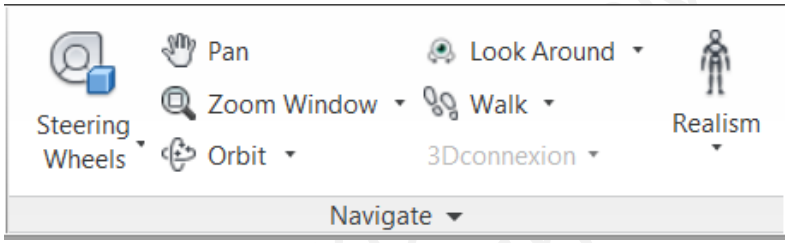


Prepared By Kambiz Kordani
Do Not Distribute

2.1 Introduction to Autodesk Navisworks

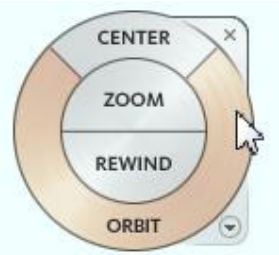
NAVIGATING THROUGH THE MODEL

Autodesk Navisworks provides various navigation tools to help you navigate through the model.



Steering Wheels

Navigation Bar > Steering Wheels
Viewpoint Ribbon Tab > Navigate Ribbon Panel > Full Navigation Wheel



View Object Display

Tour Building Display

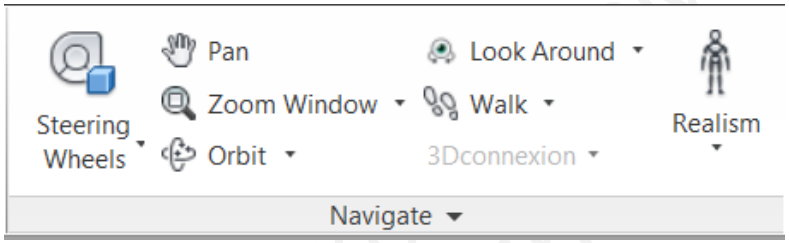
Full Navigation Display

www.dralavipour.com ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.1 Introduction to Autodesk Navisworks

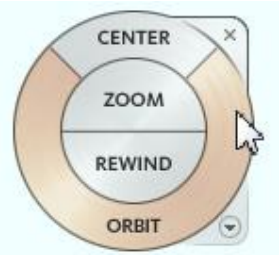
NAVIGATING THROUGH THE MODEL

Autodesk Navisworks provides various navigation tools to help you navigate through the model.



Steering Wheels

Navigation Bar > Steering Wheels
Viewpoint Ribbon Tab > Navigate Ribbon Panel > Full Navigation Wheel



View Object Display

Tour Building Display

Full Navigation Display

www.dralavipour.com ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.1 Introduction to Autodesk Navisworks

Zoom Tools

Navigation Bar > Zoom Flyout
Viewpoint Ribbon Tab > Navigate Ribbon Panel > Zoom Flyout

Zoom Window

This tool allows you to define a window by dragging the mouse and specifying two corners. The part of the scene that is within that window will be displayed to the extents of the graphics window.

Zoom

This tool lets you interactively zoom in and out of the current scene using the current pivot point. When you invoke this tool, the cursor changes to the magnifying glass. Press and hold down the left mouse button and drag the mouse up to zoom in and down to zoom out. If you want to zoom around a different pivot point, press and hold the left mouse button on that vertex of the scene and drag the mouse.

Zoom Selected

This tool lets you zoom the current scene such that the selected object is displayed to the extents of the graphics window. You can select the object before invoking this tool or after invoking this tool.

Zoom All

This tool zooms to the extents of the Autodesk Navisworks model.

2.1 Introduction to Autodesk Navisworks

Orbit Tools

Navigation Bar > Orbit Flyout
Viewpoint Ribbon Tab > Navigate Ribbon Panel > Orbit Flyout

Orbit
This tool allows you to dynamically orbit around the pivot point of the model without allowing the camera to roll.

Free Orbit
This tool allows the unrestricted orbiting around the pivot point of the model. Remember that because orbit is unrestricted, the camera can be rolled in any direction.

Constrained Orbit
This tool allows only a turntable orbiting around the pivot point, maintaining the up direction of the current scene.

Focus Tool

Navigation Bar > Look Around Flyout > Focus
Viewpoint Ribbon Tab > Navigate Ribbon Panel > Look Around Flyout > Focus

The **Focus** tool allows you to select a vertex on the model. The selected vertex is brought to the center of the graphics window and is also used as the pivot point for the orbit tools.

ذکر مطالب فوق تنها ،

CHANGING THE CAMERA OPTIONS

Field of View (F.O.V.) Slider

Viewpoint Ribbon Tab > Camera Ribbon Panel > F.O.V.

Perspective Flyout

Viewpoint Ribbon Tab > Camera Ribbon Panel > Perspective Flyout

Align Camera Flyout

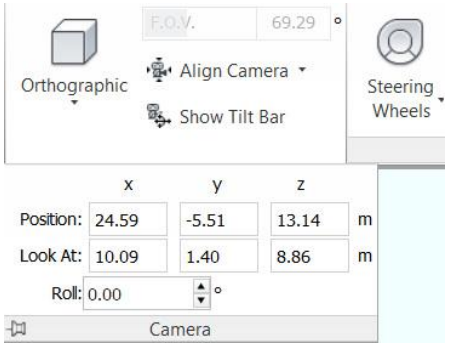
Viewpoint Ribbon Tab > Camera Ribbon Panel > Align Camera

Show Tilt Bar

Viewpoint Ribbon Tab > Camera Ribbon Panel > Show Tilt Bar

Position Entry Boxes

Viewpoint Ribbon Tab > Camera Ribbon Panel > Position



۲. مدلسازی اطلاعات ساخت با استفاده از نرم افزار

Navisworks

Kambiz Kordani PMP,MSc
Linkedin: @kordani-kambiz
Instagram: @acemi_social

2.2 Selecting and Manipulating Objects in the Scene

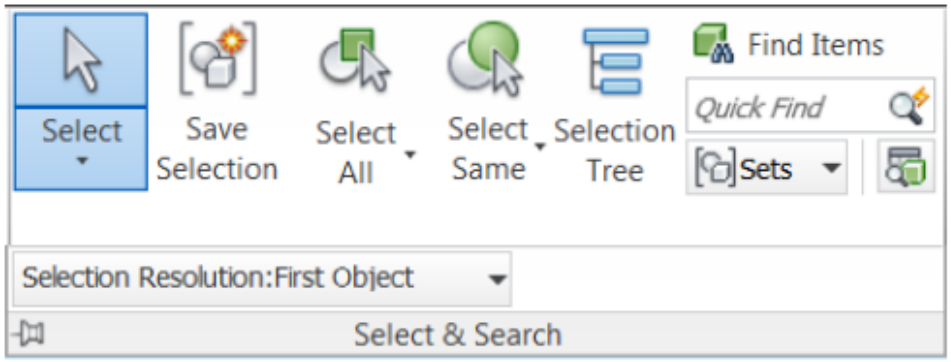
- Familiarize you with various selection resolutions.
- Explain various methods of selecting objects.
- Teach you how to control the visibility of the objects in the scene.
- Explain the use of various item tools.
- Teach you how to override or restore the object appearance.
- Explain how to change the units, location, and orientation of a CAD file.
- Teach you how to create selection sets and search sets.
- Explain how to view native object properties and create custom object properties.
- Teach you how to display quick object properties of the model in the graphics window.

Kambiz Kordani PMP, MSc
Do Not Distribute

www.dralavipour.com ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

OBJECT SELECTION

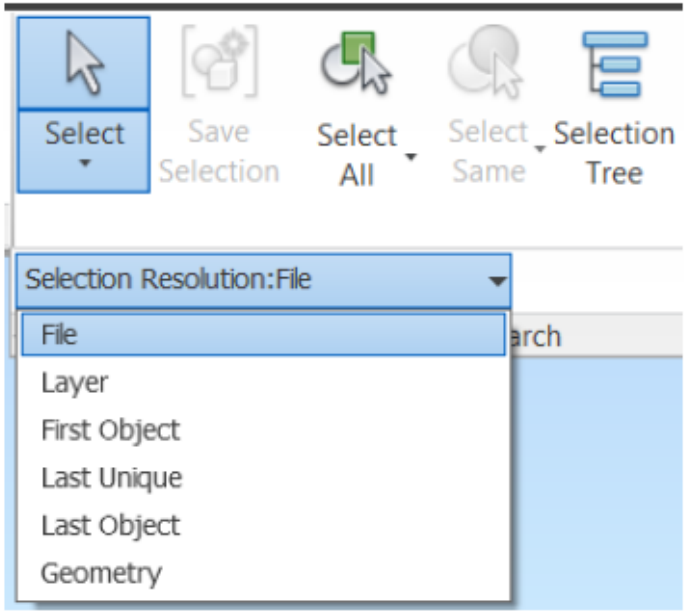
Selecting objects is an important part of any design collaboration process. Autodesk Navisworks provides you various tools to effectively select objects. These tools are available on the **Home** ribbon tab > **Select & Search** ribbon panel.

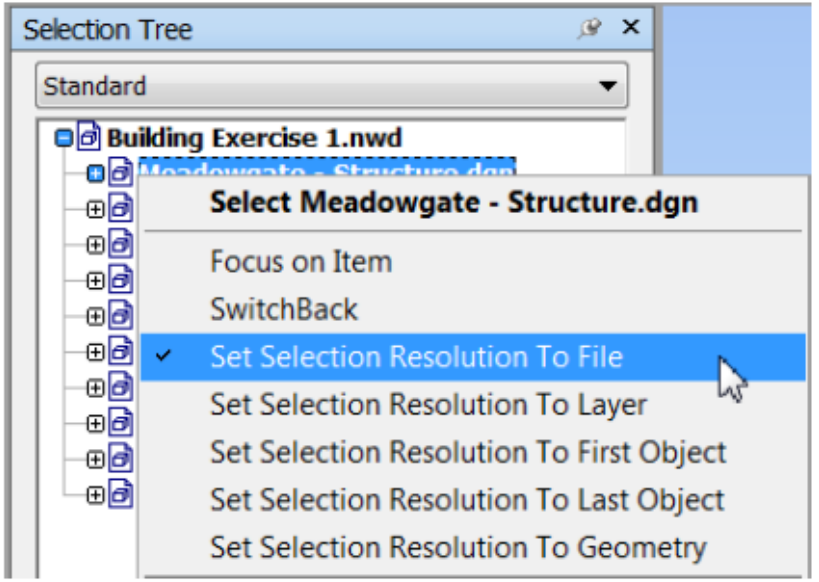


Prepared By
Do No

Selection Resolution

The selection resolution controls what gets selected when you try to make a selection. Autodesk Navisworks provides you various selection resolutions that can be changed from the **Selection Resolution** flyout in the **Select & Search** ribbon panel,



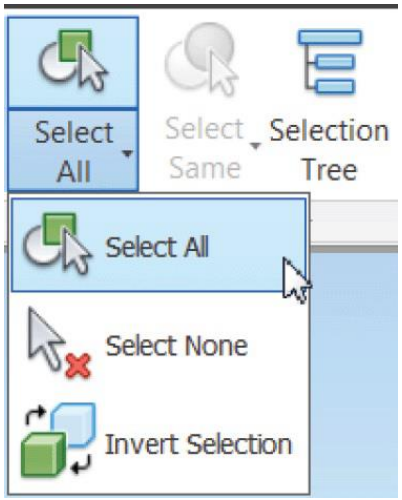


ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

Select Flyout

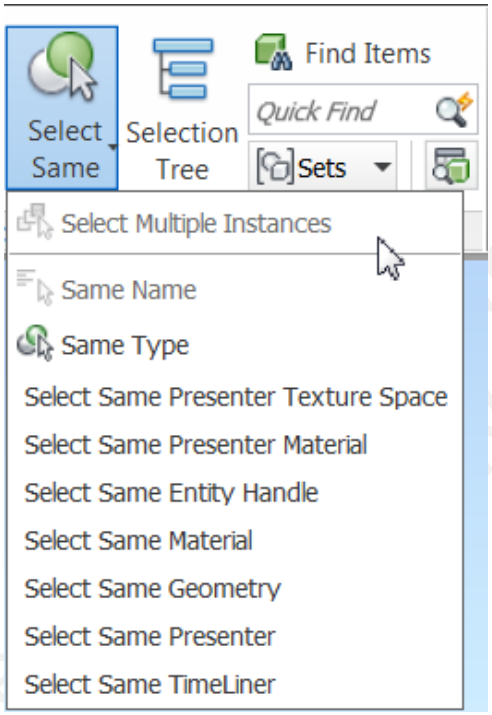
***Note**: Holding down the SHIFT key and selecting any object will select everything in the Autodesk Navisworks scene. Repeating this process will cycle through various selection resolutions.*

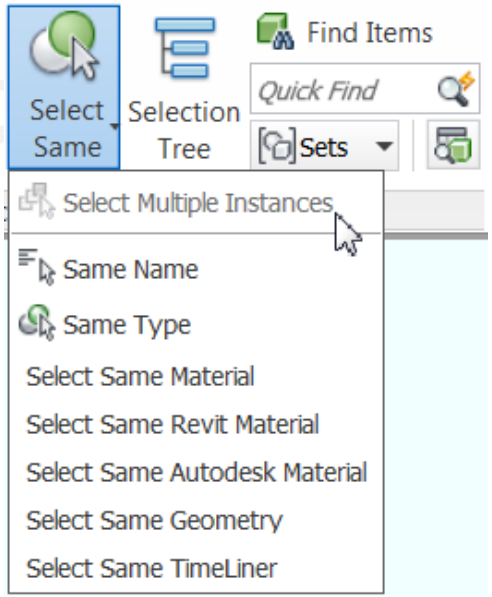
***Tip**: Holding down the CTRL key and clicking on an object will allow you to add or remove it from the current selection.*



Select Same Flyout

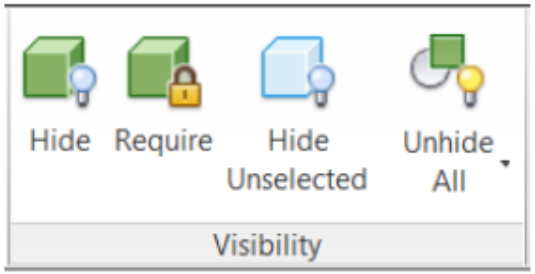
This flyout will only be active if you have made a selection. The options in this flyout will vary depending on the type of file you are working with and the type of object selected.

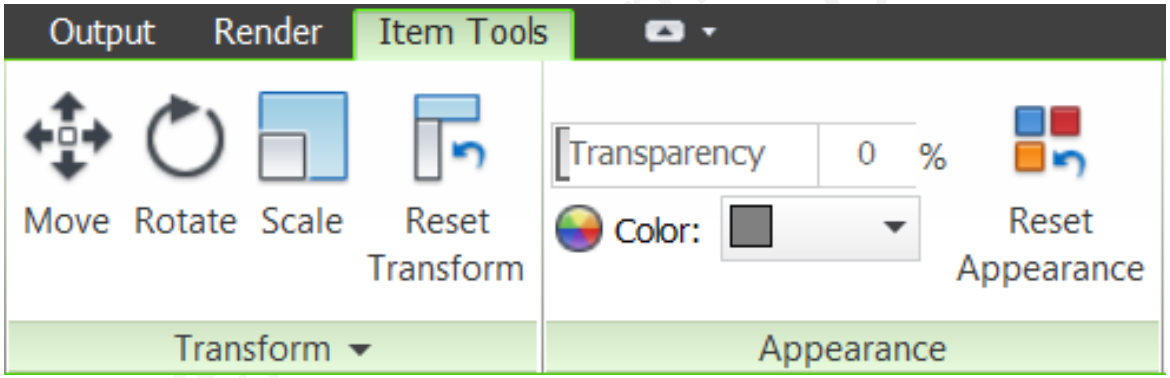


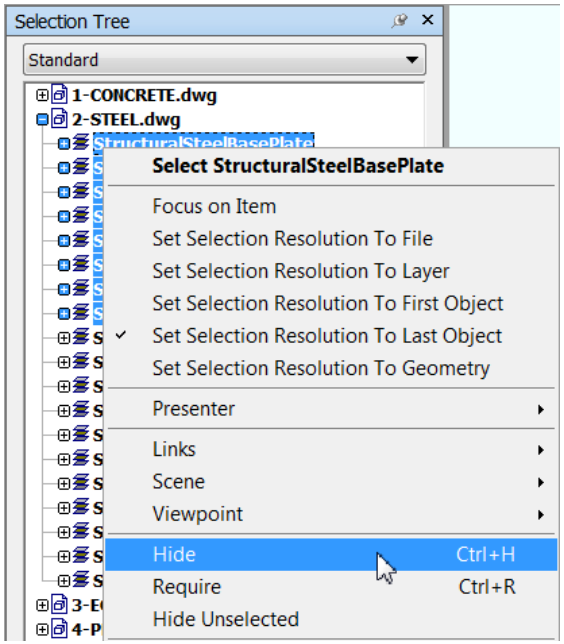


CONTROLLING THE VISIBILITY OF THE OBJECTS

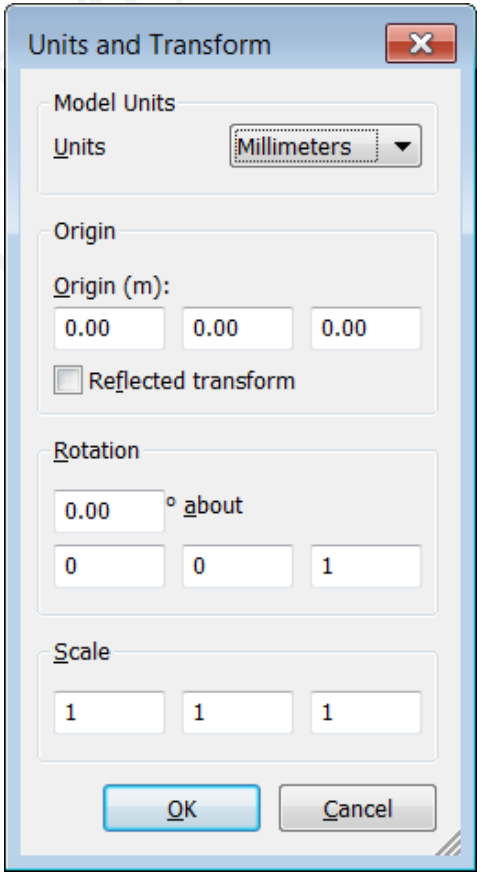
Autodesk Navisworks provides you various tools to turn on or off the visibility of the selected objects. This can be done using the **Visibility** ribbon panel on the **Home** ribbon tab,







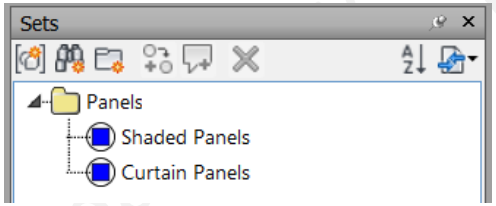
MODIFYING UNITS, SIZE, AND LOCATION OF A CAD FILE

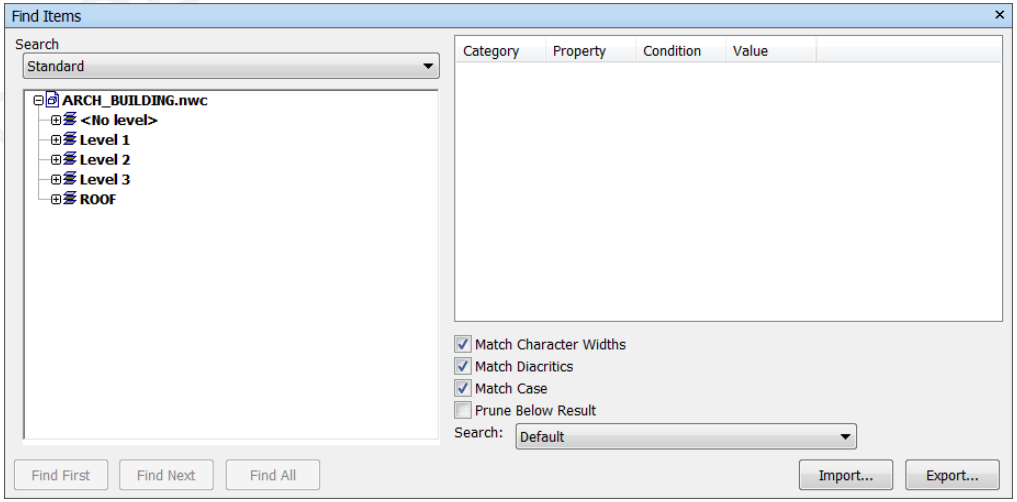


Prepared By Kambiz Kordani
Do Not Distribute

SELECTION SETS AND SEARCH SETS

Selecting objects is a task performed frequently in Autodesk Navisworks. However, selecting multiple objects becomes a tedious job, especially if you have to regularly select a large number of objects. To make this process simpler, Autodesk Navisworks lets you group the objects that you regularly select in a set so that next time when you need to select the same objects, you can simply click on the set. If you have large number of sets in the Autodesk Navisworks scene, these sets can also be organized in folders for ease of use. There are two types of sets available, selection sets and search sets. Both these are discussed next.

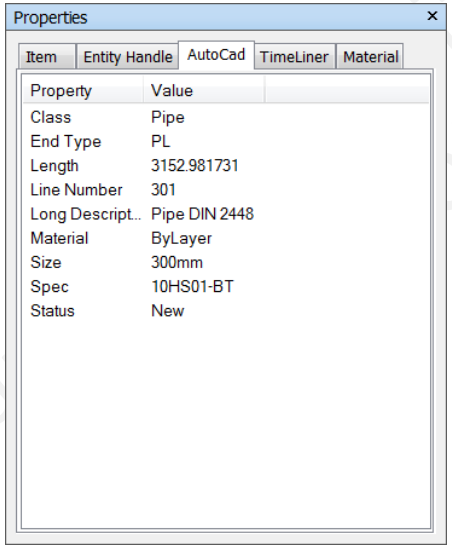


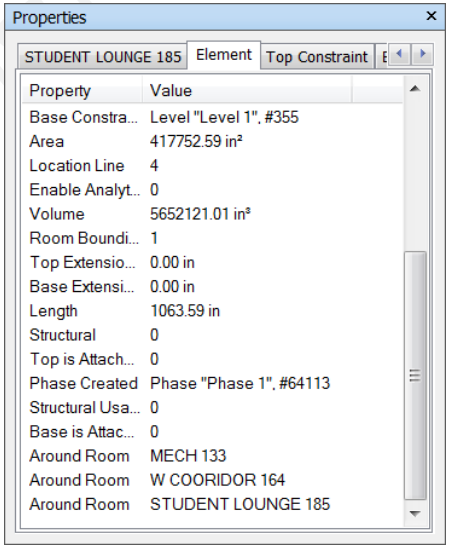


www.dralavipour.com ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

VIEWING OBJECT PROPERTIES

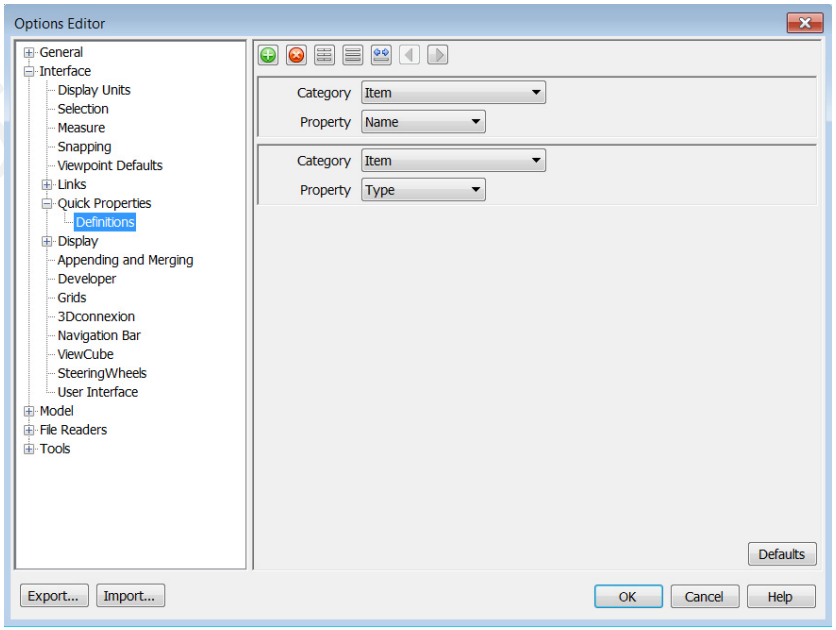
Autodesk Navisworks engine is able to import the native object properties from most of the CAD programs. These properties let you better understand the design intent and requirements. These properties will be displayed in the **Properties** window when you select an object. Depending on the object selected, there will be various tabs available in the **Properties** window representing various categories and the properties related to those categories. Note that the properties displayed are dependent on the selection resolution. As a result, if the selection resolution is set to **File**, only the file properties will be displayed not of any object inside that file.





DYNAMICALLY DISPLAYING OBJECT PROPERTIES ON THE MODEL

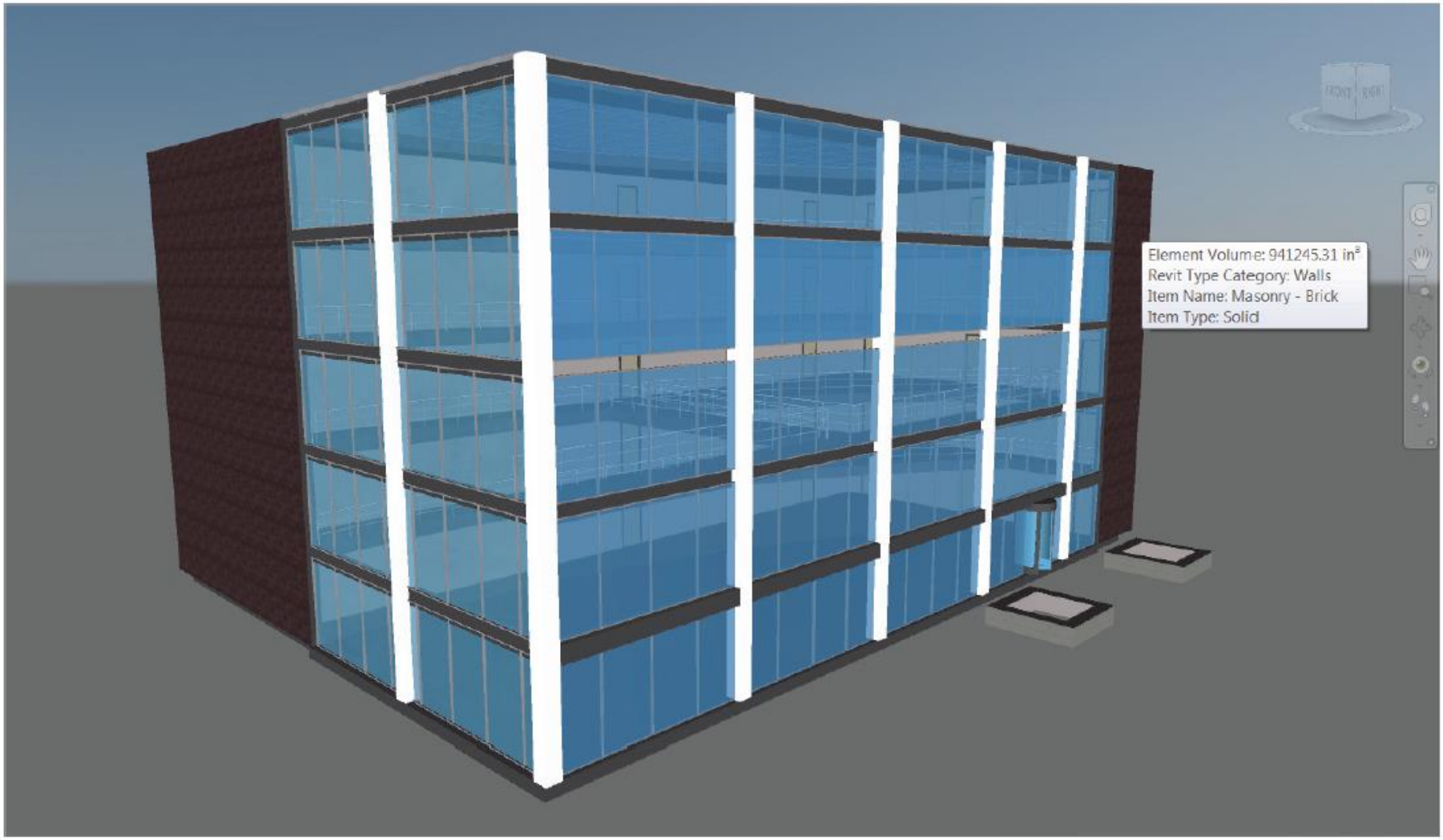
Autodesk Navisworks allows you to dynamically display some of the object properties by simply moving the cursor over the model in the graphics window. These are called quick properties. To display quick properties on the model, turn them on by clicking the **Quick Properties** button on the **Home** ribbon tab > **Display** ribbon panel. When the quick properties are on, move the cursor over any object in the graphics window. By default, the **Item Name** and **Item Type** properties are dynamically displayed.



Prepared By Kambiz
Do Not D

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.2 Selecting and Manipulating Objects in the Scene

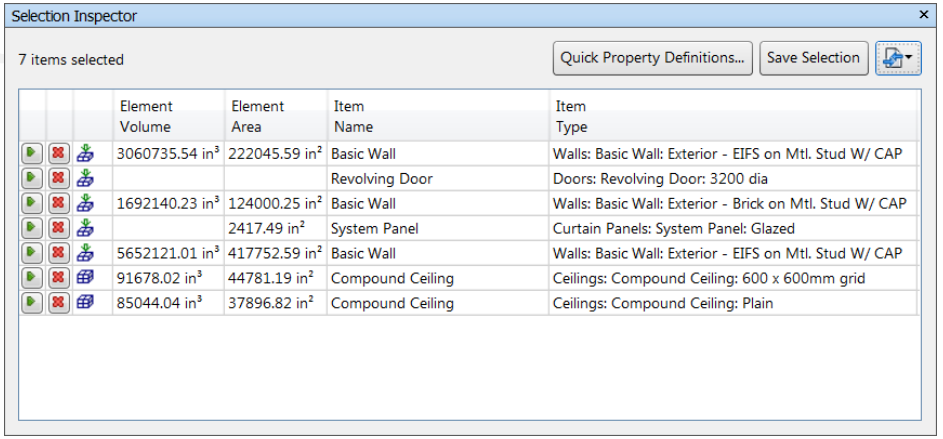


www.dralavipour.com ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

SELECTION INSPECTOR

Home Ribbon Tab > Select & Search Ribbon Panel > Selection Inspector
View Ribbon Tab > Workspace Ribbon Panel > Windows Flyout > Selection Inspector

The **Selection Inspector** window lists the objects currently selected and all their configured quick properties. You can select one or multiple objects from the graphics window or the **Selection Tree**. The number of objects currently selected are displayed near the top left corner of this window. The selected objects can be saved as a selection set from this window. You can also export a CSV file of all the objects listed in this window. Figure 24 shows the **Selection Inspector** window showing various objects selected and their respective quick properties. The options in this window are discussed next.

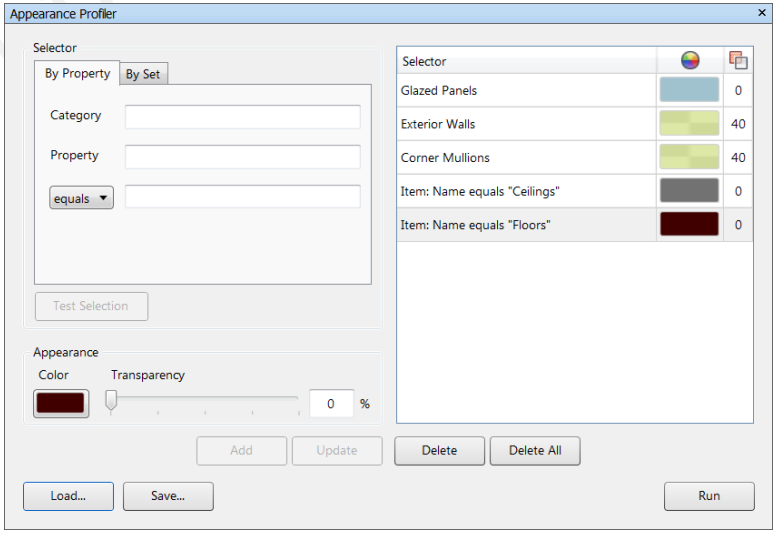


www.dralavipour.com ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

APPEARANCE PROFILER

Home Ribbon Tab > Tools Ribbon Panel > Appearance Profiler
View Ribbon Tab > Workspace Ribbon Panel > Windows Flyout > Appearance Profiler

This is a really smart tool in Autodesk Navisworks that lets you select objects and then assign color and transparency to them for ease of identification. You can select objects based on their properties or use a selection set or a search set already created in the current scene. The common appearances can also be exported and then imported in multiple files.



Prepared By Kambiz
Do Not Di

۲. مدلسازی اطلاعات ساخت با استفاده از نرم افزار

Navisworks

Kambiz Kordani PMP,MSc
Linkedin: @kordani-kambiz
Instagram: @acemi_social

2.3 Viewpoints, Animations, and Measurements

- Show you how to walk and fly in a scene.
- Teach you how to reduce the object dropouts during navigation.
- Familiarize you with the process of creating viewpoints.
- Teach you how to add realism to your navigation.
- Teach you how to create walkthrough animations.
- Teach you how to create animations using viewpoints.
- Show you how to export animations.
- Explain how to measure objects in the scene.
- Show you how to move selected objects by the measured distance value.

Kambiz Kordani PMP, MSc Do Not Distribute

www.dralavipour.com ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

REALTIME NAVIGATION IN THE DESIGN

As mentioned in the previous chapters, Autodesk Navisworks lets you create virtual models in which you can navigate in realtime by walking for flying around your design. This gives you a realistic experience of the model before even a single brick is laid on site. The realtime walkthroughs and flythroughs can also be recorded and played again in future. The realtime navigation tools are discussed next.

Walk Tool

Navigation Bar > Walk
Viewpoint Ribbon Tab > Navigate Ribbon Panel > Walk

Steering Wheel > Walk

Navigation Bar > Steering Wheel
Viewpoint Ribbon Tab > Navigate Ribbon Panel > Steering Wheel

Fly Tool

Navigation Bar > Walk Flyout > Fly
Viewpoint Ribbon Tab > Navigate Ribbon Panel > Walk Flyout > Fly

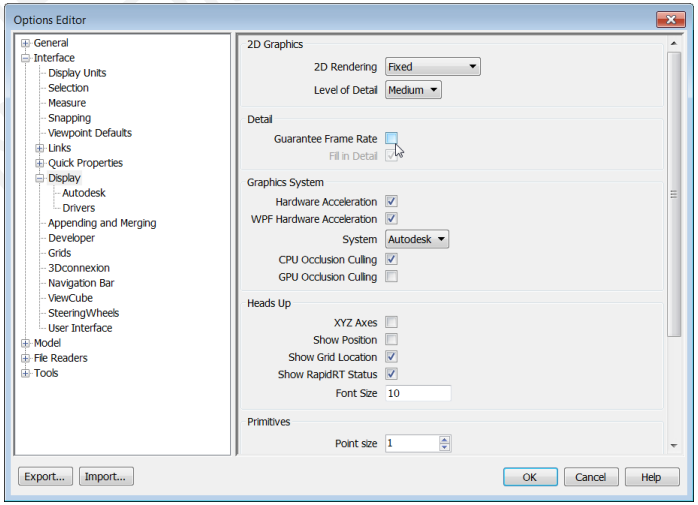
ذکر مطالب فوق تنها در صورت ذکر منبع بلا

Reducing the Object Drop Out Using the File Options

The **Speed** tab of the **File Options** dialog box can be used to reduce the object drop outs during navigation. To display this dialog box, right-click in the blank area of the graphics window and select **File Options** from the shortcut menu to display the **File Options** dialog box. Holding

Stopping Object Drop Outs Using the Options Editor

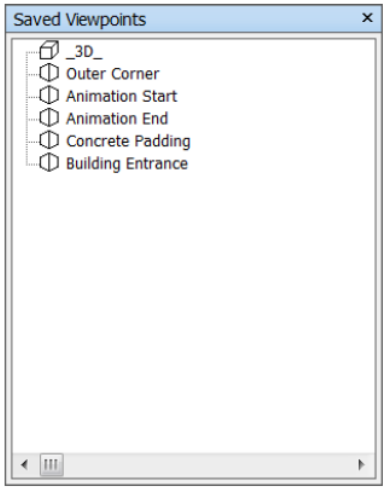
If you want none of the objects to be dropped out during navigation, you can do it using the **Options Editor** dialog box. Similar to the previous option, the drawback in using this option is also that the navigation will be jerky as the frames will be skipped during the navigation. The

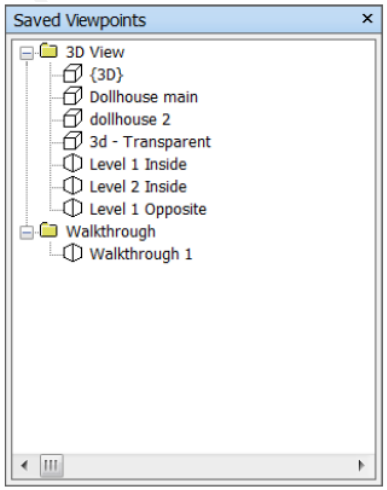


www.dralavipour.com ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

VIEWPOINTS

Autodesk Navisworks lets you save a still shot of the current scene by saving the camera location and camera target. These still shots are called viewpoints. The viewpoints are created and saved in the **Saved Viewpoints** window. In addition to saving the camera related information, viewpoints also allows you to save the visibility and appearance of the objects in the current scene as well as the design review information, such as markups and comments. The viewpoints that are regularly used in your project can also be exported from one file and then imported in other files.





www.dralavipour.com ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

VIEWPOINT SHORTCUT MENU

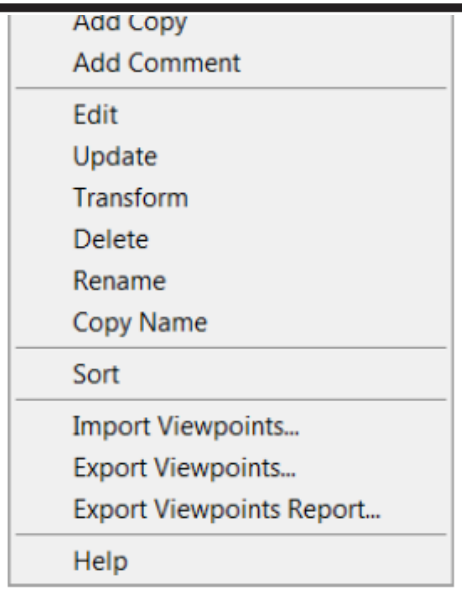
ADDING REALISM TO THE NAVIGATION

ht-click on it,

Autodesk Navisworks lets you add realism to the navigation by adding a third person, turning on the gravity and collision, and also letting the third person crouch under certain objects. All these tools are discussed next.

Third Person

Navigation Bar > Walk Flyout > Third Person
Viewpoint Ribbon Tab > Navigate Ribbon Panel > Third Person
Shortcut Key = CTRL + T



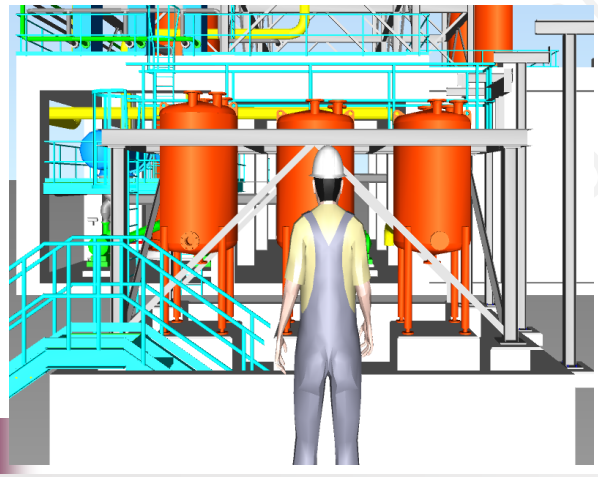
ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

ADDING REALISM TO THE NAVIGATION

Autodesk Navisworks lets you add realism to the navigation by adding a third person, turning on the gravity and collision, and also letting the third person crouch under certain objects. All these tools are discussed next.

Third Person

Navigation Bar > Walk Flyout > Third Person
Viewpoint Ribbon Tab > Navigate Ribbon Panel > Third Person
Shortcut Key = CTRL + T



www.dralavipour.com ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.3 Viewpoints, Animations, and Measurements

Changing Third Person Avatar

	Radius	Height	Distance
Female Avatar	11 Inches 0.3 Meters	68 Inches 1.72 Meters	150 Inches 3.8 Meters
Male Avatar	11 Inches 0.3 Meters	72 Inches 1.9 Meters	170 Inches 4 Meters

Gravity

Navigation Bar > Walk Flyout > Third Person > Gravity
Viewpoint Ribbon Tab > Navigate Ribbon Panel > Third Person > Gravity
Shortcut Key = CTRL + G

www.dralavipour.com ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

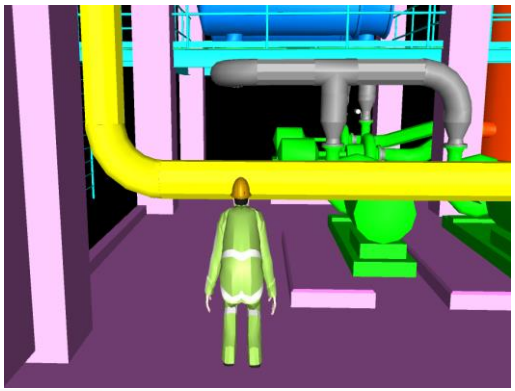
2.3 Viewpoints, Animations, and Measurements

Collision

Navigation Bar > Walk Flyout > Third Person > Collision
Viewpoint Ribbon Tab > Navigate Ribbon Panel > Third Person > Collision
Shortcut Key = CTRL + D

Crouching

Navigation Bar > Walk Flyout > Third Person > Crouching
Viewpoint Ribbon Tab > Navigate Ribbon Panel > Third Person > Crouching



Prepared By
Do Not

GLOBAL VIEWPOINT DEFAULT SETTINGS

Autodesk Navisworks lets you specify the global settings for the viewpoints. As a result, every new file that you open, the viewpoints will follow those settings. The following steps show you how to do this.

CREATING WALKTHROUGH ANIMATIONS

Autodesk Navisworks lets you create walkthrough animations by recording the walk or by using the viewpoints. Both these methods are discussed next.

Recording the Walk

Animation Ribbon Tab > Create Ribbon Panel > Record

Creating Viewpoint Animation

2.3 Viewpoints, Animations, and Measurements

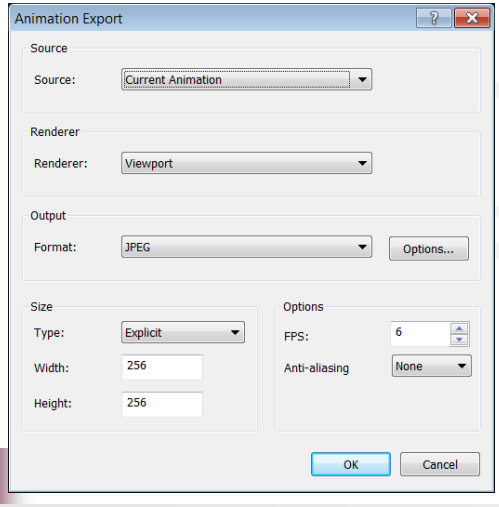
Playing the Animations

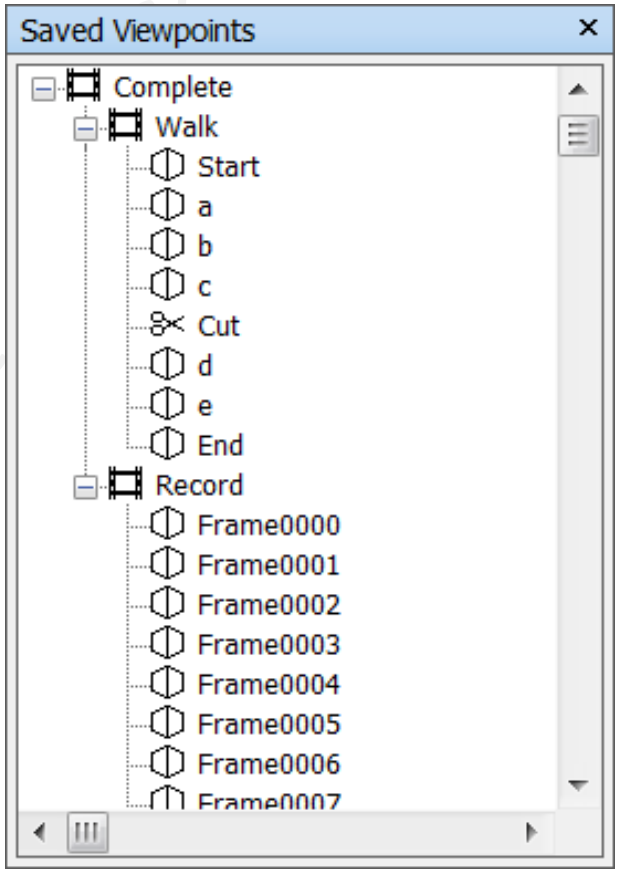
Animation Ribbon Tab > Playback Ribbon Panel > Play

Combining Multiple Animations

Exporting Animations

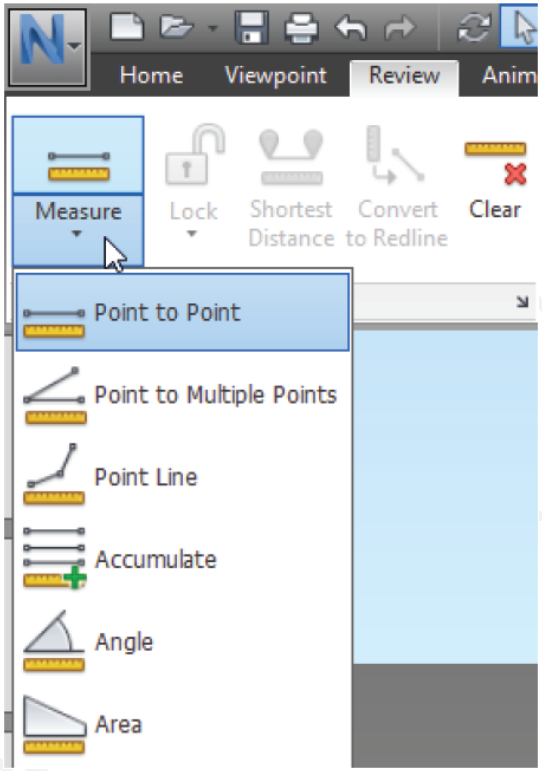
Animation Ribbon Tab > Export Ribbon Panel > Export Animation

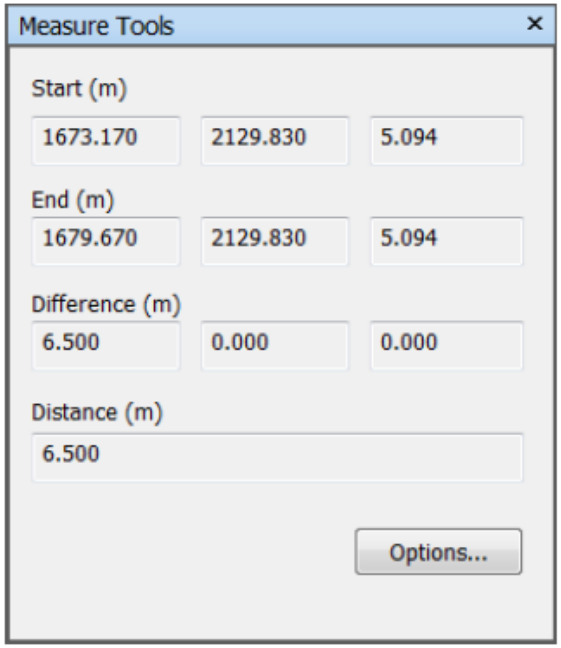




ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

MEASURING OBJECTS





2.3 Viewpoints, Animations, and Measurements

Measuring Point to Point Distance

Review Ribbon Tab > Measure Ribbon Panel > Measure Flyout > Point to Point

Clearing Measurement

Review Ribbon Tab > Measure Ribbon Panel > Clear

Measuring Point to Multiple Points

Review Ribbon Tab > Measure Ribbon Panel > Measure Flyout > Point to Multiple Points

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.3 Viewpoints, Animations, and Measurements

Measuring Point Line

Review Ribbon Tab > Measure Ribbon Panel > Measure Flyout > Point Line

Measuring Accumulated Distances

Review Ribbon Tab > Measure Ribbon Panel > Measure Flyout > Accumulate

Measuring Angles

Review Ribbon Tab > Measure Ribbon Panel > Measure Flyout > Angle

Measuring Area

Review Ribbon Tab > Measure Ribbon Panel >Measure Flyout > Area

Measuring Shortest Distance Between Two Objects

Review Ribbon Tab > Measure Ribbon Panel > Shortest Distance

Transforming Objects

Review Ribbon Tab > Expanded Measure Ribbon Panel > Transform Selected Items

ذکر مطالب فوق تنها در صورت ذ

Converting Measurements to Redline

Review Ribbon Tab > Measure Ribbon Panel > Convert to Redline

Locking Measurements

Review Ribbon Tab > Measure Ribbon Panel > Lock Flyout

Prepared By Kordani
Do Not Distribute

۲. مدلسازی اطلاعات ساخت با استفاده از نرم افزار

Navisworks

Kambiz Kordani PMP,MSc
Linkedin: @kordani-kambiz
Instagram: @acemi_social

2.4 Reviewing and Sectioning the Design

- Familiarize you with the process of reviewing the designs in Autodesk Navisworks.
- Show you how to add redline markups.
- Familiarize you with the process of creating and reviewing comments in the design.
- Familiarize you with the process of creating and reviewing tags in the design.
- Show you how to find specific comments in the design.
- Teach you how to section the models using one or more section planes.
- Show you how to link section planes.
- Teach you how to section the models using a section box.

Kambiz Kordani PMP, MSc
Do Not Distribute

www.dralavipour.com ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.4 Reviewing and Sectioning the Design

Converting Measurements to Redline

Review Ribbon Tab > Measure Ribbon Panel > Convert to Redline

Locking Measurements

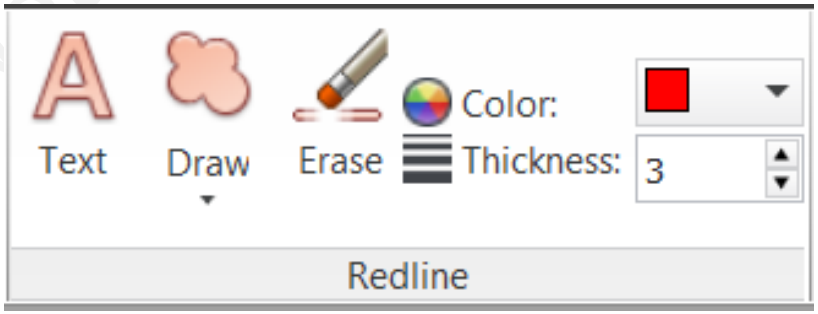
Review Ribbon Tab > Measure Ribbon Panel > Lock Flyout

Prepared By Kordani
Do Not Distribute

DESIGN REVIEW IN AUTODESK NAVISWORKS

Autodesk Navisworks provides you various tools to review your design. These include redline markups, comments, and tags. To add redline markups or comments can only be added in a saved viewpoint. In case of tags, if the viewpoint is not already saved, it is automatically created and saved as a part of the process. The design review components added to the model and published as a .NWD file can also be viewed in the Autodesk Navisworks Freedom viewer. The following sections explain all these design review tools in detail.

Redline Markups



Prepared By Kambiz
Do Not Distribute

2.4 Reviewing and Sectioning the Design

Text

Review Ribbon Tab > Redline Ribbon Panel > Text

Moving a Redline Text

Editing a Redline Text

Draw Flyout

Review Ribbon Tab > Redline Ribbon Panel > Draw Flyout

Erase

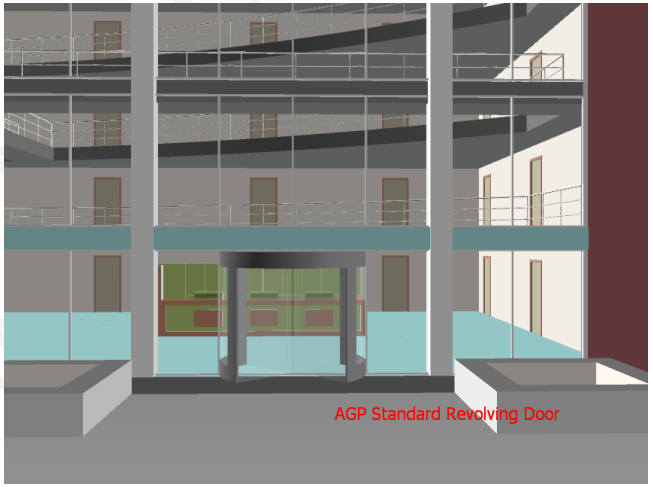
Review Ribbon Tab > Redline Ribbon Panel > Erase

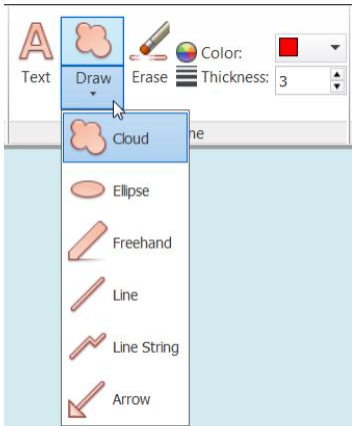
Color

Review Ribbon Tab > Redline Ribbon Panel > Color

Thickness

Review Ribbon Tab > Redline Ribbon Panel > Thickness





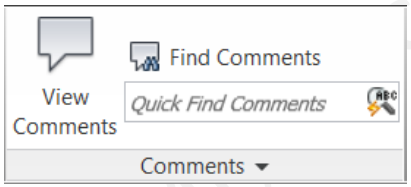
ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

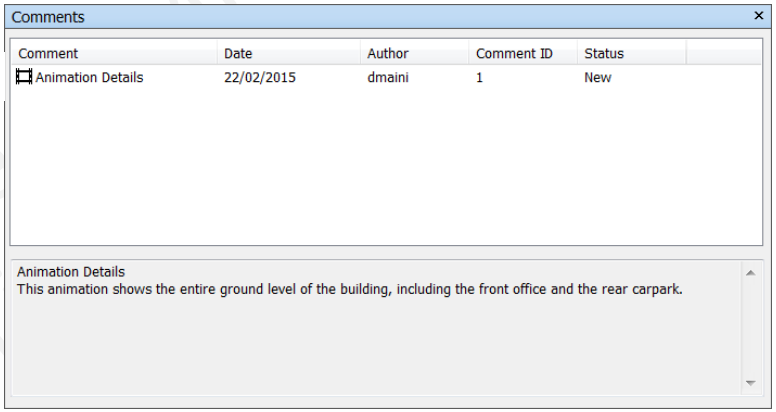
2.4 Reviewing and Sectioning the Design

Adding Comments to a Saved Viewpoint

Adding Comments to a Saved Viewpoint Animation

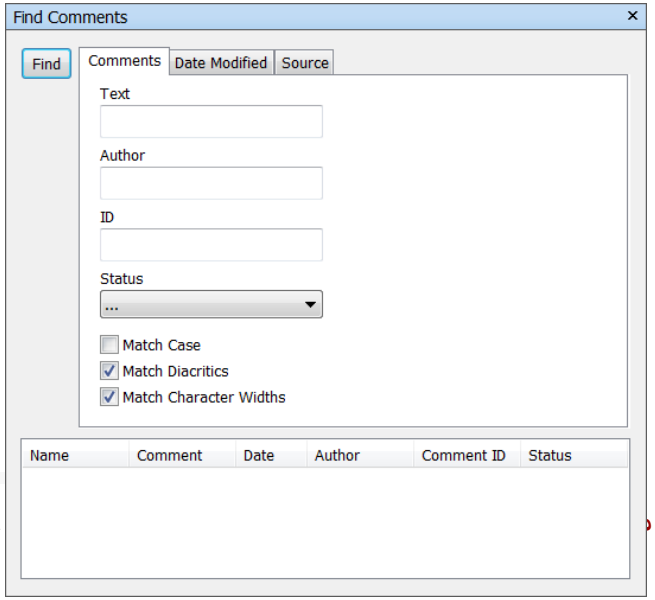
Adding Comments to a Selection Set or a Search Set





Finding Comments

Review Ribbon Tab > Comments Ribbon Panel > Find Comments



2.4 Reviewing and Sectioning the Design

Quick Find Comments

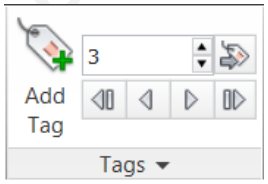
Review Ribbon Tab > Comments Ribbon Panel > Quick Find Comments

Renumbering Comment IDs

Review Ribbon Tab > Expanded Comments Ribbon Panel > Renumber Comment IDs

Tags

Adding tags to the design is a sophisticated way of writing comments that are directly related to a viewpoint. While adding tags, you are allowed to specify the location of the tag identification number in the Autodesk Navisworks scene. A viewpoint will automatically be created, if it not already available, and you will be allowed to write comments associated with that tag in the **Add Comment** dialog box. As a result, these tag comments will also be listed in the **Comments** window and you will be able to search for them at any stage.

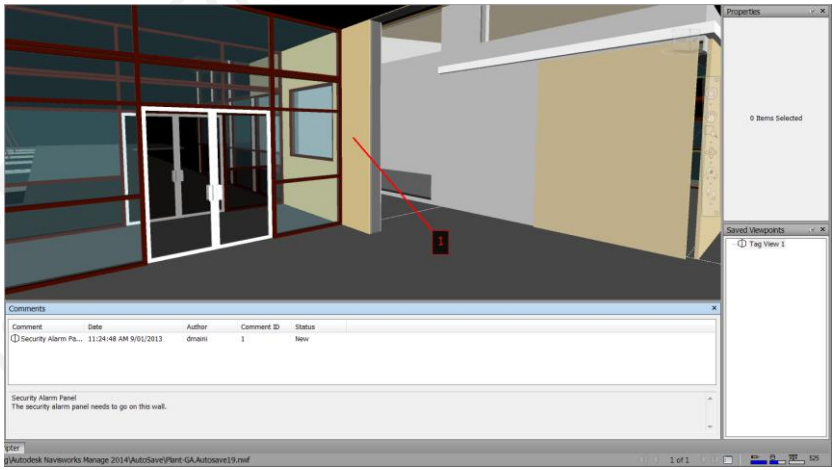


ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.4 Reviewing and Sectioning the Design

Add Tags

Review Ribbon Tab > Tags Ribbon Panel > Add Tags



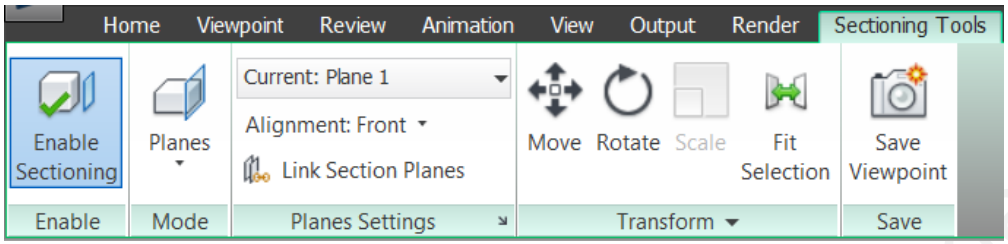
Searching Tag IDs

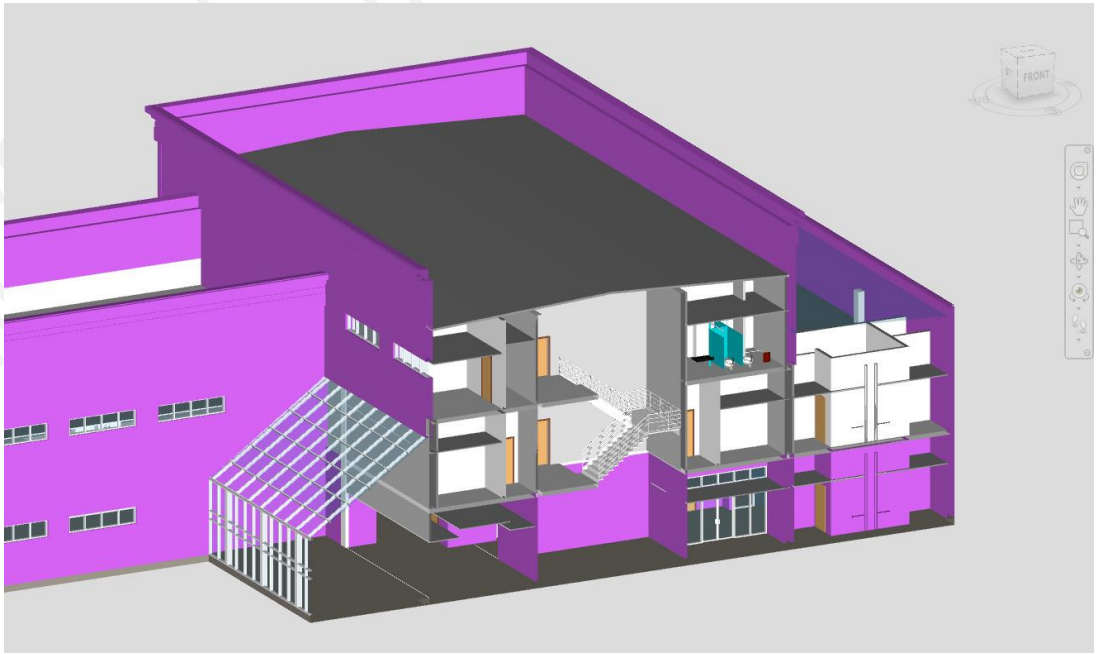
Review Ribbon Tab > Tags Ribbon Panel > Tag ID

Renumbering Tag IDs

Review Ribbon Tab > Expanded Tags Ribbon Panel > Renumber Tag IDs

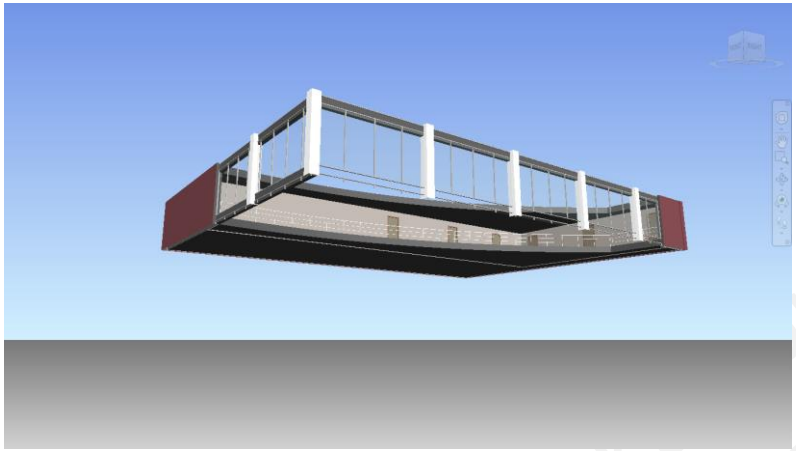
SECTIONING THE MODEL

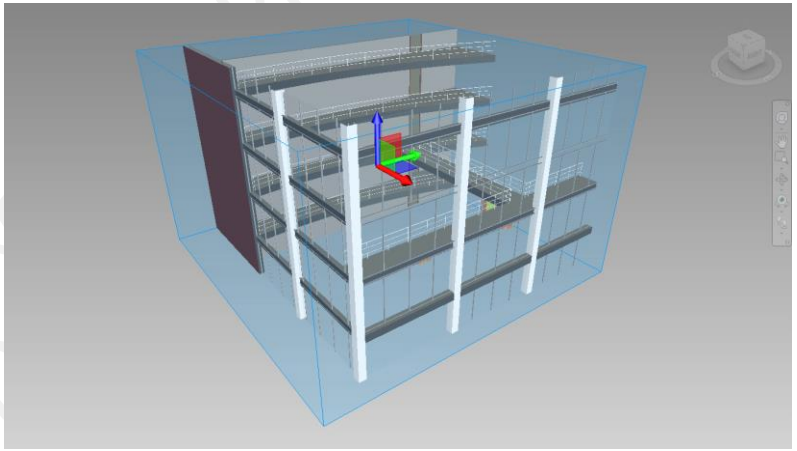




www.dralavipour.com ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

SECTIONING THE MODEL



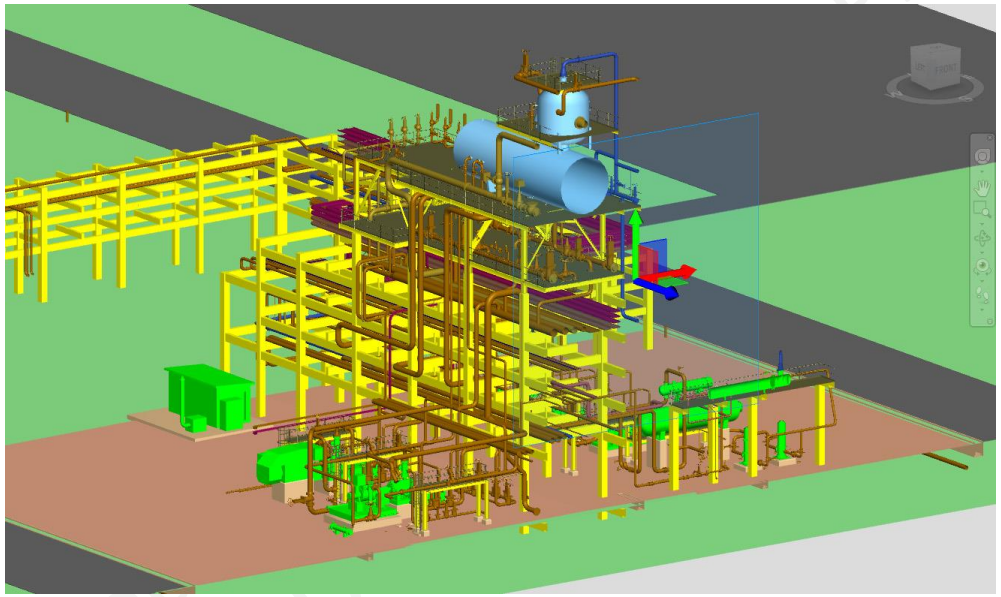


Prepared By K.Kordani PMP, MSc
Do Not D

2.4 Reviewing and Sectioning the Design

Saving a Sectioned View

Sectioning Tools Ribbon Tab > Save Ribbon Panel > Save Viewpoint



ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

۲. مدلسازی اطلاعات ساخت با استفاده از نرم افزار

Navisworks

Kambiz Kordani PMP,MSc
Linkedin: @kordani-kambiz
Instagram: @acemi_social

2.5 Autodesk Navisworks Productivity Tools

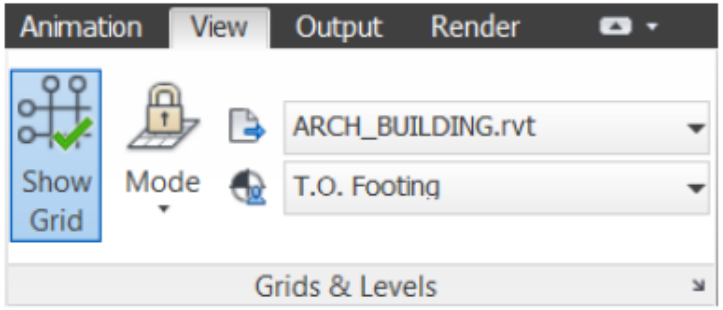
- Introduce you to grids and levels in the Autodesk Revit files.
- Show you how to add links to the elements in the model.
- Show you how to compare two versions of the models to find differences.
- Teach you how to split the Autodesk Navisworks window.
- Teach you how to work with full screen mode.

Kambiz Kordani PMP, MSc
Do Not Distribute

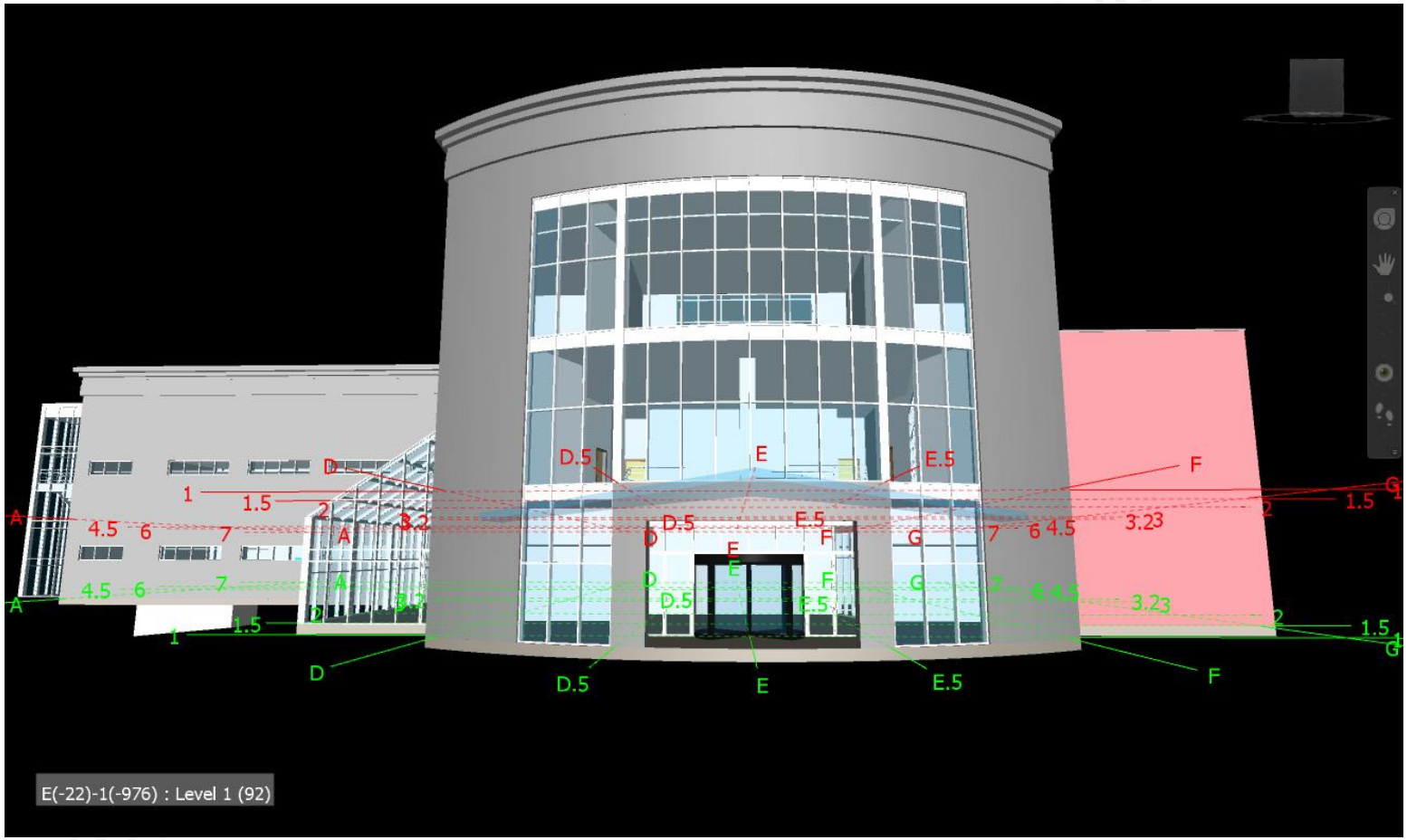
www.dralavipour.com ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

DISPLAYING GRIDS AND LEVELS IN AN AUTODESK REVIT FILE

When you open or append a native Autodesk Revit (*.RVT) file, Autodesk Navisworks allows you to turn on or off the grids and levels created in the native file. This is done using the **Grids & Levels** ribbon panel available on the **View** ribbon tab, as shown in Picture . Note that the options in this ribbon panel are grayed out if you open a non-Autodesk Revit file.

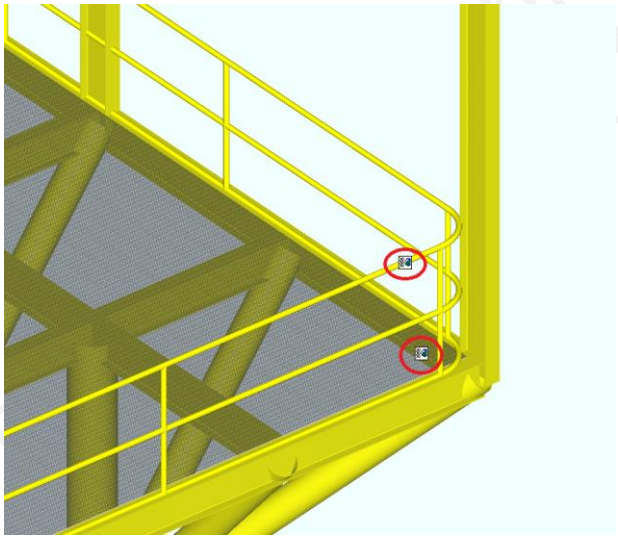


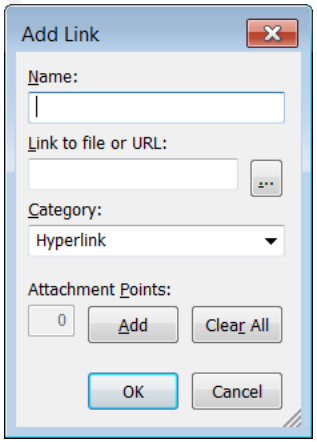
2.5 Autodesk Navisworks Productivity Tools



LINKING FILES AND URLS TO THE OBJECTS IN THE SCENE

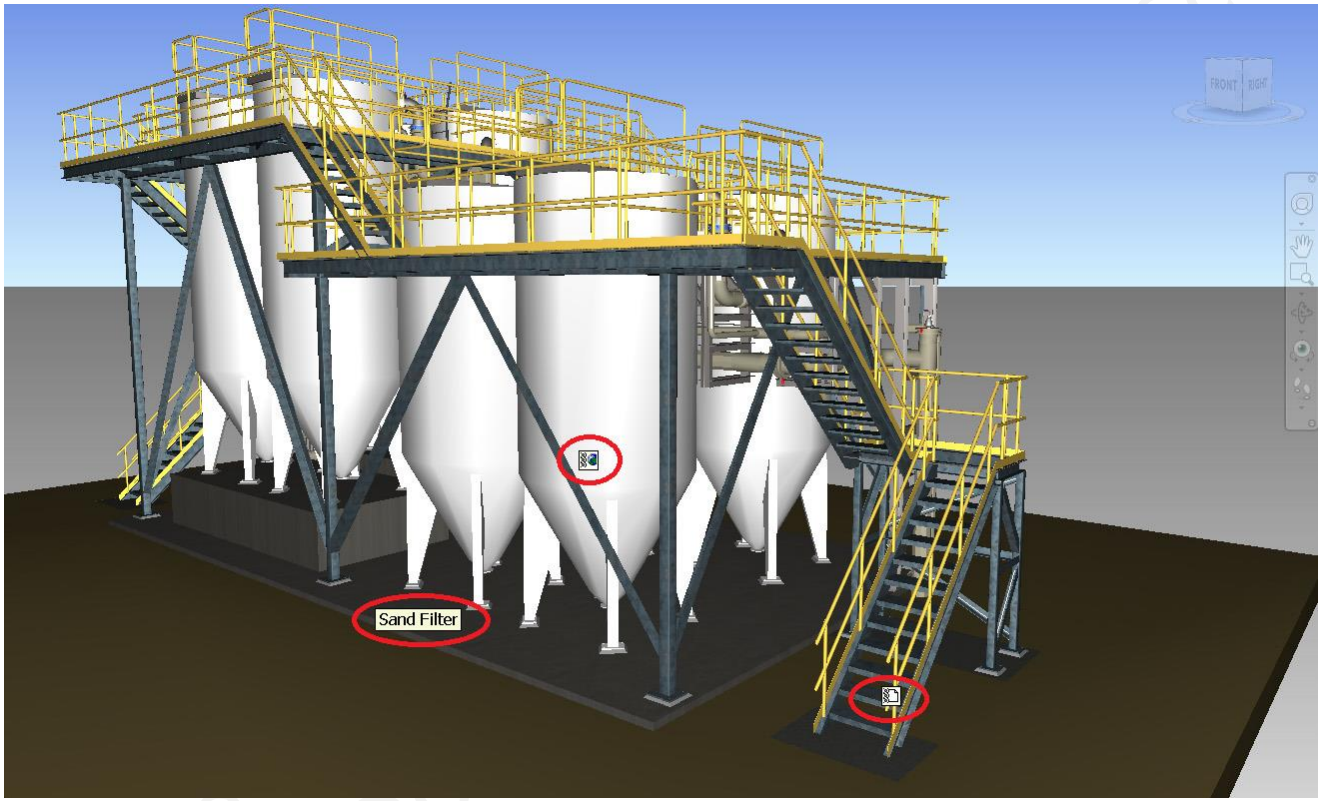
Autodesk Navisworks allows you to link external files or URLs to the objects in the scene. These links are really useful to provide information such as specification sheets, service schedules, Web site addresses, and so on with the model. Consider a case where a pump in the desalination plant develops a fault. To find the details of the pump, the project manager can navigate to that pump in the Autodesk Navisworks model and click on the link associated with the pump to get all its specifications and also the supplier's Web site to order it.





www.dralavipour.com ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.5 Autodesk Navisworks Productivity Tools



***Tip**: If you select an object that has been assigned links, the **Properties** window will show an additional tab called **Hyperlink** that shows the name, URL, and category of the link assigned to the object.*

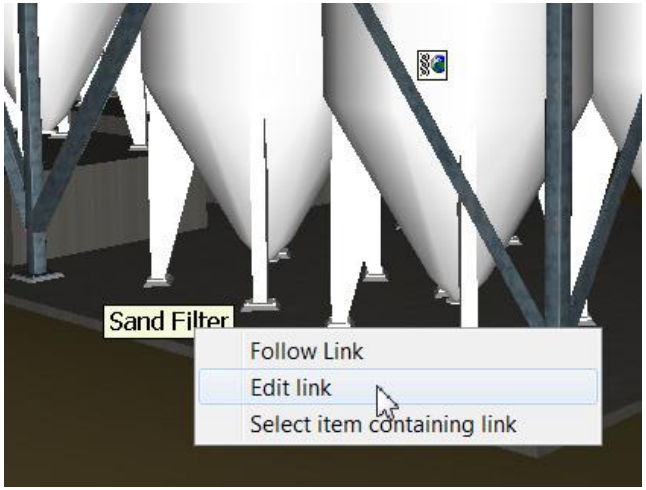
Turning On the Visibility of the Links in the Scene

Home Ribbon Tab > Display Ribbon Panel > Links

Controlling the Global Display of Links

When you turn on the visibility of links in the scene, by default, Autodesk Navisworks not only displays the text and icons of links, but also displays the viewpoint icons, redline icons, timeliner icons, clash detective icons, and so on. With all these icons turned on, the screen looks cluttered. To reduce this cluttering, you can use the **Options Editor** dialog box and turn off the visibility of the unwanted icons. The following steps show you how to do this.

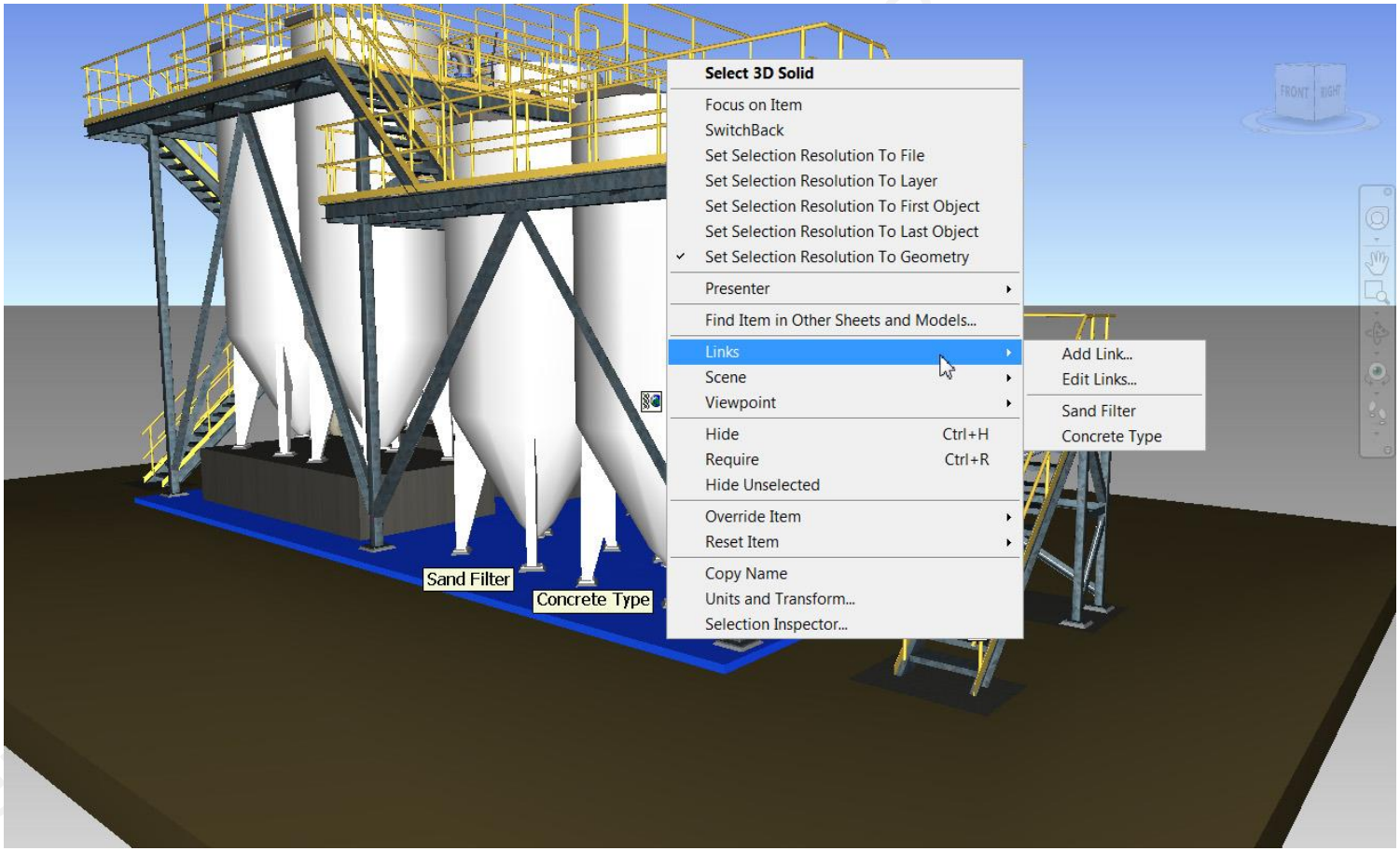
Editing the Links Assigned to the Objects



ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.5 Autodesk Navisworks Productivity Tools

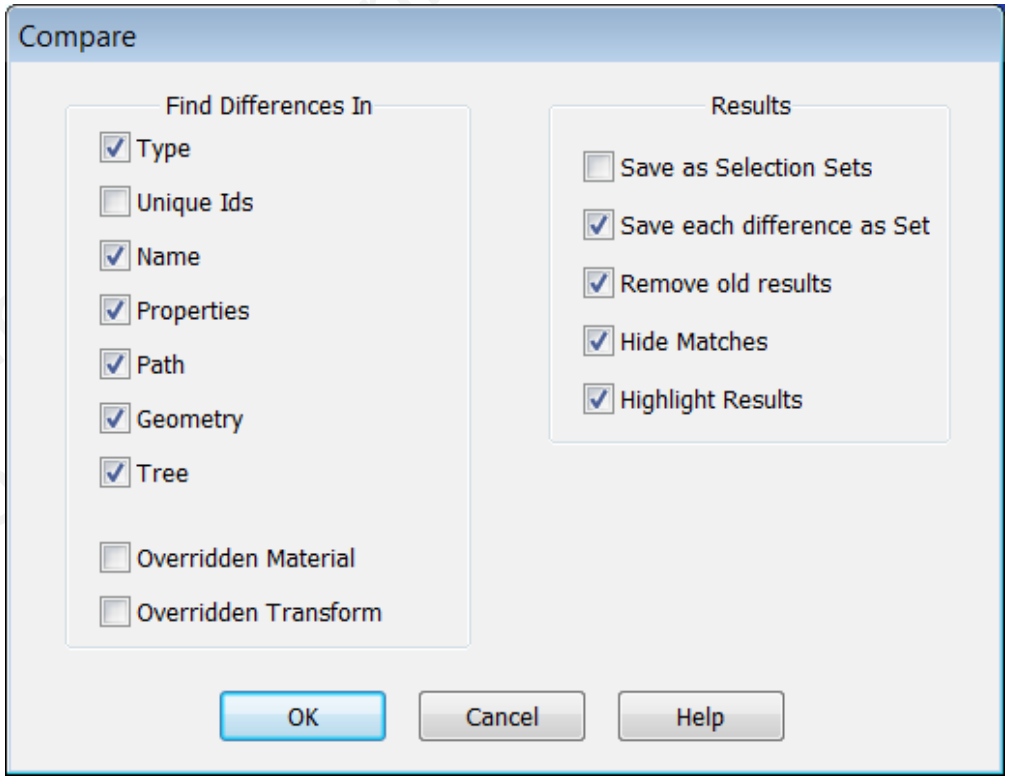
Following the Links Assigned to the Objects



ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

COMPARING MODELS OR OBJECTS IN THE SCENE

Home Ribbon Tab > Tools Ribbon Panel > Compare

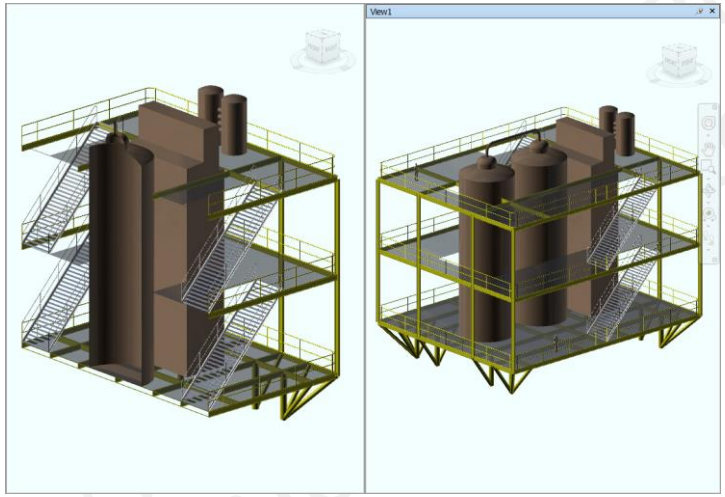


www.dralavipour.com ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.5 Autodesk Navisworks Productivity Tools

SPLITTING THE AUTODESK NAVISWORKS VIEW

View Ribbon Tab > Scene View Ribbon Panel > Split View Flyout



WORKING WITH FULL SCREEN MODE

View Ribbon Tab > Scene View Ribbon Panel > Full Screen
Shortcut Key: F11

۲. مدلسازی اطلاعات ساخت با استفاده از نرم افزار

Navisworks

Kambiz Kordani PMP,MSc
Linkedin: @kordani-kambiz
Instagram: @acemi_social

2.6 Working with the Autodesk Rendering Module

- Introduce you to the Autodesk Rendering module of Autodesk Navisworks.
- Explain how to apply materials to the objects in the scene.
- Explain how to apply lighting to the scene.
- Teach you how to create a photorealistic rendering of the scene.
- Teach you how to export a photorealistic image of the scene.
- Teach you how to render your designs using the Autodesk 360 cloud services.
- Show you how to preview the renderings using the Autodesk 360 render gallery.

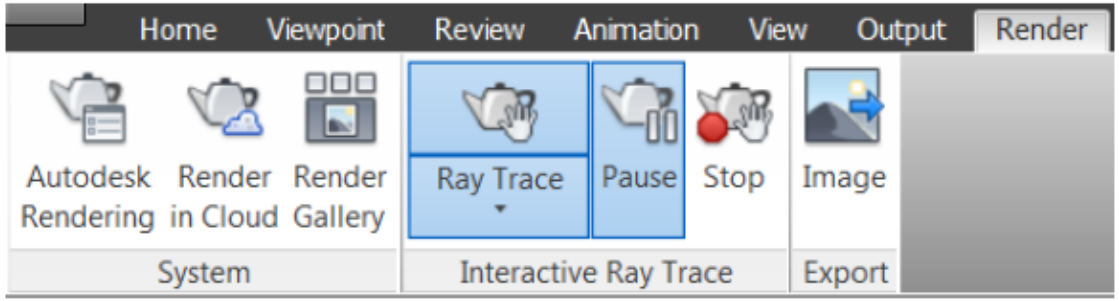
Kambiz Kordani PMP, MSc
Do Not Distribute

www.dralavipour.com ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

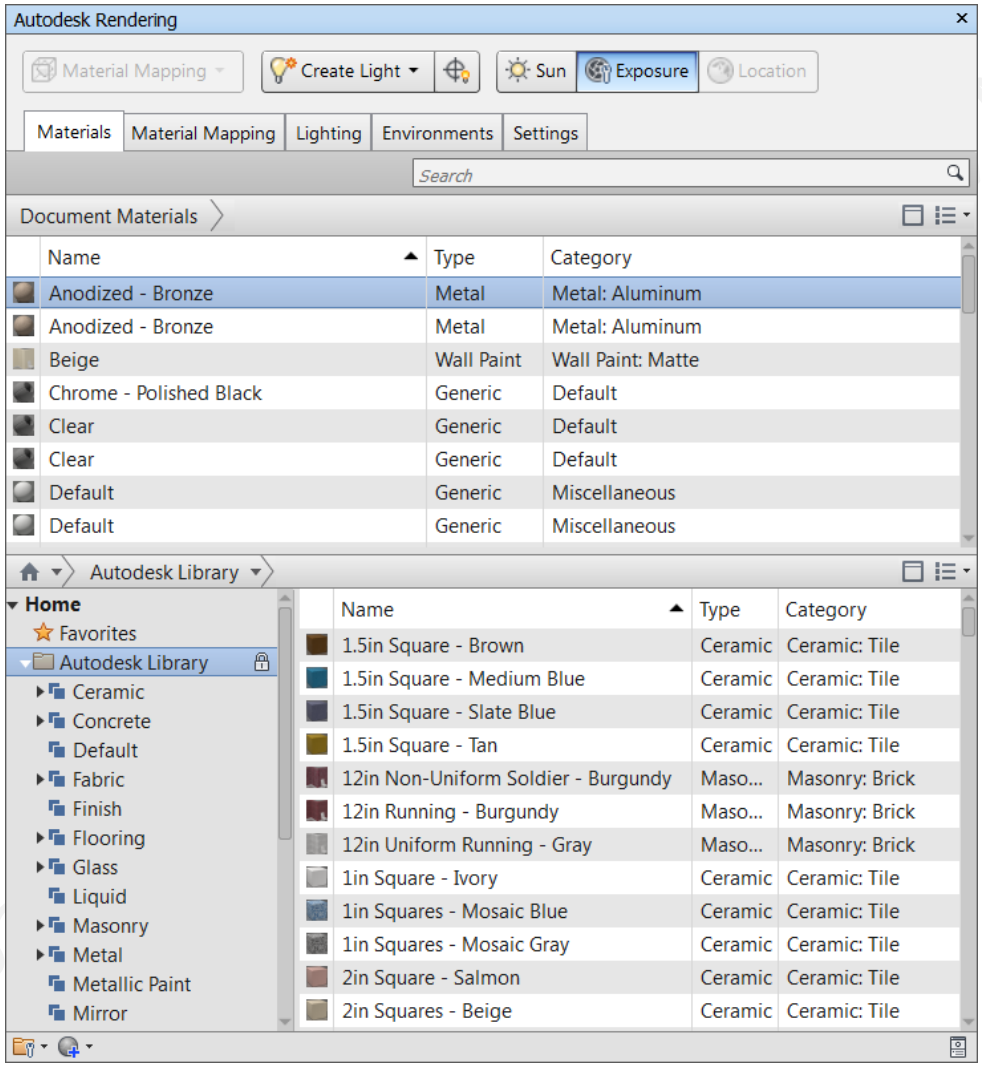
CREATING PHOTOREALISTIC RENDERINGS

Render Ribbon Tab > System Ribbon Panel > Autodesk Rendering

Autodesk Navisworks allows you to create a high quality rendered image or animation of the scene using the **Autodesk Rendering** module. These rendered images or animations can be used in the upstream or downstream departments for creating marketing material, presentations, technical documents, and so on.



2.6 Working with the Autodesk Rendering Module



2.6 Working with the Autodesk Rendering Module

Procedure for Assigning Materials Using the Autodesk Rendering Window

The following steps show how to use the **Materials** tab of the **Autodesk Rendering** window to assign material to the objects in the scene.

a. Activate the **Materials** tab of the **Autodesk Rendering** window, if not already active.

b. Select the objects or the set of objects to which you need to assign the material.

c. From the **Document Materials** area in the upper half of the **Materials** tab or from the **Favorites** or **Autodesk Library** in the lower half, select the material you want to assign. Alternatively, you can use the **Search** box in the upper half of the dialog box to search for a material.

d. Right-click on the selected material and from the shortcut menu, select **Assign To Selection**; the material will be assigned to the selected objects and will also be added to the **Document Materials** area. Alternatively, you can drag and drop the material on the object to which you want to assign the material.

***Note**: To see the realistic effects of the materials assigned, you need to render the scene using the **Render** ribbon tab > **Interactive Ray Trace** ribbon panel > **Ray Trace** tool.*

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.6 Working with the Autodesk Rendering Module

Procedure for Adding Materials to the Favorites Library

As mentioned earlier, the materials that you commonly use can be added to the **Favorites** library. These materials are then available for use in all the files that you open henceforth. The following steps show how to add materials to the **Favorites** library.

a. Select the material from the **Document Materials** area or the **Autodesk Library** and then right-click on it to display the shortcut menu.

b. From the shortcut menu, click **Add to > Favorites**.

c. You can also right-click on the **Favorites** library and create categories in it. This way, the materials can directly be added to that category of the **Favorites** library.

Prepared By K... Do Not...

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

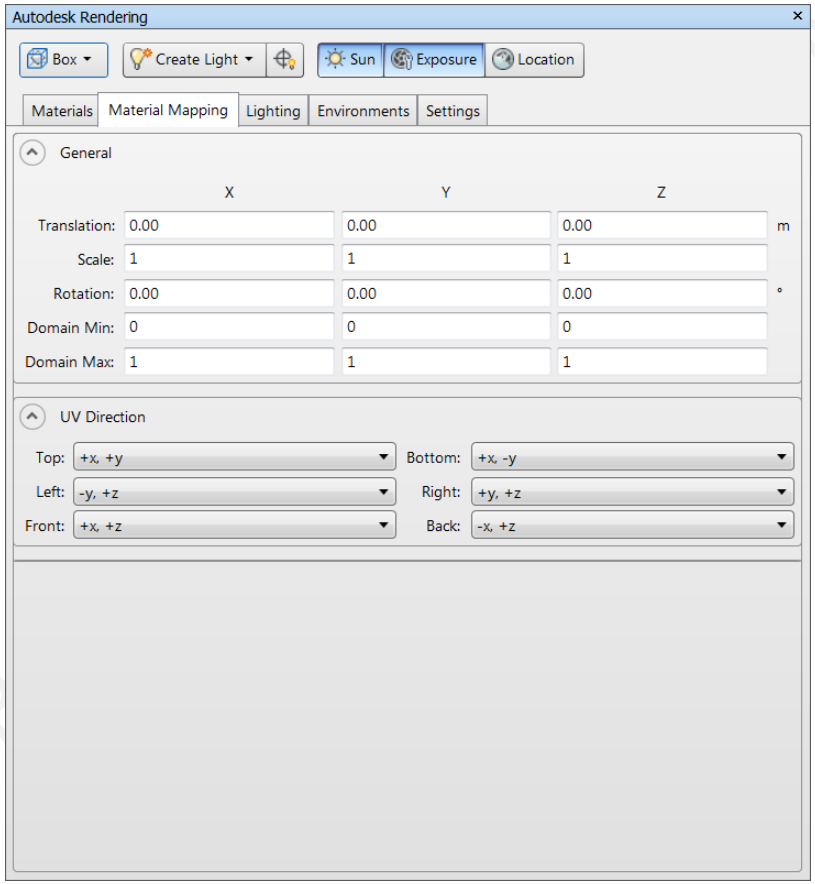
2.6 Working with the Autodesk Rendering Module

Material Mapping Tab

When you assign materials to the objects, you need to make sure the materials are mapped correctly on the objects. This is done using the **Material Mapping** tab. This tab by default is blank and will only show options when you select the geometry object from the graphics window or the **Selection Tree**. Depending on the object to which you have assigned the material, the Autodesk Navisworks will decide the best mapping based on planar, box, cylindrical, and spherical mappings. To change the mapping type, click on the **Box** flyout on the top left corner of the **Autodesk Rendering** window. Based on the mapping selected, the options will change in the **Material Mapping** tab. Figure 4 shows this tab with the box mapping options.

Prepared By Kamu
Do Not Di

2.6 Working with the Autodesk Rendering Module



Prepared By: K.Kordani PMP, MSc

2.6 Working with the Autodesk Rendering Module

Procedure to Change the Material Mapping Type

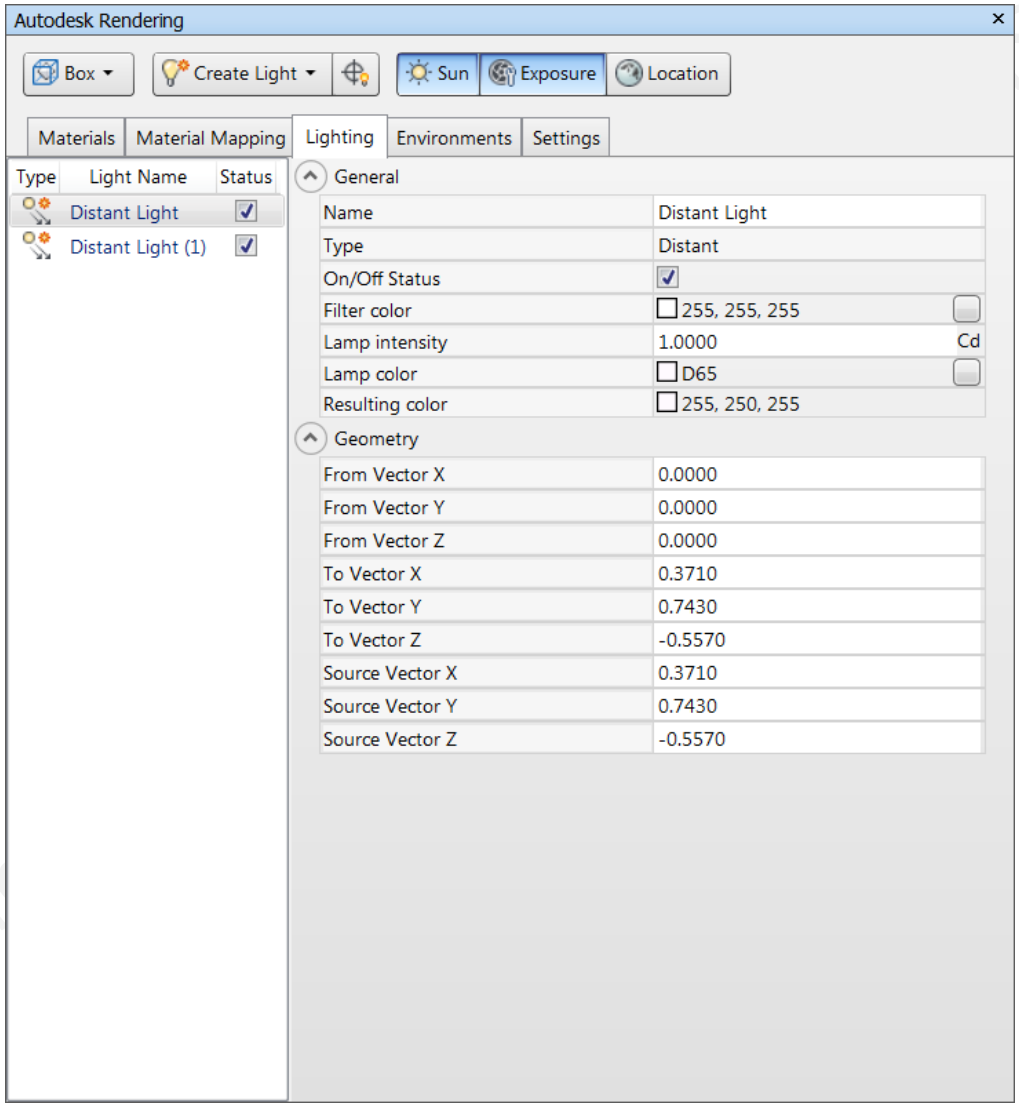
The following steps show how to change the material mapping type specified for the selected object.

a. Set the object whose mapping type you want to change. Note that if you are selecting the object from the graphics window, you will need to change the selection resolution to last unique.

b. From the **Autodesk Rendering** window, click the **Material Mapping** flyout on the top right corner and select the required mapping type.

Prepared By Kan... Do Not D...

2.6 Working with the Autodesk Rendering Module



ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.6 Working with the Autodesk Rendering Module

Procedure for Adding a Point Light to the Scene

The following steps show how to add a point light to the scene.

a. From the **Create Light** flyout, click **Point Light**; if you cursor is currently on an object in the scene, the point light glyph will be displayed so that you can specify the location of the light. However, if the cursor is in the blank area of the scene, it will change to the hand cursor. It is recommended that you place the point lights on certain objects in the scene, such as ceilings, walls, and so on.

b. Move the cursor over the face of the object (generally ceilings) and click to specify the location of the point light; the light will be added to the scene and the move gizmo will be displayed on the light glyph in the graphics window. Also, the light information will be displayed in the **Lighting** tab of the **Autodesk Rendering** window.

***Tip**: If you do not want a light to be located on the face of an object, you can initially pick the face while placing the light and then use the move gizmo to move the light to another location.*

c. If need be, use the move gizmo on the light glyph to change the location of the light.

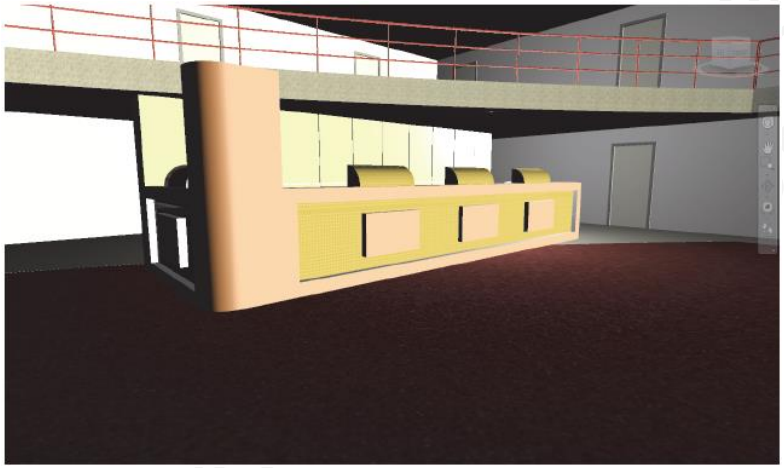
2.6 Working with the Autodesk Rendering Module

d. In the light properties on the **Lighting** tab, click on the **Lamp intensity** swatch to display the **Light Intensity** dialog box in which you can change the light intensity.

e. Click on the **Lamp color** swatch to display the **Lamp Color** dialog box. Select one of the standard lamp colors from the **Standard color** list or click on the **Kelvin colors** radio button and enter a value.

f. Click in the blank area below the lights in the left pane of the **Lighting** tab to turn off the visibility of the move gizmo on the light glyph.

***Tip**: You can use the **Light Glyph** button on the right of the **Create Light** flyout to turn off the visibility of the light glyph from the graphics window.*



ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.6 Working with the Autodesk Rendering Module

Procedure for Adding a Spot Light to the Scene

The spot light needs a position and a target to be defined. The following steps explain how to add a spot light to the scene.

a. From the **Create Light** flyout, click **Spot Light**; if your cursor is currently on an object in the scene, the spot light glyph will be displayed with a location cursor so that you can specify the location of the light. However, if the cursor is in the blank area of the scene, it will change to the hand cursor. It is recommended that you place these lights on certain object in the scene.

b. Move the cursor over the face of the object (generally ceilings) and click to specify the location of the spot light; the light position will be defined.

c. Move the cursor over the object on which you want to target the spot light and click on it; the spot light will be added to the scene and the move gizmo will be displayed on the position of the light glyph in the graphics window. Also, the light information will be displayed in the **Lighting** tab of the **Autodesk Rendering** window.

d. If need be, use the move gizmo on the light glyph to change the location of the light.

e. Click on the sphere displayed at the target location; the move gizmo will be displayed at the target. If need be, use the move gizmo at the target location to change the target.

2.6 Working with the Autodesk Rendering Module

f. In the light properties on the **Lighting** tab, modify the hotspot and falloff angles, if need be.

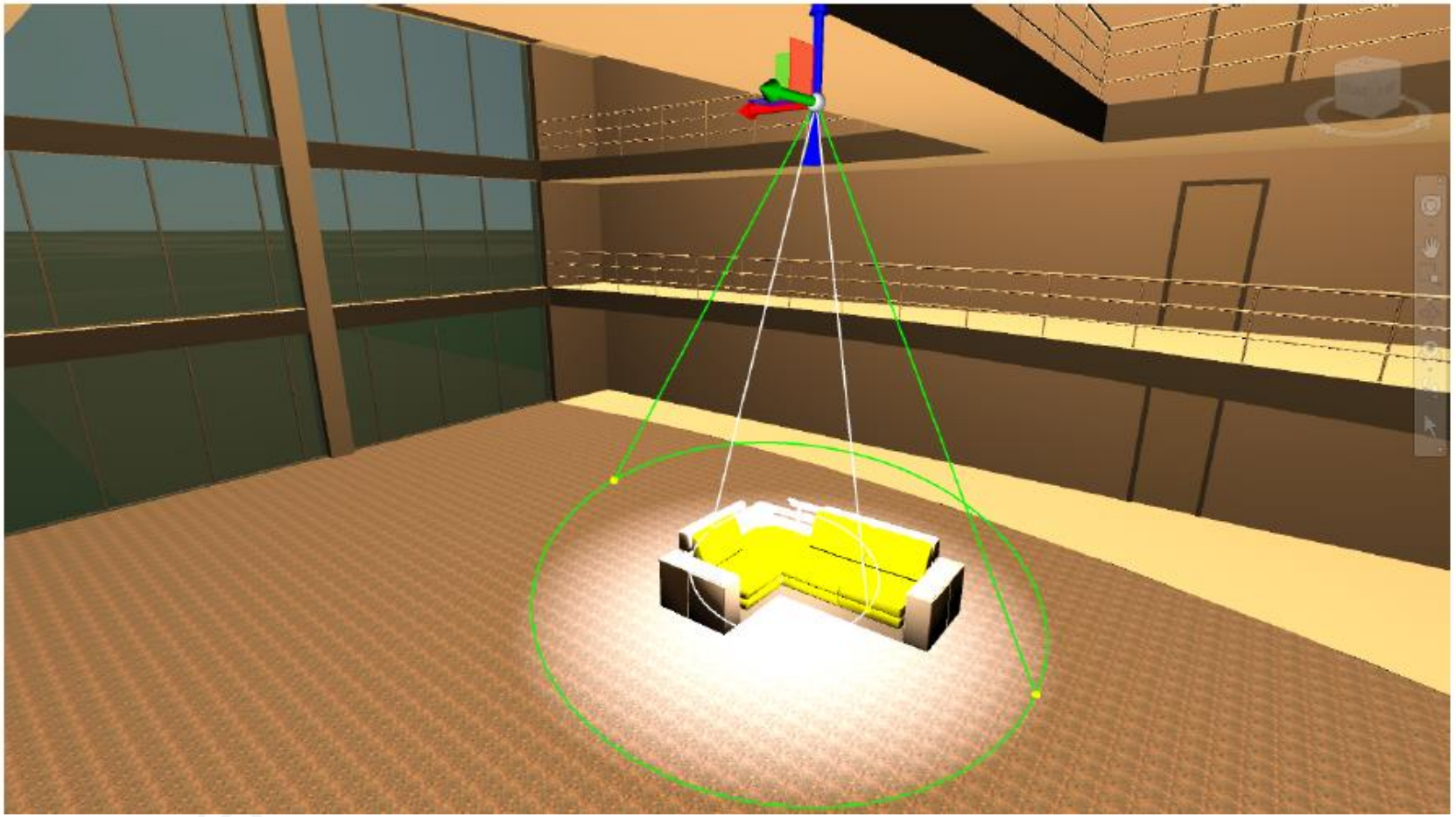
g. Click on the **Lamp intensity** swatch to display the **Light Intensity** dialog box in which you can change the light intensity.

h. Click on the **Lamp color** swatch to display the **Lamp Color** dialog box. Select one of the standard lamp colors from the **Standard color** list or click on the **Kelvin colors** radio button and enter a value.

i. Click in the blank area below the lights in the left pane of the **Lighting** tab to turn off the visibility of the move gizmo on the light glyph.

Prepared By Kambiz MSc
Do Not Distribute

2.6 Working with the Autodesk Rendering Module



ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.6 Working with the Autodesk Rendering Module

Procedure for Adding a Distant Light to the Scene

The following steps show how to add a distant light to the scene.

a. From the **Create Light** flyout, click **Distant Light**; if you cursor is currently on an object in the scene, the distant light glyph will be displayed so that you can specify the location of the light. However, if the cursor is in the blank area of the scene, it will change to the hand cursor. This is because the distant lights cannot be located in the blank area. You need to place them on certain objects in the scene.

b. Move the cursor over the face of the object and click to specify the location of the distant light; the light position will be defined.

c. Move the cursor over the object on which you want to target the distant light and click on it; the distant light will be added to the scene and the move gizmo will be displayed on the position of the light glyph in the graphics window. Also, the light information will be displayed in the **Lighting** tab of the **Autodesk Rendering** window.

d. If need be, use the move gizmo on the light glyph to change the location of the light.

e. Click on the sphere displayed at the target location; the move gizmo will be displayed at the target. If need be, use the move gizmo at the target location to change the target.

www.dralavipour.com ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.6 Working with the Autodesk Rendering Module

f. Click on the **Lamp intensity** edit box to change the light intensity.

g. Click on the **Lamp color** swatch to display the **Lamp Color** dialog box. Select one of the standard lamp colors from the **Standard color** list or click on the **Kelvin colors** radio button and enter a value.

h. Click in the blank area below the lights in the left pane of the **Lighting** tab to turn off the visibility of the move gizmo on the light glyph.

Prepared By Kambiz
Do Not Distr

2.6 Working with the Autodesk Rendering Module

Procedure for Adding a Web Light to the Scene

The following steps show how to add a web light to the scene.

a. From the **Create Light** flyout, click **Web Light**; if you cursor is currently on an object in the scene, the web light glyph will be displayed so that you can specify the location of the light. However, if the cursor is in the blank area of the scene, it will change to the hand cursor. This is because the web lights cannot be located in the blank area. You need to place them on certain objects in the scene.

b. Move the cursor over the face of the object and click to specify the location of the web light; the light position will be defined.

c. Move the cursor over the object on which you want to target the web light and click on it; the web light will be added to the scene and the move gizmo will be displayed on the position of the light glyph in the graphics window. Also, the light information will be displayed in the **Lighting** tab of the **Autodesk Rendering** window.

d. If need be, use the move gizmo on the light glyph to change the location of the light.

e. Click on the sphere displayed at the target location; the move gizmo will be displayed at the target. If need be, use the move gizmo at the target location to change the target.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.6 Working with the Autodesk Rendering Module

f. Click on the **Lamp intensity** edit box to change the light intensity.

g. Click on the **Lamp color** swatch to display the **Lamp Color** dialog box. Select one of the standard lamp colors from the **Standard color** list or click on the **Kelvin colors** radio button and enter a value.

h. If you have the .IES format file of the light distribution information provided by the light manufacturer, click the **Web file** swatch; the **Open** dialog box will be displayed. Browse and select the file and click **Open** in the dialog box.

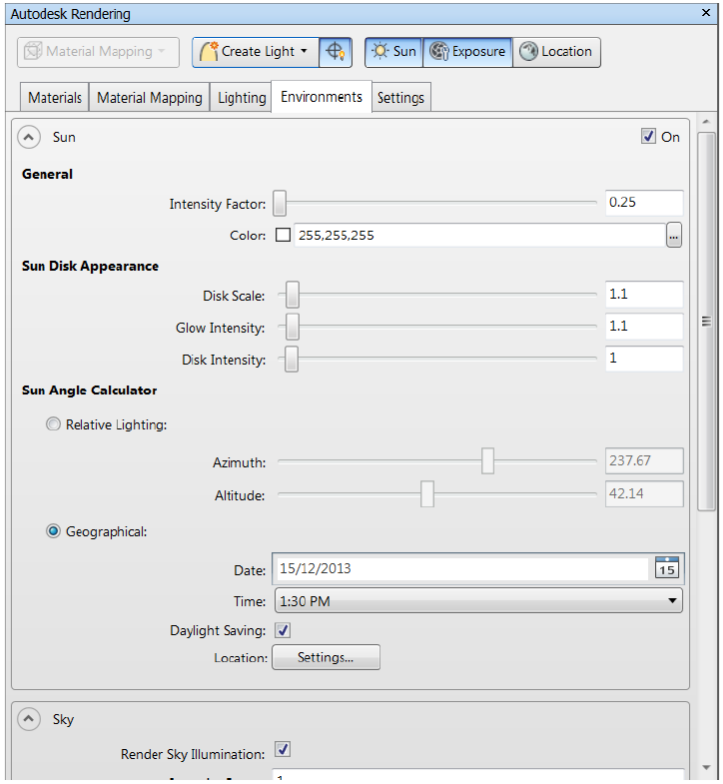
i. Click in the blank area below the lights in the left pane of the **Lighting** tab to turn off the visibility of the move gizmo on the light glyph.

Prepared By K... Do Not...

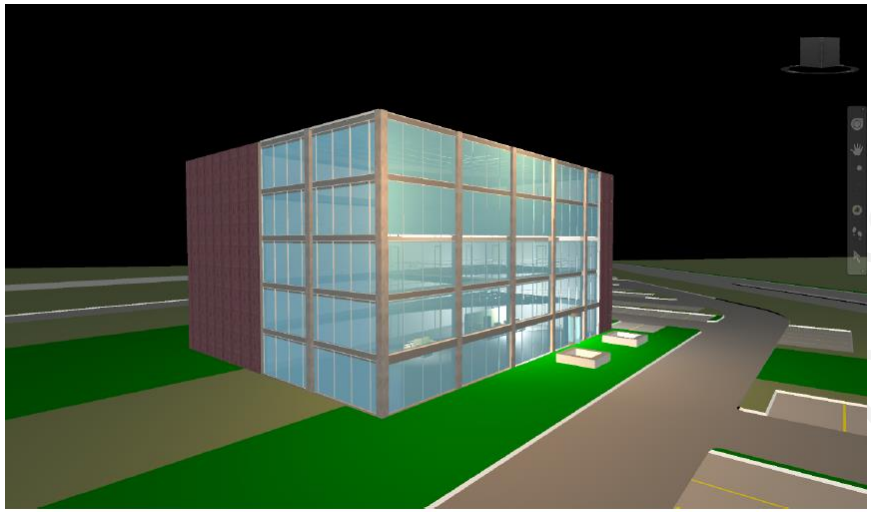
2.6 Working with the Autodesk Rendering Module

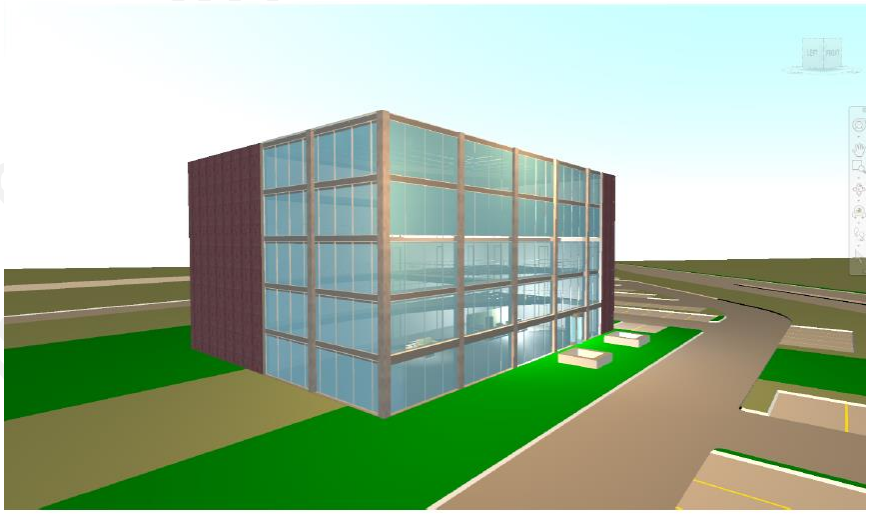
Environments Tab

The **Environments** tab is used to control the display and settings of the sun light, sky and exposure in the scene, as shown in Figure 9. The three areas displayed in this tab are discussed next.



2.6 Working with the Autodesk Rendering Module

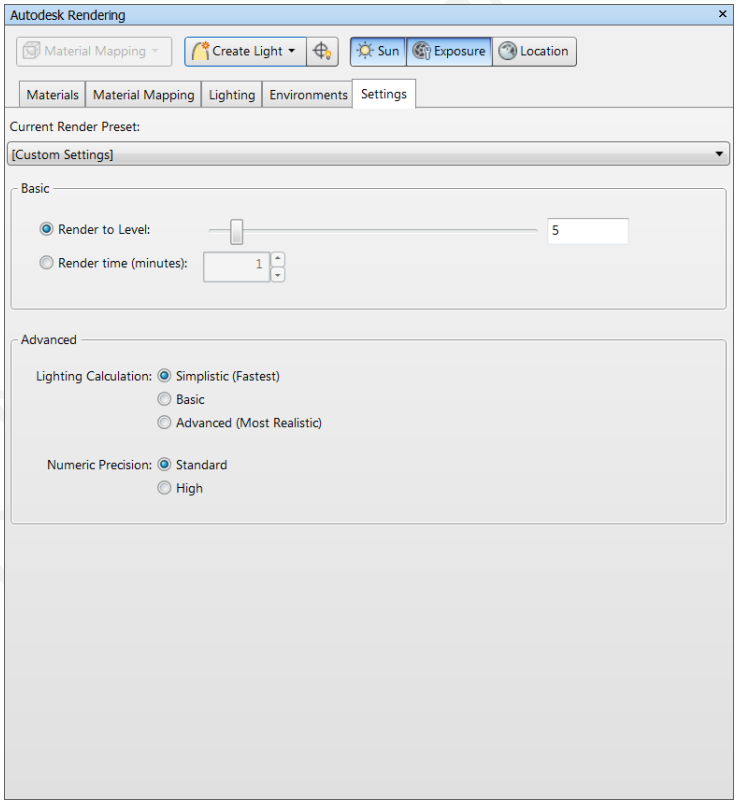




Prepared By: K.Kordani PMP, MSc
Do Not Distribute

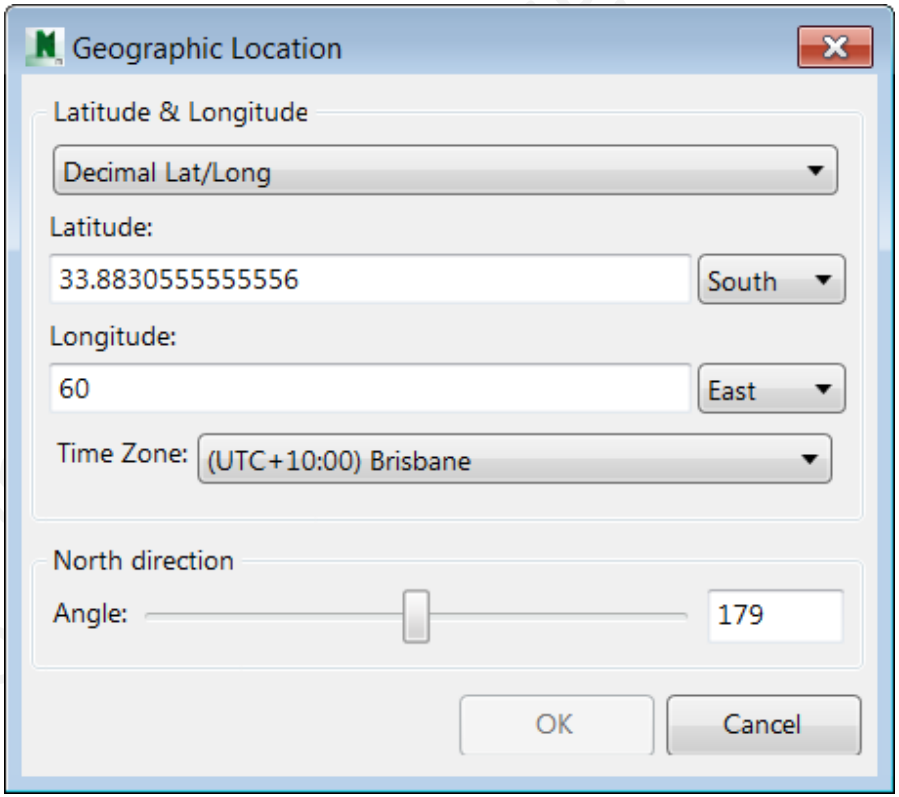
Settings Tab

This tab, shown in Figure 12, is used to control the render style presets that are available in the **Ray Trace** flyout in the **Render** ribbon tab > **Interactive Ray Trace** ribbon panel. The various options available in the tab are discussed next.



ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

Location Settings



www.dralavipour.com ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

RAY TRACE RENDERING OF THE AUTODESK NAVISWORKS SCENE

Render Ribbon Tab > Interactive Ray Trace Ribbon Panel > Ray Trace Flyout

The **Ray Trace** flyout allows you to interactively render your Autodesk Navisworks scene based on one of the render presets. The following are the various presets available in this flyout to render the scene.

Low Quality: This rendering is used when you want to quickly check how the scene looks with the current material and lighting. This preset renders the scene to level 5 with the lighting calculation set to fastest and the numeric precision set to standard in the **Settings** tab of the **Autodesk Rendering** window.

Medium Quality: This rendering is used to preview the final rendering. This preset renders the scene to level 10 with the lighting calculation set to basic and the numeric precision set to standard in the **Settings** tab of the **Autodesk Rendering** window.

High Quality: This rendering is used to create high quality rendered images of your scene. This preset renders the scene to level 20 with the lighting calculation set to most realistic and the numeric precision set to high in the **Settings** tab of the **Autodesk Rendering** window.

www.dralavipour.com ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.6 Working with the Autodesk Rendering Module

Coffee Break Rendering: This preset is similar to **Low Quality** preset, with the difference being the render time in this preset is set to 10 minutes so the rendering can be completed over the coffee break.

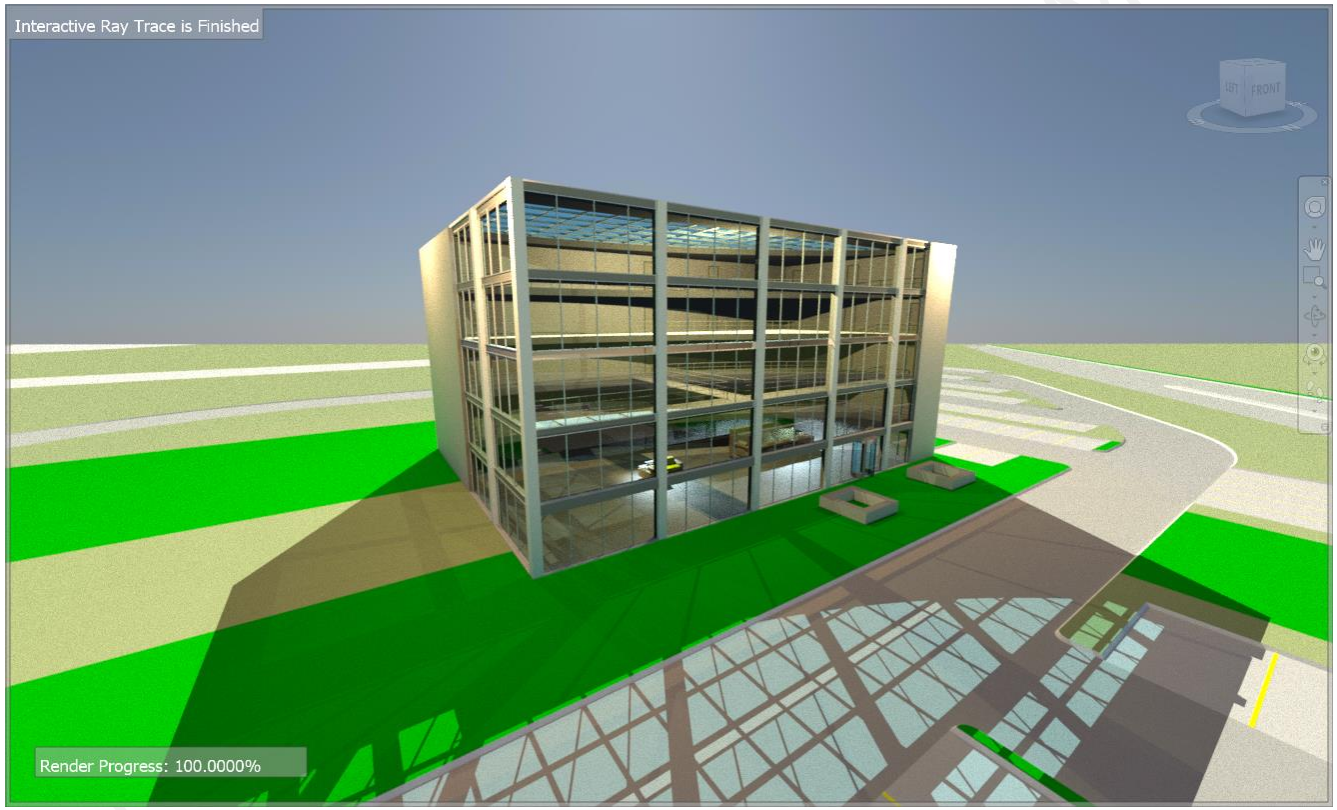
Lunch Break Rendering: This preset is similar to **Medium Quality** preset, with the difference being the render time in this preset is set to 60 minutes so the rendering can be completed over the lunch break.

Overnight Rendering: This preset is similar to **High Quality** preset, with the difference being the render time in this preset is set to 720 minutes. This allows the scene to be rendered overnight and create the best quality rendered image.

Custom Settings: This preset allows you to render the scene using the custom settings defined in the **Settings** tab of the **Autodesk Rendering** window.

www.dralavipour.com ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.6 Working with the Autodesk Rendering Module



ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

Pausing/Stopping the Interactive Rendering

Render Ribbon Tab > Interactive Ray Trace Ribbon Panel > Pause/Stop

SAVING RENDERED IMAGES

Render Ribbon Tab > Export Ribbon Panel > Image

Autodesk Navisworks allows you to save the rendered Autodesk Navisworks scene in various image formats. This is done using the **Image** tool. Note that this tool will only be available once the ray trace rendering of the scene is completed. When you choose this button, the **Save As** dialog box will be displayed that allows you to specify the name and location of the rendered image. You can select the required image format type from the **Save as type** list in this dialog box.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

RENDERING IN CLOUD

Render Ribbon Tab > System Ribbon Panel > Render in Cloud

Rendering in cloud is a new option added to Autodesk Navisworks 2015. This option allows you to offload the rendering process to Autodesk 360 cloud rendering services, ensuring your computer resources remain free for other tasks. It is important to remember that this is an Autodesk Subscription service and is only available to the customers who have active subscription with Autodesk software.

***Note**; While rendering in cloud, only the **Exterior: Sun only** or the **Interior: Sun only** lighting scheme will be used.*

Prepared By Kar... Kordani
Do Not Distribute

۲. مدلسازی اطلاعات ساخت با استفاده از نرم افزار Navisworks

Kambiz Kordani PMP,MSc
Linkedin: @kordani-kambiz
Instagram: @acemi_social

2.7 Working with the Animator and Scripter Modules

- Introduce you to the Animator module of Autodesk Navisworks.
- Familiarize you with the process of creating animations of moving objects.
- Explain how to control the visibility of the objects during the animation.
- Teach you how to add a camera animation to the animator scene.
- Teach you how to add a section plane animation to the animator scene.
- Introduce you to the Scripter module of Autodesk Navisworks.
- Teach you how to add various types of scripts to the animations.
- Teach you how to create animations of objects falling from the sky to be linked to the timeliner simulation.

Kambiz Kordani PMP, MSc
Do Not Distribute

www.dralavipour.com ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

THE ANIMATOR MODULE

Home Ribbon Tab > Tools Ribbon Panel > Animator

In the previous chapters, you have learned about how to create animations of navigating through the model. However, a number of times you need to display animation of objects moving in the model. For example, in case of a plant model, you can show the animation of a skid moving to its assembly position in the design. Similarly, in case of a building model, you may want a truck to bring the load to the construction site and a crane lifting the load from the truck. All these things can be done using the **Animator** module of Autodesk Navisworks.

Animations Based on Animation Sets

Animation set is a selection of objects that will be animated. In this type of animation, you can translate the selected objects, rotate them, scale them, and change their visibility and colors. This is done by capturing keyframes at regular intervals that capture the changes in the selected objects at those intervals.

Camera Animation

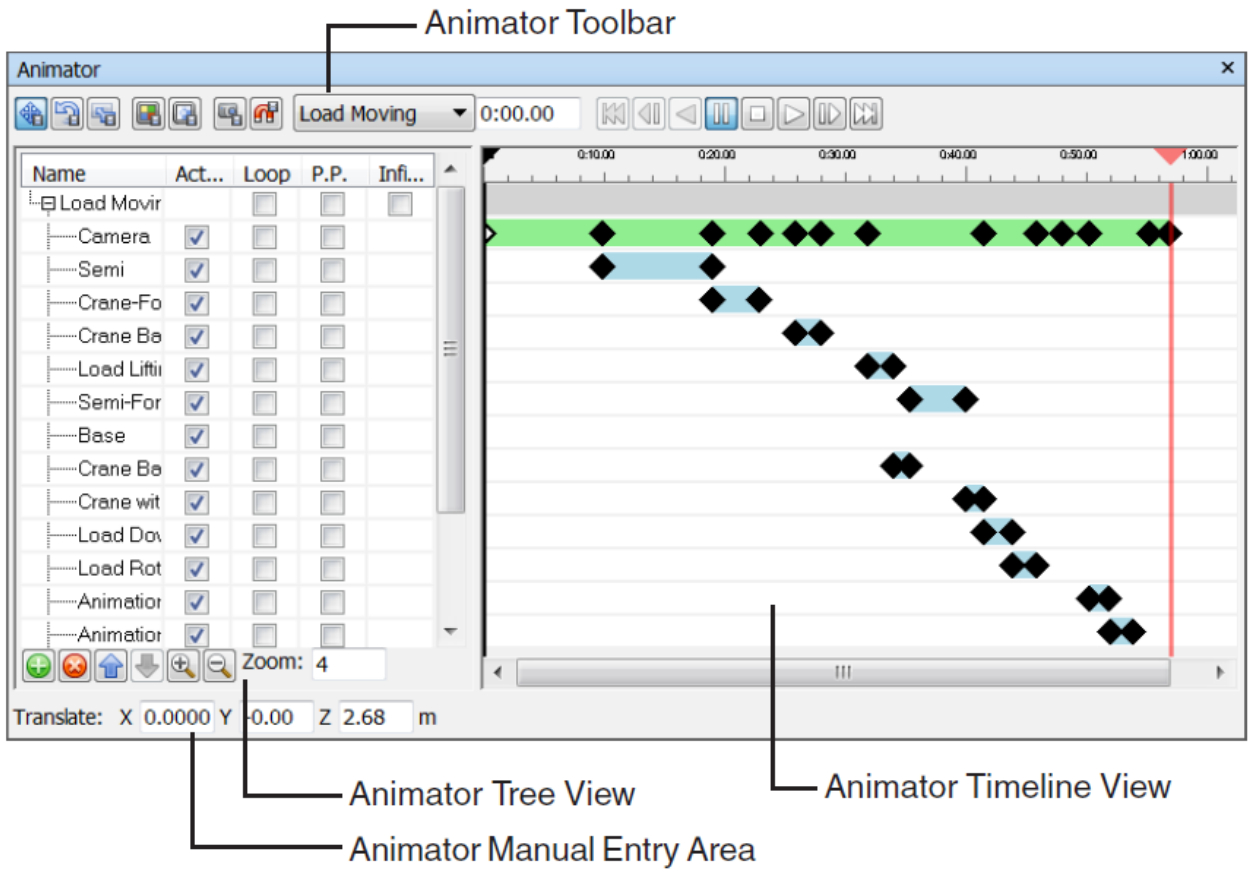
To navigate through the model as the objects are getting animated.

Section Plane Animation

To animate the sectioning of a model.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.7 Working with the Animator and Scripter Modules



PROCEDURE FOR CREATING ANIMATOR ANIMATIONS

The following sections show you how to create various types of animator animations.

Animating the Object Transformation

The following steps show you how to animate the object movements:

a. If the **Animator** window is not already docked on the screen, from the **Home** ribbon tab > **Tools** ribbon panel, click the **Animator** button to display this window.

b. Dock the **Animator** window below the graphics window.

c. Right-click in the **Animator** tree view area and from the shortcut menu, click **Add Scene**; a new animator scene will be added.

d. Rename the animator scene to a relevant name.

e. From the **Selection Tree** or the graphics window, select the objects that you want to animate. You can also click on a saved selection or search set to select the objects.

f. Right-click on the scene that you added earlier and from the shortcut menu, select **Add Animation Set > From current selection** or **From current search/selection set**; a new animation set is added to the scene with the default name of **Animation Set 1**.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.7 Working with the Animator and Scripter Modules

g. Rename the newly added animation set to a relevant name.

h. With the animation selected in the animator scene, click the **Translate animation set**, **Rotate animation set**, or **Scale animation set** button from the **Animator** toolbar, depending on whether you want to move, rotate, or scale the selected objects.

i. Press and hold the CTRL key down and drag the transform tool gizmo to the required location on the selected object in the graphics window.

What I do

*While transforming the animation set, especially using the **Rotate animation set** tool, it is very important that the rotate gizmo is properly aligned with the rotation axis even before you capture the start keyframe. In AutoCAD-based programs, I generally draw a line to represent the center of the axis so that I can snap to it while dragging the gizmo.*

j. Click the **Capture keyframe** button from the **Animator** toolbar; a keyframe is added at 0 seconds.

k. In the **Time position** box of the **Animator** toolbar, enter the time duration for which you want to tun this part of the animation. For example, if you want this animation to run for 10 seconds, type 10 in this box and press ENTER.

2.7 Working with the Animator and Scripter Modules

l. From the **Animator** toolbar, click the **Translate animation set**, **Rotate animation set**, or **Scale animation set** button, depending on whether you want to move, rotate, or scale the selected objects.

m. Using the gizmo or the **Animator** manual entry area, specify the transformation values. If you are rotating the objects, make sure you click on the origin of the gizmo and move it to the proper rotation center. Alternatively, you can use the **cX**, **cY**, and **cZ** boxes in the **Animator** manual entry area to specify the coordinates of the center point of rotation.

***Tip**: While transforming the objects, you can navigate around the model to make sure you are making the right selections.*

n. Once the object is transformed to the new position, click the **Capture keyframe** button on the **Animator** toolbar; a new keyframe will be added at the specified time.

o. From the **Animator** toolbar, click the **Stop** button to move the timeline back to the start. Also, click the button of the tool that you used to transform the objects to exit that tool and turn off the gizmo from the screen.

p. Press the ESC key to deselect the objects.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.7 Working with the Animator and Scripter Modules

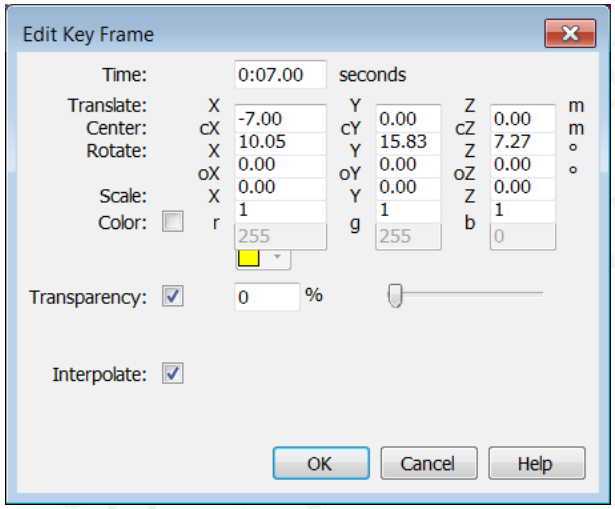
q. From the **Animator** toolbar, click the **Play** button to play the animation.

***Tip**: While adding the animation set in step f, if you made a mistake in the object selection, you can reselect the correct objects and right-click on the added animation set and update it using the shortcut menu.*

Prepared By Kambiz Korda, MSc
Do Not Distribute

Controlling the Transparency of the Objects in the Animation Set

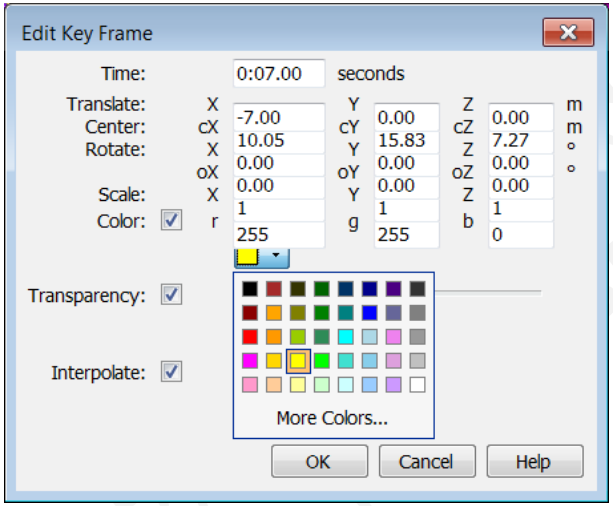
As mentioned earlier, while animating objects using the **Animator** window, you can also control the transparency of the objects in the animation set.



www.dralavipour.com ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

Changing the Color of the Objects in the Animation Set

As mentioned earlier, while animating objects using the **Animator** window, you can also change the color of the objects in the animation set.



Adding a Blank Camera Animation to the Animator Scene

The following steps show you how to add a blank camera animation to the animator scene:

a. Press ESC to make sure nothing is selected.

b. In the **Animator** tree view area, right-click on the animator scene and from the shortcut menu, select **Add Camera > Blank camera**; a new camera animation will be added with the default name of **Camera**.

c. Rename the camera animation to a relevant name.

d. From the **Animator** toolbar, click the **Capture keyframe** button; a keyframe is added at 0 seconds.

e. In the **Time position** box of the **Animator** toolbar, enter the time duration that you want the camera to take in navigating from the start position to the end position. For example, if you want the camera to take 10 seconds to navigate from the start position to the end position, type 10 in this box and press ENTER.

f. Using any navigation tool, preferably the **Walk** tool, navigate to the area of the design where you want the camera to be at the end of the animation.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.7 Working with the Animator and Scripter Modules

g. Click the **Capture keyframe** button on the **Animator** toolbar; a new keyframe will be added at the specified time.

h. From the **Animator** toolbar, click the **Stop** button to move the timeline back to the start.

i. From the **Animator** toolbar, click the **Play** button to play the animation; the camera moves from the start position to the end position during the time that you specified. If there was any other animation created during the same time period, that animation will also play simultaneously with the camera animation.

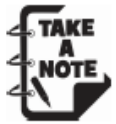
Prepared By Kambiz ... MSc
Do Not Distri...

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

Adding a Saved Viewpoint Animation to the Animator Scene

Before adding a saved viewpoint animation to the animator scene, it is important for you to know that the viewpoint animation will be added with original time duration that it was created with. As a result, you need to first edit the time duration of this animation to match the time duration that you want it to play for in the animator scene. If need be, make a copy of this animation before editing it. The following steps show you how to add a saved viewpoint animation to the animator scene.

a. From the **Saved Viewpoints** window, select the animation that you want to add to the animator scene. Make sure its duration is edited to match the duration you want it to run for in the animator scene.

b. In the **Animator** tree view area, right-click on the animator scene and from the shortcut menu, select **Add Camera > From current viewpoint animation**; a new camera animation will be added with the default name of **Camera**.

***Note**: If the viewpoint animation was created using the saved viewpoints, keyframes will be added for each viewpoint in the animation. However, if the viewpoint animation was created by recording the navigation, a keyframe will be added for each frame in the animation. This will make the **Animator** timeline view look messy.*

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.7 Working with the Animator and Scripter Modules

c. Rename the camera animation to a relevant name.

d. From the **Animator** toolbar, click the **Stop** button to move the timeline back to the start.

e. From the **Animator** toolbar, click the **Play** button to play the animation. If there was any other animation created during the same time period, that animation will also play simultaneously with the camera animation.

Prepared By Kambiz Ko
Do Not Distribu

Adding a Section Plane Animation to the Animator Scene

The following steps show you how to add a section plane animation to animate the sectioning of the model:

a. Press ESC to make sure nothing is selected.

b. In the **Animator** tree view area, right-click on the animator scene and from the shortcut menu, select **Add Section Plane**; a new animation called **Section Plane** is added to the scene.

c. From the **View** ribbon tab > **Sectioning** ribbon panel, click **Enable Sectioning**; the **Sectioning Tools** ribbon tab will be displayed.

d. Using the **Current Plane** list in the **Plane Settings** ribbon panel, select the plane that you want to use for sectioning. You can also select two planes and link them, if required.

e. Using the **Move** tool, move the section plane to the start position of sectioning.

f. In the **Animator** toolbar, click **Capture keyframe**; a keyframe will be added for the section animation.

2.7 Working with the Animator and Scripter Modules

g. In the **Time position** box of the **Animator** toolbar, enter the time duration that you want the sectioning to run for. For example, if you want the section plane to take 10 seconds to move from the start position to the end position, type 10 in this box and press ENTER.

h. In the graphics window, use the **Move** gizmo to move the section plane to the end position.

i. In the **Animator** toolbar, click **Capture keyframe**; a keyframe will be added for the section animation and the timeline bar will be displayed in Red color.

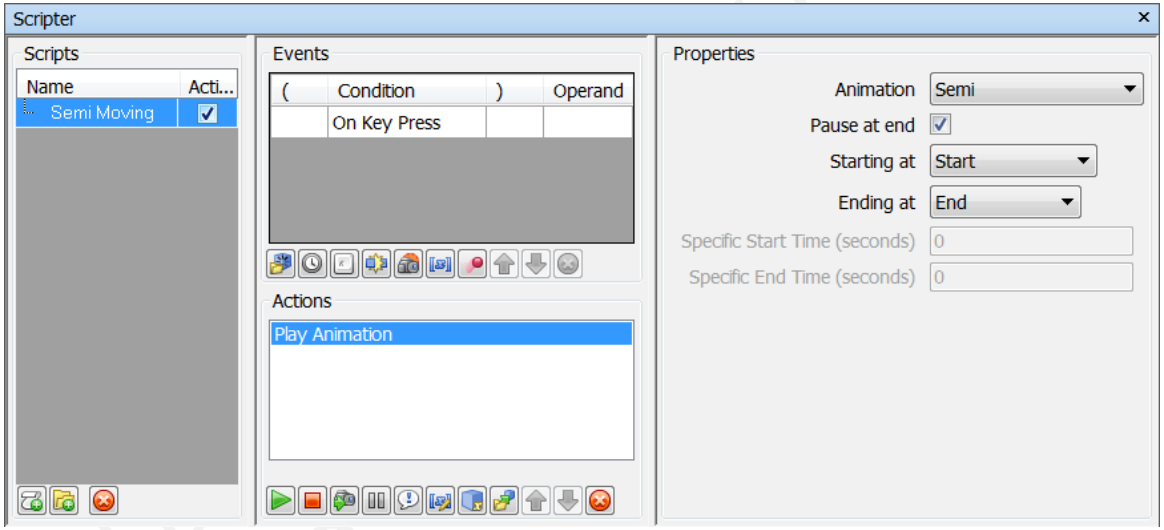
j. From the **Sectioning Tools** ribbon tab > **Transform** ribbon panel, turn off the **Move** tool by clicking on it; the section plane visibility will be turned off.

k. In the **Animator** toolbar, click the **Stop** button to stop the animation; the section plane will disappear and the model will be displayed without sectioning.

l. Click the **Play** button to play the animation of sectioning of the model.

THE SCRIPTER MODULE

Home Ribbon Tab > Tools Ribbon Panel > Scripter



ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

ENABLING THE SCRIPTS

Animation Ribbon Tab > Script Ribbon Panel > Enable Scripts

PROCEDURE TO ADD INTERACTIVITY TO THE MODEL USING SCRIPTS

The following steps show you how you can add interactivity to the Autodesk Navisworks model using scripts:

a. If the **Scripter** window is not already docked on the screen, from the **Home** ribbon tab > **Tools** ribbon panel, click the **Scripter** button to display this window.

b. Dock the **Scripter** window below the graphics window.

c. From the **Animation** ribbon tab > **Script** ribbon panel, click **Enable Scripts**, if it is not already turned on.

d. In the **Scripter** window, right-click in the **Scripts** area to add a new script. Rename it to the desired name.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.7 Working with the Animator and Scripter Modules

e. With the newly added script selected, select the required event to trigger the script from the toolbar at the bottom of the **Events** area. For example, if you want the script to be triggered at the press of a key, click the **On Key Press** button.

f. From the **Properties** area, set the information related to the event that you selected. For example, if you selected the **On Key Press** event, specify the key and its trigger in the **Properties** area.

g. With the event selected, select the action to be performed when the script is triggered from the toolbar in the **Actions** area. For example, if you want to start playing an animation, click the **Play Animation** button.

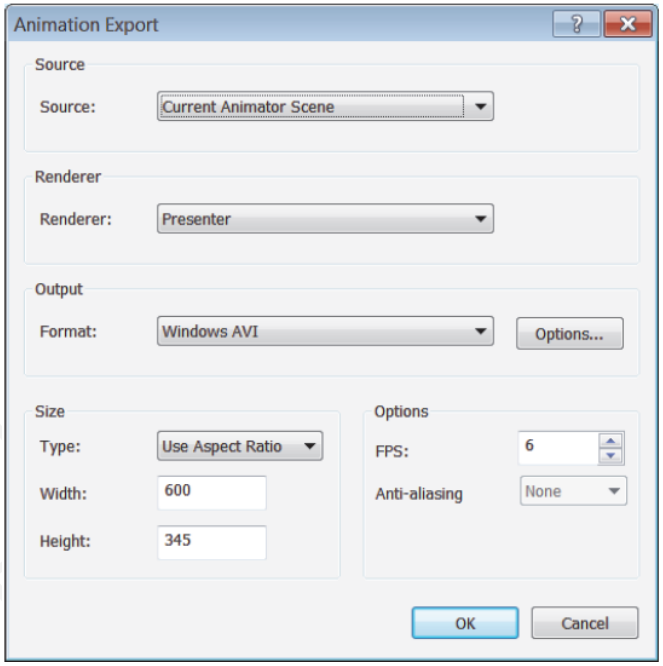
h. From the **Properties** area, set the information related to the action that you selected. For example, if you selected the **Play Animation** action, specify the animation to play and its related information in the **Properties** area.

i. Repeat steps d to g to add more events and actions to the script.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

EXPORTING ANIMATIONS

Output Ribbon Tab > Visuals Ribbon Panel > Animation



Prepared By K.Kordani PMP, MSc

۲. مدلسازی اطلاعات ساخت با استفاده از نرم افزار

Navisworks

Kambiz Kordani PMP,MSc
Linkedin: @kordani-kambiz
Instagram: @acemi_social

2.8 Creating Construction Simulations Using the TimeLiner Module

- Introduce you to the TimeLiner module of Autodesk Navisworks.
- Familiarize you with the TimeLiner window.
- Teach you how to automatically create tasks for all sets in Autodesk Navisworks scene.
- Teach you how to manually create tasks within Autodesk Navisworks.
- Teach you how to create timeliner schedules from external data source and project files.
- Teach you how to use rules to link external projects to the objects in the scene.
- Teach you how to link animator and scripter scenes to the timeline tasks.
- Teach you how to modify the overlay text in the simulation animation.

Kambiz Kordani PMP,MSc
Do Not Distribute

www.dralavipour.com ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

THE TIMELINER MODULE

Home Ribbon Tab > Tools Ribbon Panel > TimeLiner

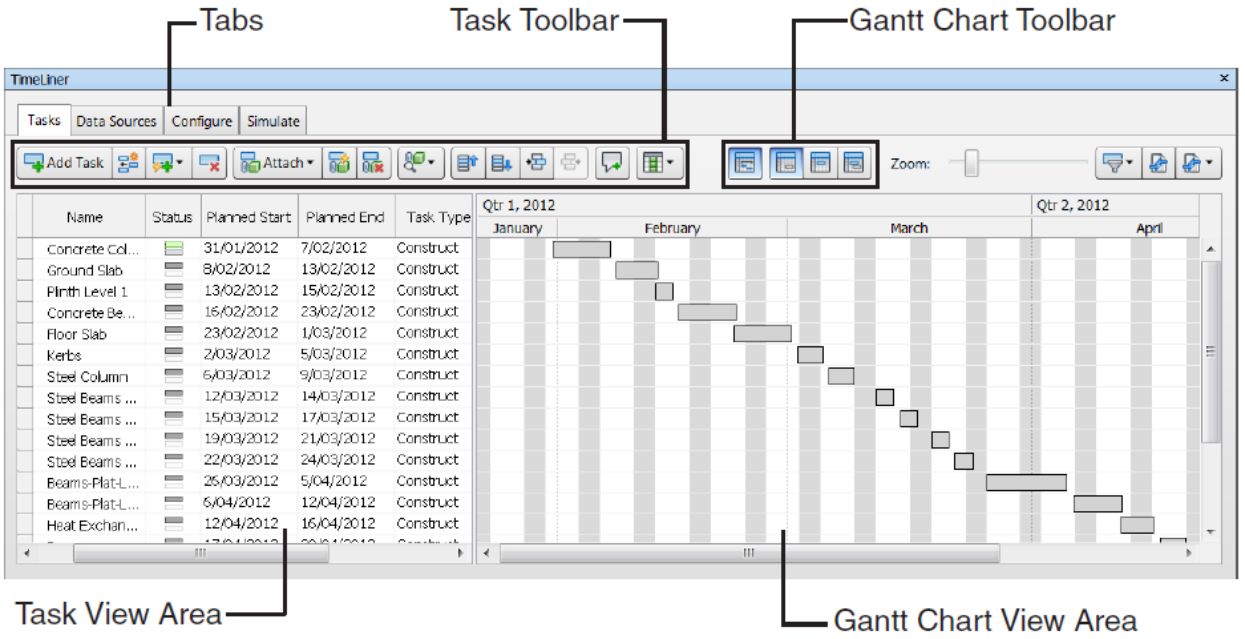
The **TimeLiner** module of Autodesk Navisworks allows you to create construction simulations by attaching the objects in the Autodesk Navisworks scene to the scheduling tasks. These tasks can be manually created within the Autodesk Navisworks file or can be imported from an external scheduling project file. You can also automatically create the tasks based on the sets in the current file. All this is done using the **TimeLiner** window.

Prepared By Ka... Do Not D...

2.8 Creating Construction Simulations Using the TimeLiner Module

Tasks Tab

This tab displays the tasks and their associated Gantt chart and also the tool buttons to manipulate them. All these are discussed next.



Tasks Tab

Task View Area

This area shows the names of the tasks created in the Autodesk Navisworks file or imported from the external project file and their associated information in various columns. You can add or remove columns from this area by using the **Columns** flyout, which is the last button in the **Task** toolbar.

Task Toolbar

The **Task** toolbar displays various buttons to add and manipulate the timeliner tasks. These buttons are discussed next.

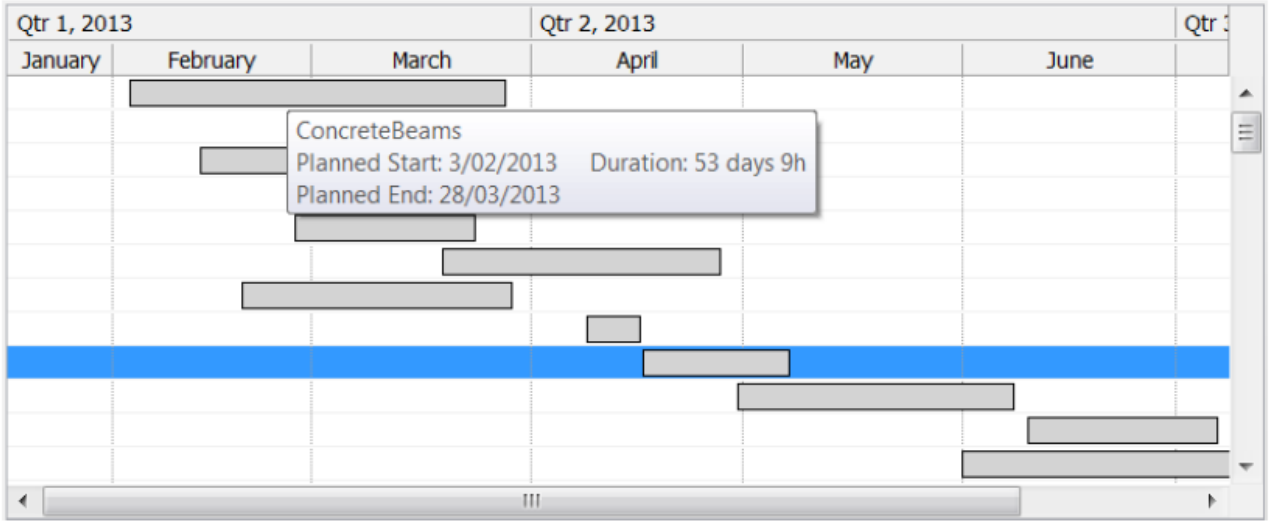
Prepared By Kan... Do Not D...

2.8 Creating Construction Simulations Using the TimeLiner Module

Tasks Tab

Gantt Chart View Area

This area shows the schedule tasks in the form of bars representing the start and end dates of each task. Moving the cursor over a bar shows you the details of that task, such as the name of the task, the start and end dates, and the duration of the task. This is a quick way to find out if there are any overlapping tasks in the current schedule. Figure 7 shows a Gantt chart view showing the information about a task.



2.8 Creating Construction Simulations Using the TimeLiner Module

Tasks Tab

Zoom Slider

This slider is located on the right of the Gantt chart toolbar and is used to zoom in or out of the Gantt chart view area.

Filter by Status Flyout

This flyout allows you to filter the timeliner tasks based on the options shown in Figure 8.

Export to Sets

This button creates the selection sets of objects contained in each timeliner task. The name of the sets will be the same as the name of the timeliner task. Also, after exporting the sets, the **Attached** column in the **Tasks** tab will show the sets attached instead of **Explicit Attachment** that it was showing before you exported the sets. Figure 9 shows the **TimeLiner** window showing various tasks and the sets created for the objects contained in those tasks. Also, notice the **Attached** column now shows sets attached.

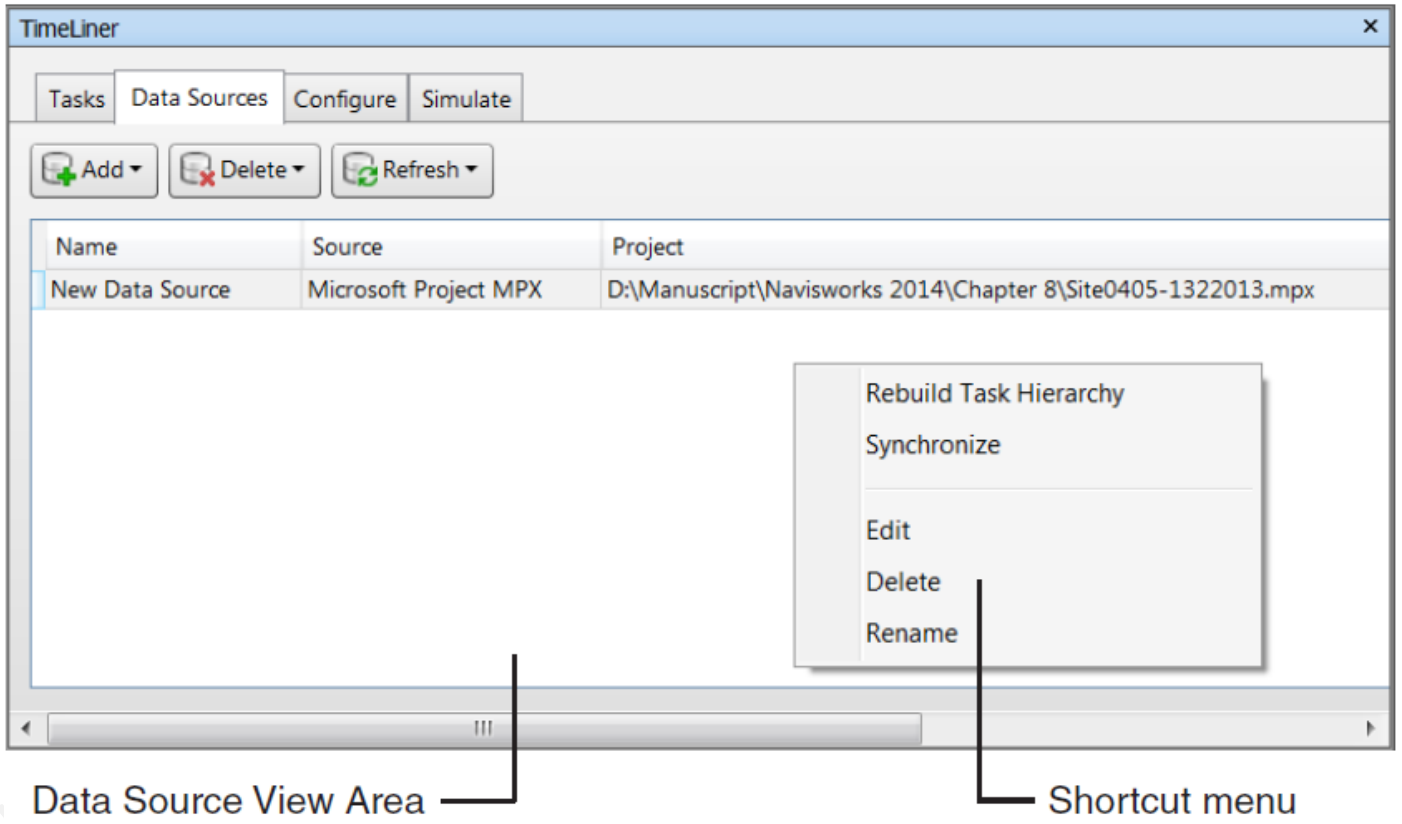
Export the schedule Flyout

This flyout allows you to export the current timeliner tasks as schedules in a CSV file or a Microsoft Project XML file. When you click any of the options from the flyout, the **Export** dialog box will be displayed that allows you to specify the name and the location of the exported file.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.8 Creating Construction Simulations Using the TimeLiner Module

Data Sources Tab



ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

Data Sources Tab

Add Flyout

Delete

This button deletes the data source selected from the data source view area. Note that by default, when you delete a data source, its tasks are not deleted automatically. You need to go the **Tasks** tab and then delete the root data source task at the top of the list.

Refresh Flyout

Clicking this flyout shows you the options of refreshing the selected data source or all the data sources. When you click either of these options, the **Refresh from Data Source** dialog box will be displayed, as shown in Figure 12. These options allow you to refresh the data source and rebuild the task hierarchy from the data source or only synchronize the data source and still maintain the existing task structure.

Data Source View Area

This area shows all the project data sources that are added to the current Autodesk Navisworks scene.

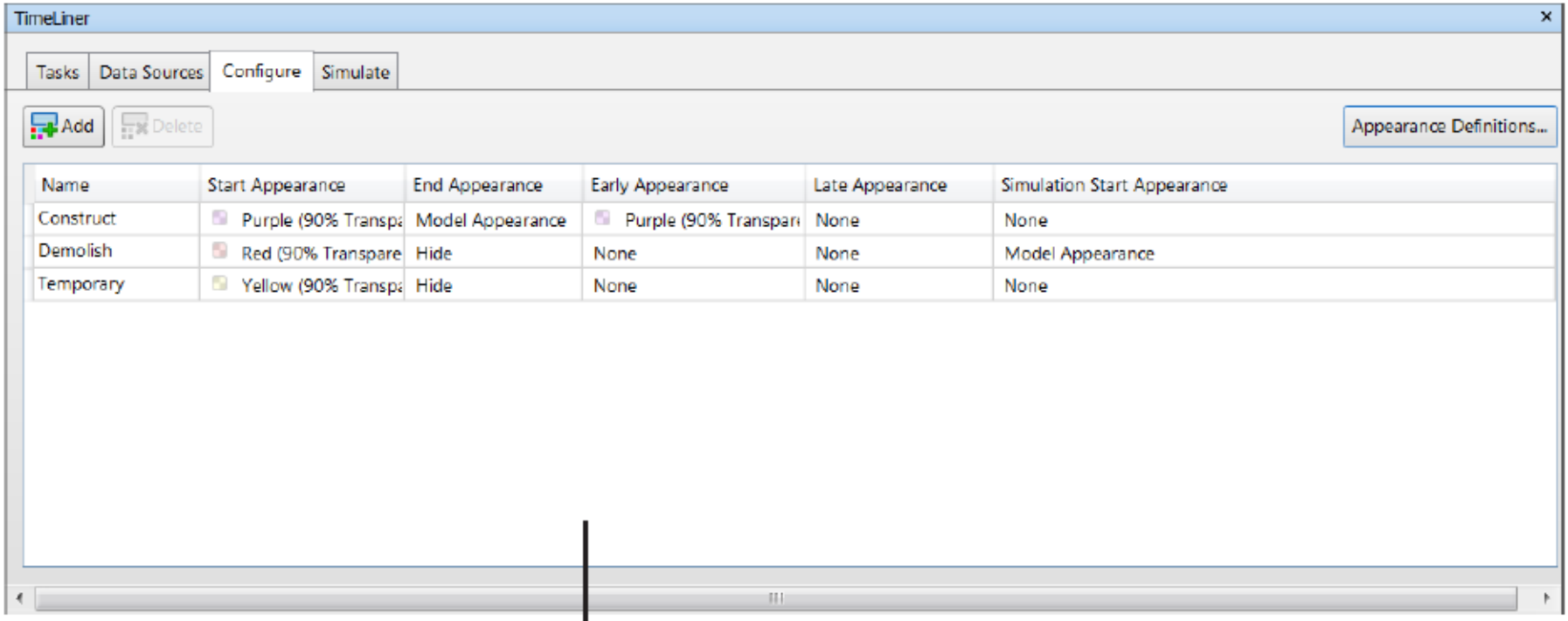
Shortcut Menu

This menu is displayed when you right-click on an existing data source. The options in this shortcut menu are discussed next.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.8 Creating Construction Simulations Using the TimeLiner Module

Configure Tab



ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

Configure Tab

Configuration View Area

This area shows various tasks in rows and their associated appearances in various columns. To change appearance of a task, double-click on the field under the appearance column you want to change and select a different appearance style.

Add

This button allows you to add a new task type as a row that can then be assigned various appearances.

Delete

This button is used to delete the selected task type.

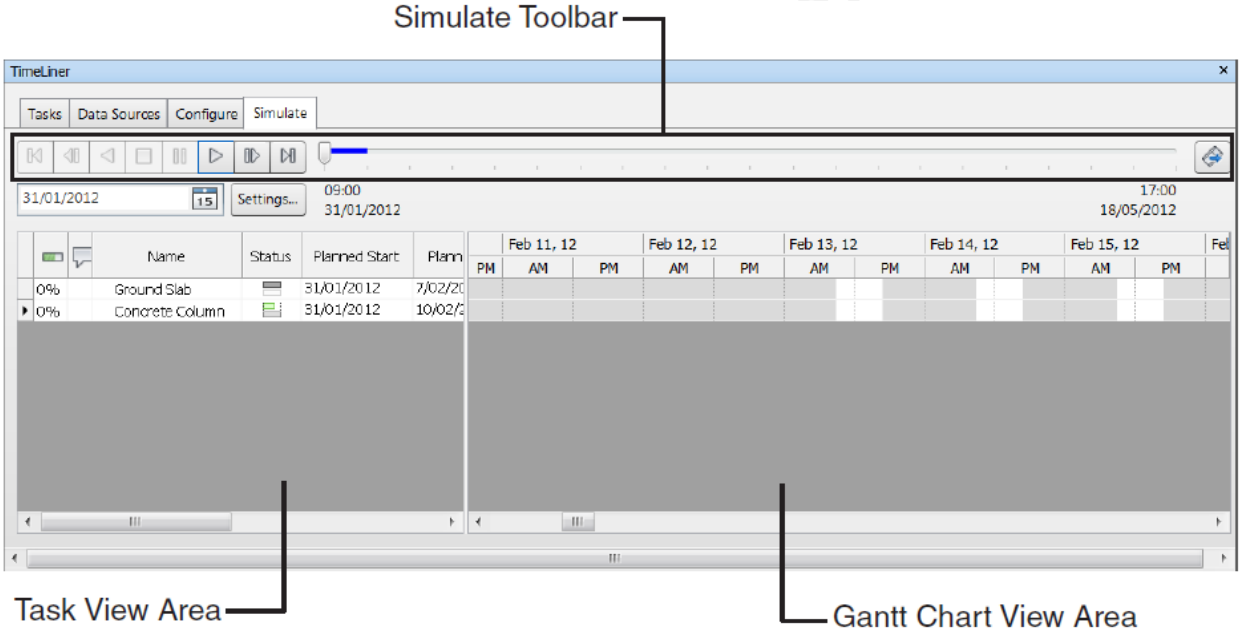
Appearance Definition

Clicking this button, located near the top right of this tab, displays the **Appearance Definitions** dialog box that shows all the existing appearance definitions and also lets you create a new appearance definition.

www.dralavipour.com ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.8 Creating Construction Simulations Using the TimeLiner Module

Simulate Tab



2.8 Creating Construction Simulations Using the TimeLiner Module

Simulate Tab

Simulate Toolbar

This toolbar provides various buttons to playback and export the animations. These buttons are discussed next.

Calendar Box

This box allows you to select a date and the animation will be moved to that date showing you how the construction will look like on that particular date. You can either type the date or click the button and select the date from the calendar.

Settings

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.8 Creating Construction Simulations Using the TimeLiner Module

Simulate Tab

Task View Area

This area shows the tasks that are currently being simulated in the timeliner simulation on the graphics window.

Gantt Chart View Area

This area shows the Gantt chart bar of the tasks that are currently being simulated in the timeliner simulation on the graphics window.

Prepared By Kambiz N... MSc
Do Not Distri...

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

PROCEDURE FOR AUTOMATICALLY ADDING TASKS

The following are the procedures to automatically add various tasks to the timeliner simulation.

Automatically Adding Tasks for Every Topmost Layer

The following steps show you how to add tasks for every topmost layer in the scene.

a. Open the file in which you want to add the timeliner tasks.

b. If the **TimeLiner** window is not already docked, click the **TimeLiner** button from the **Home** ribbon tab > **Tools** ribbon panel.

c. Dock the **TimeLiner** window below the graphics window.

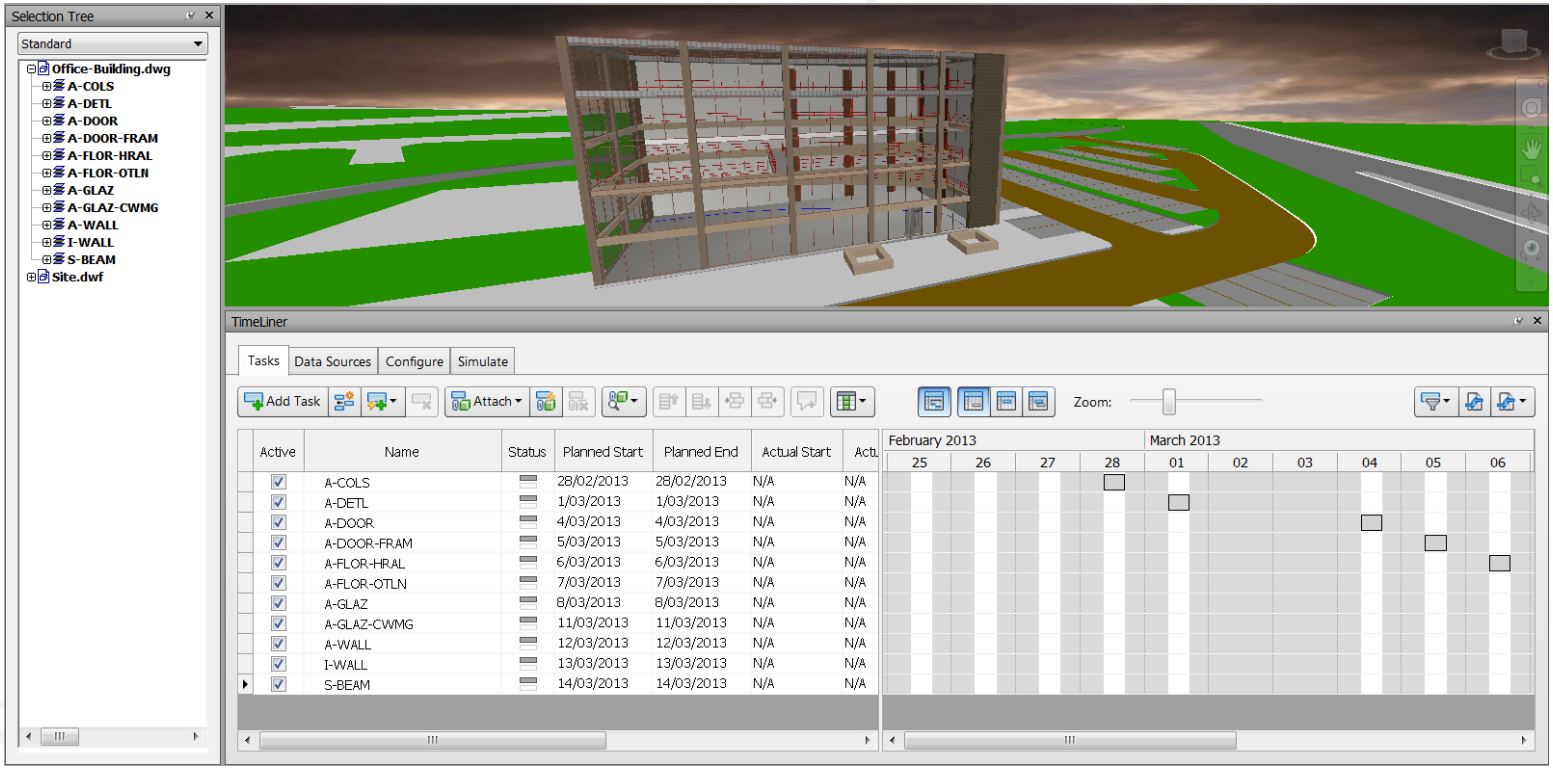
d. From the **Task** toolbar of the **Tasks** tab, click the **Auto-Add Tasks** flyout and click **For Every Topmost Layer**; a task for every topmost layer will be added and the items in those layers will be automatically attached to those tasks.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.8 Creating Construction Simulations Using the TimeLiner Module

e. If required, click on the **Planned Start** and **Planned End** dates for each of the tasks and modify them.

f. Right-click on any task and click **Find > Unattached/Uncontained Items** from the shortcut menu to make sure no object in the model is unattached or uncontained in the tasks.



Automatically Adding Tasks for Every Topmost Item

The following steps show you how to add tasks for every topmost item in the model.

a. Open the file in which you want to add the timeliner tasks.

b. If the **TimeLiner** window is not already docked, click the **TimeLiner** button from the **Home** ribbon tab > **Tools** ribbon panel.

c. Dock the **TimeLiner** window below the graphics window.

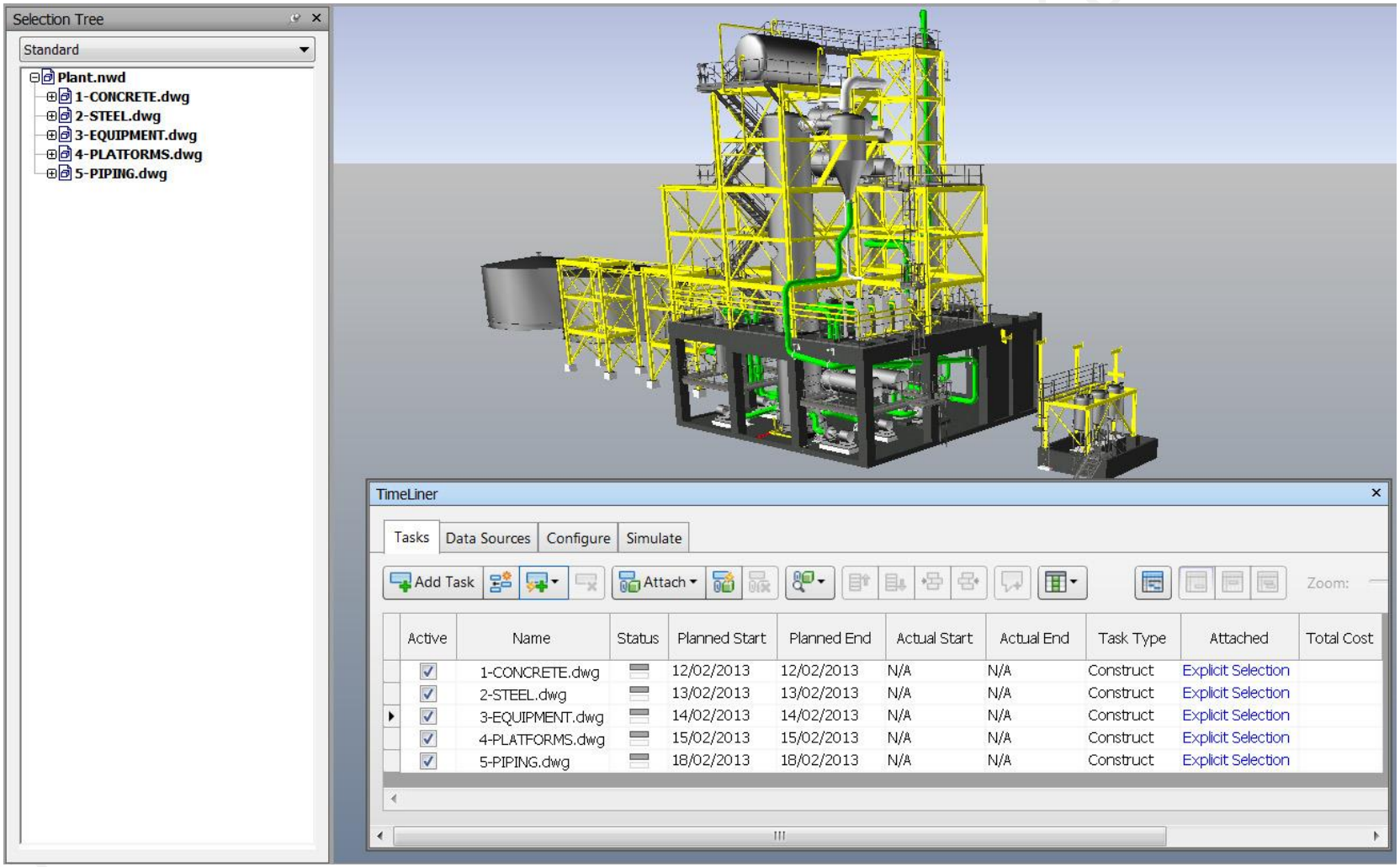
d. From the **Task** toolbar, click the **Auto-Add Tasks** flyout and click **For Every Topmost Item**; a task for every topmost item will be added and those items will be automatically attached

e. If required, click on the **Planned Start** and **Planned End** dates for each of the tasks and modify them.

f. Right-click on any task and from the shortcut menu, click **Find > Unattached/Uncontained Items** to make sure no object in the model is unattached or uncontained in the tasks.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.8 Creating Construction Simulations Using the TimeLiner Module



Automatically Adding Tasks for Every Set

The following steps show you how to add tasks for every set in the model.

a. Open the model that has all the required sets created.

b. From the **Task** toolbar of the **Tasks** tab of the **TimeLiner** window, click the **Auto-Add Tasks** flyout and click **For Every Set**; a task for every set will be added and those sets will be automatically attached to the tasks. Figure 18 shows all the sets in the building model and the **TimeLiner** window showing the corresponding tasks added for all those sets.

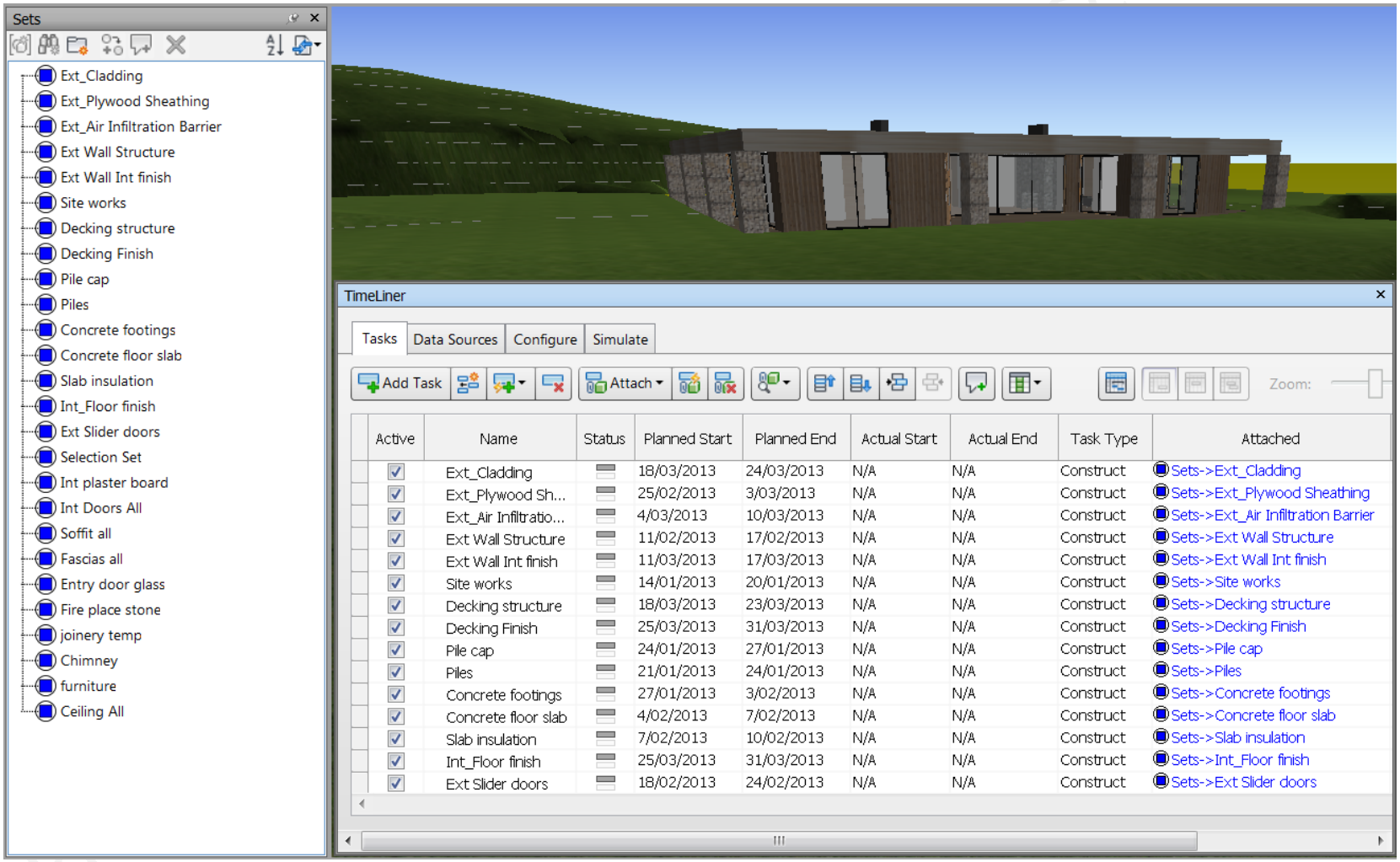
***Note**: Notice the **Attached** column in Figures 16 and 17 show **Explicit Selection**, but in Figure 18 this column shows the name of the set attached to each task. You can click on any of these attachments to highlight the attached objects in the graphics window and the **Selection Tree**,*

c. If required, click on the **Planned Start** and **Planned End** dates for each of the tasks and modify them.

d. Right-click on any task and from the shortcut menu, click **Find > Unattached/Uncontained Items** to make sure no object in the model is unattached or uncontained in the tasks.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.8 Creating Construction Simulations Using the TimeLiner Module



PROCEDURE FOR CREATING SCHEDULING TASKS BY LINKING EXTERNAL PROJECT FILES

Autodesk Navisworks allows you to import the scheduling tasks created in external project files or databases to create construction simulations. Autodesk Navisworks supports the following scheduling software or database to be linked:

Linking CSV Files

The following steps show you how to link a CSV file to the timeliner schedule:

a. On the **Data Sources** tab of the **TimeLiner** window, click the **Add** flyout and then click **CSV Import**; the **Open** dialog box will be displayed.

b. Browse to the CSV file that you want to link and then double-click on the file; the **Field Selector** dialog box will be displayed.

c. If the first row of the CSV file is the header row, click the **Row 1 contains headings** tick box in the **CSV Import Settings** area. Using the other options in this area, you can import the date and time settings from the CSV file or specify your own date and time settings.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

Linking Microsoft Project MPX Files

The following steps show you how to link a Microsoft Project MPX file to the timeliner schedule:

a. On the **Data Sources** tab of the **TimeLiner** window, click the **Add** flyout and then click **Microsoft Project MPX**; the **Open** dialog box will be displayed.

b. Browse to the MPX file that you want to link and then double-click on the file; the **Field Selector** dialog box will be displayed.

c. Click on the rows under the **External Field Name** column and select the fields corresponding to the names listed under **Column**.

d. Click **OK** in the **Field Selector** dialog box; the MPX link is displayed in the **Data Source** tab.

e. In the **Data Source** tab, right-click on the data link and click **Rename** from the shortcut menu to rename the data source to the required name.

f. In the **Data Source** tab, right-click on the data link again and click **Rebuild Task Hierarchy** from the shortcut menu; the task hierarchy is created and displayed in the **Tasks** tab with the timeliner columns mapped to the fields you selected in the **Field Selector** dialog box.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.8 Creating Construction Simulations Using the TimeLiner Module

g. If the task names match the object names, layer names, or sets in the Autodesk Navisworks model, use the **TimeLiner Rules** dialog box to attach items to the tasks.

h. If the task names do not match, then manually select the objects from the **Selection Tree** or from the graphics window.

i. In the **Tasks** tab, right-click on the task to which you want to attach the selected objects and then click **Attach Current Selection** from the shortcut menu.

Prepared By Kambiz Kor
Do Not Distribu

Linking Primavera Project Management 6-8 Schedules

Before linking the Primavera Project Management 6-8 schedules, you need to get your computer ready to connect to it. The following steps show you how to do this:

a. Make sure you have installed Primavera Project Management locally on your computer or the program is remotely accessible.

b. Use the Primavera Project Management install media to install the Primavera Software Development Kit (SDK), which is listed under **Custom** installation. Make sure you expand **Other Components** during the installation process to view **Software Development Kit**.

c. After the SDK installation is complete, you will be prompted to configure a database. Generally, Microsoft SQL Server/MSDE database works well in this case.

Once all this is configured, you are ready to link the Primavera Project Management link.

d. On the **Data Sources** tab of the **TimeLiner** window, click the **Add** flyout and then click **Primavera Project Management 6-8**; the **Primavera DSN Data Source Login** dialog box will be displayed.

e. Enter the username and password in the dialog box and click **OK**; the **Select Primavera Project** dialog box will be displayed.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.8 Creating Construction Simulations Using the TimeLiner Module

f. Select the project from which you want to import the schedule and click **OK** in the dialog box; the **Field Selector** dialog box will be displayed.

g. Click on the rows under the **External Field Name** column and select the fields corresponding to the names listed under **Column**.

h. Click **OK** in the **Field Selector** dialog box; the MPX link is displayed in the **Data Source** tab.

i. In the **Data Source** tab, right-click on the data link and click **Rename** from the shortcut menu to rename the data source to the required name.

j. In the **Data Source** tab, right-click on the data link again and click **Rebuild Task Hierarchy** from the shortcut menu; the task hierarchy is created and displayed in the **Tasks** tab with the timeliner columns mapped to the fields you selected in the **Field Selector** dialog box.

k. If the task names match the object names, layer names, or sets in the Autodesk Navisworks model, use the **TimeLiner Rules** dialog box to attach items to the tasks.

l. If the task names do not match, then manually select the objects from the **Selection Tree** or from the graphics window.

m. In the **Tasks** tab, right-click on the task to which you want to attach the selected objects and then click **Attach Current Selection** from the shortcut menu.

Linking Primavera P6 (Web Services) V6 - V8.2

Before linking the Primavera P6 (Web Services) V6 - V8.2 schedules, you need to get your computer ready to connect to it. The following steps show you how to do this:

a. Make sure you have installed Primavera P6 V6-8.2 Web Services installed within a suitable Web Application Server such as Oracle WebLogic, JBoss, or IBM WebSphere on the same machine as your P6 database.

   Once all this is configured, you are ready to link the Primavera Project Management link.

b. On the **Data Sources** tab of the **TimeLiner** window, click the **Add** flyout and then click the required Primavera P6 Webservice that you are using; the **Please Login** dialog box will be displayed.

c. Enter the username, password, server address, and the server port in the dialog box and click **Login**; the **Primavera P6 Database Instance Selection** dialog box will be displayed.

d. Double-click on the database instance ID to select it; the **Primavera Project Selection** dialog box will be displayed.

e. Select the project from which you want to import the schedule and click **OK** in the dialog box; the **Field Selector** dialog box will be displayed.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.8 Creating Construction Simulations Using the TimeLiner Module

f. Click on the rows under the **External Field Name** column and select the fields corresponding to the names listed under **Column**.

g. Click **OK** in the **Field Selector** dialog box; the MPX link is displayed in the **Data Source** tab.

h. In the **Data Source** tab, right-click on the data link and click **Rename** from the shortcut menu to rename the data source to the required name.

i. In the **Data Source** tab, right-click on the data link again and click **Rebuild Task Hierarchy** from the shortcut menu; the task hierarchy is created and displayed in the **Tasks** tab with the timeliner columns mapped to the fields you selected in the **Field Selector** dialog box.

j. If the task names match the object names, layer names, or sets in the Autodesk Navisworks model, use the **TimeLiner Rules** dialog box to attach items to the tasks.

k. If the task names do not match, then manually select the objects from the **Selection Tree** or from the graphics window.

l. In the **Tasks** tab, right-click on the task to which you want to attach the selected objects and then click **Attach Current Selection** from the shortcut menu.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

PROCEDURE FOR AUTOMATICALLY ATTACHING ITEMS TO THE TASKS USING RULES

If the scheduling tasks are manually created in the **TimeLiner** window or are created using a linked project file, there are no items attached to those tasks by default. In that case, Autodesk Navisworks allows you to use some existing rules or create your own custom rules to attach the items to those tasks. The following steps show you how to do this.

a. Create the timeliner tasks manually or link an external project file.

b. From the **Task** toolbar of the **Tasks** tab of the **TimeLiner** window, click the **Auto-Attach Using Rules** button; the **TimeLiner Rules** dialog box will be displayed, as shown in Figure 19.

c. If the task names match the names of the items in the model also have matching case, click the tick box of the first default rule and then click the **Apply Rules** button at the lower right corner of the **TimeLiner Rules** dialog box.

d. If the tasks names matching the names of the sets in the model also have matching case, click the tick box of the second default rule and then click the **Apply Rules** button at the lower right corner of the **TimeLiner Rules** dialog box.

e. If the tasks names matching the names of the layers in the model also have matching case, click the tick box of the third default rule and then click the **Apply Rules** button at the lower right corner of the **TimeLiner Rules** dialog box.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.8 Creating Construction Simulations Using the TimeLiner Module

f. If you have the task names matching a combination of items, sets, and layers, select the tick boxes of all the three default rules and then click the **Apply Rules** button.

g. To create your own custom rules, click the **New** button in the **TimeLiner Rules** dialog box;

h. Click on the first rule template in this dialog box to create a new rule that attaches the items, layers, or sets to the tasks with the same name, but ignoring the case.

i. Click on the second rule template to create a new rule that attaches the items with the properties matching the specified criteria to any of the specified columns of the tasks.

j. Rename the new rule to any suitable name and then click **OK** in the dialog box.

k. Select the tick box of the new rule and then click the **Apply Rules** button to attach the items matching the rules criteria to the tasks.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

PROCEDURE FOR CUSTOMIZING THE SIMULATION OVERLAY TEXT

As mentioned earlier, while playing the construction simulation, you can have the information about the tasks being simulated displayed as overlay text in the graphics window.

a. Click the **Simulate** tab of the **TimeLiner** window.

b. Click the **Settings** button to display the **Simulation Settings** dialog box.

c. From the **Overlay Text** area, click the **Edit** button; the **Overlay Text** dialog box will be displayed.

d. Click at the start of the first line of text to move the cursor there.

e. Click the **Colors** button and specify the color for the first line of text.

2.8 Creating Construction Simulations Using the TimeLiner Module

f. Click at the end of the first line of text and then press CTRL+ENTER to go to the next line.

g. Click the **Colors** button and specify the color for the second line of text.

h. Click the **Cost** button and select the cost you want to display in the overlay text. Note that if the costs are not added in the timeliner tasks, they will be displayed as 0.00 in the overlay text.

i. Click the **Font** button and select the font for the overlay text.

j. Click **OK** in the **Overlay Text** dialog box and then click **OK** in the **Simulation Settings** dialog box; the modified overlay text will be displayed in the graphics window.

Prepared By K... MSc
Do Not...

PROCEDURE TO LINK A VIEWPOINT ANIMATION OR AN ANIMATOR CAMERA ANIMATION TO THE SIMULATION

Autodesk Navisworks allows you to link a saved viewpoint animation or an animation camera animation to the timeliner schedule such that it plays throughout the timeliner simulation. The following steps show you how to do this:

a. Click the **Simulate** tab of the **TimeLiner** window.

b. Click the **Settings** button to display the **Simulation Settings** dialog box.

c. From the **Animation** area, click on the list and select the animator camera animation or the saved viewpoint animation. This animation will play throughout the timeliner simulation.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

PROCEDURE FOR ADDING AN ANIMATOR CAMERA ANIMATION AS A TASK

The **TimeLiner** tool allows you to link the animator camera animation into the timeliner schedule as a task. This means that if the camera animation represents orbiting around the model, that animation will be simulated as a task in the timeliner simulation. The following steps show how to do this.

a. In the **Tasks** tab of the **TimeLiner** window, use the **Add Task** to add the task at the bottom of the list or use **Insert Task** tool to add a task above the selected task.

b. Rename the task to **Animation**.

c. From the **Task** toolbar, click the **Column Chooser** flyout and then click **Choose Columns**; the **Choose TimeLiner Columns** dialog box will be displayed.

d. From the dialog box, select the tick boxes for the **Animation** and **Animation Behaviour** columns.

e. Click **OK** in the dialog box; the two columns are added to the task view area.

f. Specify the start and end date for the **Animation** task.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.8 Creating Construction Simulations Using the TimeLiner Module

***Tip**: If you want the animator camera animation to run throughout the timeliner simulation, specify the start date of the animation timeliner task as the start date of the simulation and the end date of the task as the end date of the simulation.*

g. In the task view area, scroll to the right to see the **Animation**, **Animation Behaviour**, and **Script** columns.

h. Click on the field in the **Animation** column and select the animator animation.

i. From the **Animation Behaviour** column, make sure **Scale** is selected.

j. From the **Script** field, select the script that plays the animation.

Prepared By K
Do Not

PROCEDURE FOR LINKING A VIEWPOINT ANIMATION OR AN ANIMATOR CAMERA ANIMATION TO THE SIMULATION

Autodesk Navisworks allows you to link a saved viewpoint animation or an animation camera animation to the timeliner schedule such that it plays throughout the timeliner simulation. The following steps show you how to do this:

a. Click the **Simulate** tab of the **TimeLiner** window.

b. Click the **Settings** button to display the **Simulation Settings** dialog box.

c. From the **Animation** area, click on the list and select the animator camera animation or the saved viewpoint animation. This animation will play throughout the timeliner simulation.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

۲. مدلسازی اطلاعات ساخت با استفاده از نرم افزار

Navisworks

Kambiz Kordani PMP,MSc
Linkedin: @kordani-kambiz
Instagram: @acemi_social

2.9 Quantification Module

- Introduce you to the Quantification module.
- Familiarize you with the Quantification Workbook window.
- Explain the item and resource catalogs.
- Explain how to add or edit formulas in the catalogs.
- Teach you how to perform quantity takeoffs on the objects in the scene.
- Teach you how to perform quantity takeoffs on virtual objects that are not present in the scene.
- Teach you how to export the quantity takeoff data in various formats.

Kambiz Kordani PMP, MSc
Do Not Distribute

www.dralavipour.com ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

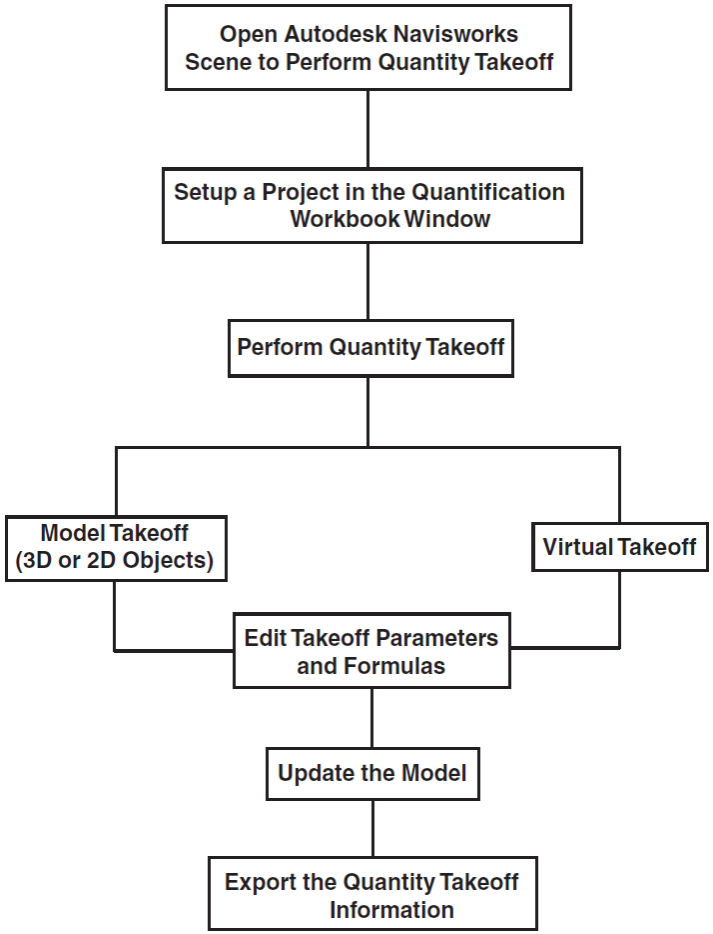
2.9 Quantification Module

THE QUANTIFICATION MODULE

The **Quantification** module is used to create a cost estimation of the project by performing quantity takeoff from 3D and 2D objects in the Autodesk Navisworks scene. You can also perform quantity takeoff from virtual objects that are not present in the scene but will be a part of the project. It is important to note that the 2D quantification is only supported with native or nonnative DWF files. Any other format 2D files will first have to be published as DWF to be used for 2D quantification. The quantity takeoff can be exported in the format such as Microsoft Excel that can be read by the project members who do not have Autodesk Navisworks.

Prepared By Kambiz
Do Not Distribute
MSc

2.9 Quantification Module



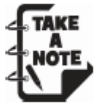
Prepared By: K.Kordani PMP, MSc

2.9 Quantification Module

TYPES OF QUANTITY TAKEOFFS

As shown in the flowchart on the previous page, Autodesk Navisworks allows you to perform model quantity takeoffs using 3D model objects or 2D DWF sheets, or using virtual objects. All these types of takeoffs are discussed next.

Model Quantity Takeoff

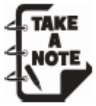
This type of takeoff is used for an Autodesk Navisworks scene created using the supported native CAD files that brings the model properties into Autodesk Navisworks. The supported CAD files include AutoCAD, AutoCAD Architecture, AutoCAD MEP, Autodesk Civil 3D, AutoCAD Map 3D, Autodesk Inventor, and Autodesk Revit. Alternatively, you can use the DWF files to perform model quantity takeoffs.

***Note**: Remember that you may need the object enablers to be installed to read the object properties for some of these native CAD files. You can use the **Scene Statistics** dialog box to find if you need any object enabler to be installed. Refer to Chapter 1 of this textbook to know more about this.*

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.9 Quantification Module

2D Quantity Takeoff

This is a new feature introduced in Autodesk Navisworks 2015 and is used to support the 2D and 3D integrated workflow from software such as Autodesk Revit. Remember that the 2D takeoff is only supported in DWF drawing sheets. Therefore, to work with the integrated workflow, you will have to export the Autodesk Revit files into DWF format. You can also publish 2D DWF sheets from other native Autodesk software to perform 2D takeoff in Autodesk Navisworks. However, if you want to use other file formats, such as a PDF file, you need to first convert it into a DWF file using the print driver software available from the Autodesk website.

While working with the integrated 2D and 3D workflow in a DWF file published from Autodesk Revit, you will need to use the **Project Browser** or the multisheet buttons on the status bar below the graphics window to switch between various sheets.

***Note**: The snap to vertex options while drawing the 2D takeoff markups are only available in the DWF files published from the native CAD software.*

2.9 Quantification Module

Virtual Quantity Takeoff

This type of takeoff is used for objects that do not exist in the model. You can also use this type of takeoff for the objects that do not have any native property defined or the native properties were lost while exporting the files from the native CAD program. The virtual takeoff allows you to create a viewpoint where you want to perform a takeoff. You can then use the measure tools in that viewpoint to measure objects with no properties or add redline markups. For the objects that do not exist, you can edit the formulas to get the required quantity.

Prepared By Kambiz Kordani
Do Not Distribute

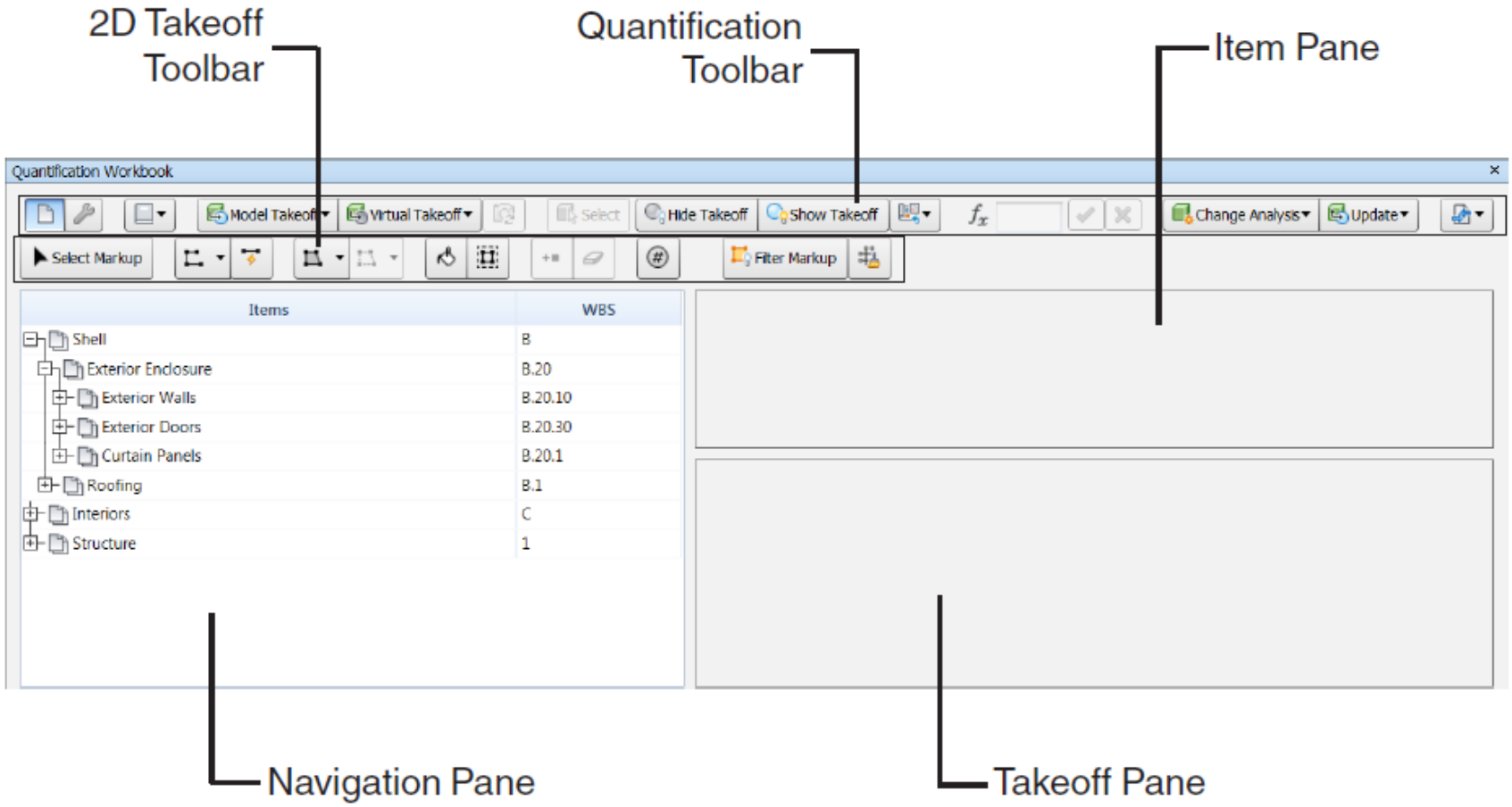
ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

QUANTIFICATION WORKBOOK

Home Ribbon Tab > Tools Ribbon Panel > Quantification

When you invoke the **Quantification** tool for the first time to perform the quantity takeoff, the **Quantification Workbook** window will be displayed. The first thing you need to do in this window is to setup the project. This involves defining the catalog that you will use for the quantity takeoff and the units that you want to work with. A catalog contains the categories of items in which you will organize the model objects to takeoff the quantity. Because you need to first setup the project, the **Quantification Workbook** window will be blank when you open it and only the **Project Setup** button will be available on the top left corner of this window. When you click this button, the **Quantification Setup Wizard** dialog box will be displayed showing the **Select Catalog** page,

2.9 Quantification Module



ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.9 Quantification Module

Quantification Toolbar

The **Quantification** toolbar located at the top in the **Quantification Workbook** window shows the following buttons:

Switch to Item View/ Switch to Resource View

These are the first two buttons on the **Quantification** toolbar and are used to switch the view to either the item catalog view or resource catalog view.

Show and hide Item and Resource Catalogs

This flyout is used to display the **Item Catalog** and **Resource Catalog** windows.

Model Takeoff

This flyout is used to perform a model takeoff to the catalog item selected in the left pane of the **Quantification Workbook** window or a new catalog item.

Virtual Takeoff

This flyout is used to perform a virtual takeoff of the nonexisting objects to the catalog item selected in the left pane of the **Quantification Workbook** window or a new catalog item.

2.9 Quantification Module

Add a viewpoint to the takeoff, or update an existing viewpoint

This button is only available during virtual takeoff. It allows you to create a viewpoint for the virtual takeoff so that you can perform measurements of the objects using that viewpoint. This button can also be used to update the viewpoint that you have previously created for the virtual takeoff.

Select

This button is used to select the objects in the graphics window that correspond to the takeoffs selected in the **Quantification Workbook** window. Alternatively, you can click on the name of the objects in this window to select them in the graphics window.

Hide Takeoff

This button is used to toggle on or off the visibility of the objects corresponding to the takeoff selected in the **Quantification Workbook** window.

Show Takeoff

This button is used to toggle on or off the visibility of all the objects other than the objects corresponding to the takeoff selected in the **Quantification Workbook** window.

Control the appearance of model items

This button is used to specify whether the model objects should be displayed with the quantification appearance or the original model appearance.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.9 Quantification Module

***Note**: If the model objects have been assigned materials and the render mode is set to **Full Render** in the **Viewpoint** ribbon tab > **Render Style** ribbon panel, the quantification appearance will not be displayed on the model.*

fx Function

This field displays the information of the field selected for the takeoff object.

Change Analysis

This flyout is used to analyze the model for any changes in the geometry. This is generally used when you perform model take off from a native file and model is update in the native file.

Update

This flyout is used to update the takeoff from the changed model. This flyout can also be used to delete the selected takeoff.

Import/Export Catalogs and export Quantities

This flyout is used to import or export the catalogs. You can also use this flyout to export all or selected takeoffs into a Microsoft Excel file.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.9 Quantification Module

2D Takeoff Toolbar

This toolbar is located below the **Quantification** toolbar. The tools in this toolbar are used to perform 2D takeoffs from a DWF file. These tools are discussed next.

Select Markup

This button is used to select an existing markup.

Polyline Flyout

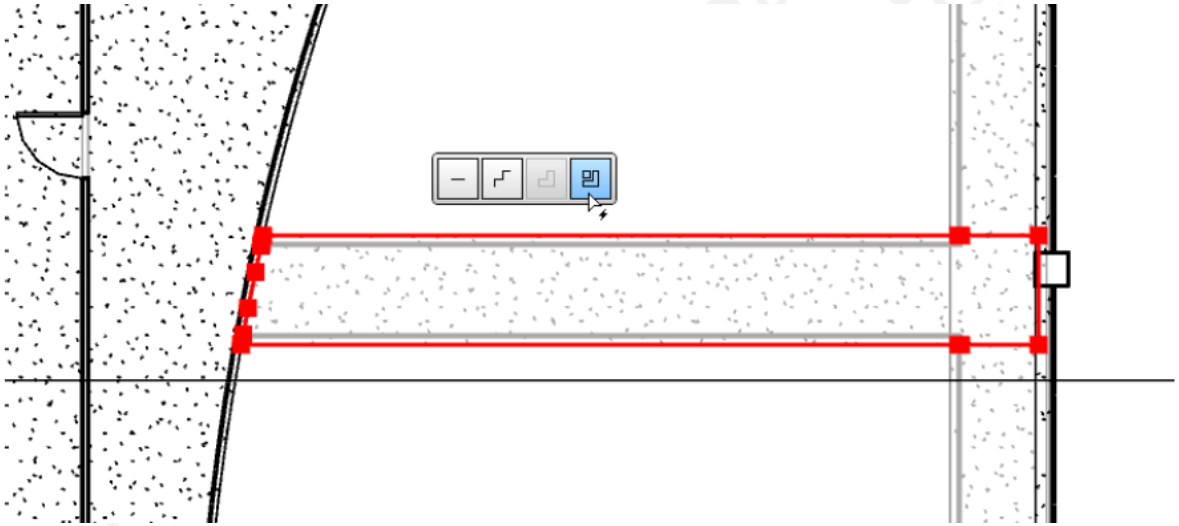
This flyout provides the **Polyline** and **Rectangle Polyline** tools to draw a markup by tracing over the objects in the 2D DWF sheets. These markups can be used for length or perimeter takeoffs. If you are using a native 2D DWF sheet, you will be able to snap to the endpoints of the objects while drawing the markups. To create linear markup, use the **Polyline** tool and click on the points to draw the polyline. After specifying all the line segments, right-click to end this tool and accept the takeoff. To create perimeter markup, use the **Rectangle Polyline** tool and specify the two opposite corners.

***Tip**: While drawing the polylines, you can hold down the SHIFT key to draw horizontal and vertical lines.*

2.9 Quantification Module

Quick Line

This tool is used to takeoff the lengths and perimeters of the existing objects in the native DWF files by directly selecting them on the sheet. When you select an object, the **Quick Line** toolbar is displayed, as shown in Figure 6. This toolbar allows you to specify whether you want to only select the object on which you clicked, all the connected lines with it, or the entire loop.



www.dralavipour.com ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.9 Quantification Module

Area Flyout

This flyout provides the **Area** and **Rectangle Area** tools to create a markup for the area takeoffs by tracing over the objects in the 2D DWF sheets. If you are using a native 2D DWF sheet, you will be able to snap to the endpoints of the objects. To create the area takeoff using the **Area** tool, pick all the required points and then right-click to accept the area. To create the area takeoff using the **Rectangle Area** tool, specify the two opposite corners.

***Tip**: At any point, you can select a markup using the **Select Markup** tool and press the DELETE key on the keyboard to delete the markup and the takeoff values associated with it.*

Backout Flyout

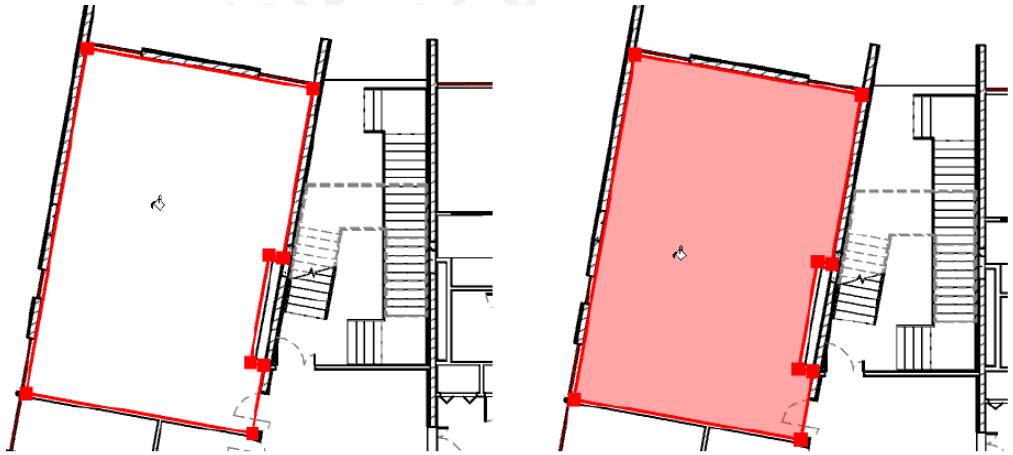
This flyout is used to remove certain region from the existing area takeoff and will only be available when you select an existing area takeoff. You can specify the region to remove using the **Backout** tool that allows you the specify the endpoints of the region you want to remove. Alternatively, you can use the **Rectangle Backout** tool and specify the two opposite corners of the region to remove.

www.dralavipour.com ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.9 Quantification Module

Bucket Fill

This is a smart tool that lets you bucket fill a region and then takeoff the measurement as a length or area. When you invoke this tool, the cursor will change to the bucket cursor. Now, click inside a closed region to highlight the vertices of that region. At this stage if you right-click, the length and perimeter takeoff values will be shown. However, if you want to change this to the area markup, click inside the same closed region again and then right-click to accept. The region will be color filled and the area and perimeter takeoff values will be shown. Figure 7 shows a region selected using the **Bucket Fill** tool to create linear markup and Figure 8 shows the same region changed to the area markup by clicking inside the region again.



ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.9 Quantification Module

***Tip**: You can toggle between the linear and area markup by repeatedly clicking in the same region. But as you as you right-click, the markup will be accepted and cannot be changed later.*

Quick Box

This tool is similar to the **Quick Line** tool and is used draw an area markup by dragging a box around the region. After dragging the box, right-click to accept the area markup.

Add Vertex

This tool is used to add a new vertex to the existing markup. Once you have added a vertex, you can then drag it to any other location in the sheet.

Delete Vertex

This tool is used to delete a vertex from the selected markup. You can click on any vertex and delete it from the existing markup.

***Note**: Remember that depending on the vertex you are deleting, you may lose an entire segment from the markup. In this case, you will have to add a new vertex and the drag and drop it on the location from where you deleted the vertex.*

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.9 Quantification Module

Count

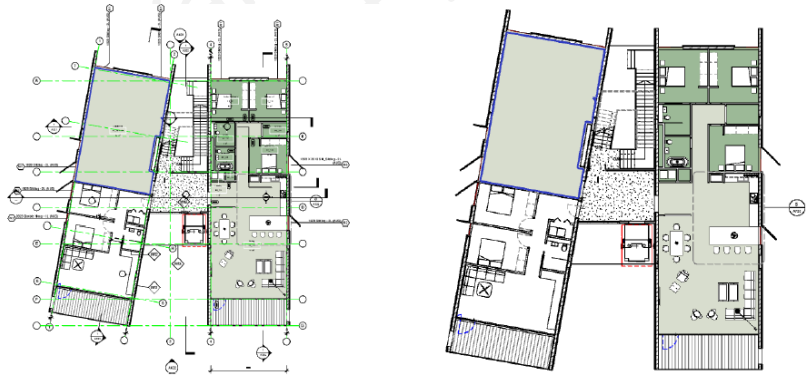
This tool is used to create a count takeoff by placing a pin on the object. This type of takeoff is generally used to count things such as number of tables, chairs, workstations, and so on.

Filter Markup

This tool is used to select an item or group of items from the **Navigation** pane of the **Quantification Workbook** and only show their associated takeoff geometries. All the rest of takeoff items will be turned off from the current display.

Hide background and annotations

This tool is used to turn off the visibility of the background and annotation objects from the drawing sheet. Figure 9 shows a 2D sheet with the background and annotation objects turned on



ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.9 Quantification Module

Navigation Pane

The navigation pane displays the groups and items of the catalog selected for the project. The group icons are displayed with the double file icon and the item icons are displayed as a single file icon.

Item Pane

The item pane displays the information about the group or item selected from the navigation pane. If you select a group or an item that has objects with quantities taken off, this pane will display the sum quantities of all the objects in the takeoff.

Takeoff Pane

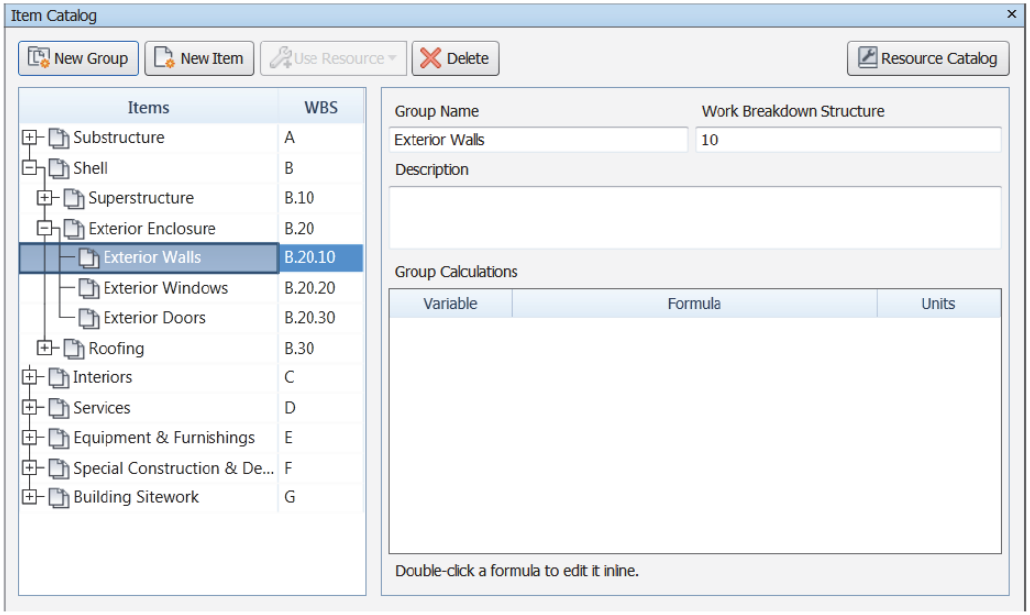
The takeoff pane displays the details of the individual quantities of the objects that were used for the quantity takeoff. If you select an item or a group that does not have any object with the quantity taken off, this pane will be blank.

Prepared By K
Do No

2.9 Quantification Module

Item Catalog

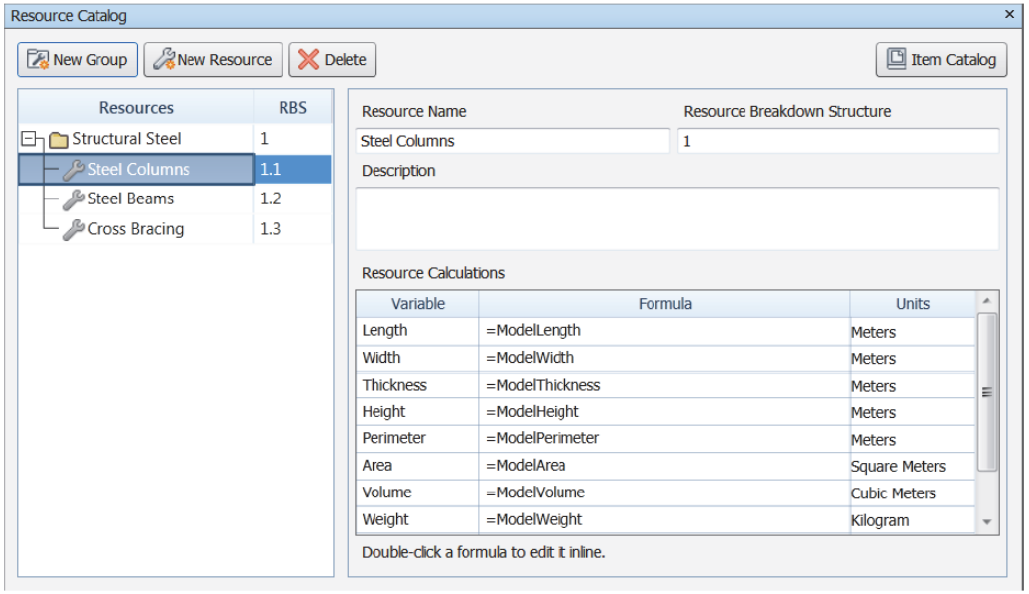
The **Item Catalog** is a database of categories and groups to associate the objects in the Autodesk Navisworks scene to perform quantity takeoff. Figure 6 shows a default **Item Catalog** window. The left pane of this window shows various categories and groups of items and the Work Breakdown Structure (WBS) of those items. The right pane shows the group information and the formulas for the calculation of these groups.



2.9 Quantification Module

Resource Catalog

The **Resource Catalog** is a database of resources that you need to perform quantity takeoff against in the current Autodesk Navisworks scene. By default, this catalog is blank. You can add groups and resources to this catalog. Figure 7 shows a **Resource Catalog** window in which a group called **Structural Steel** is created. Inside this group, three resources are created for the steel columns, beams, and bracings. The right pane of this window can be used to create or edit formulas for each of these resources.



TAKEOFF PROPERTY MAPPING

While performing model takeoffs from most native files, the properties embedded in those files are automatically mapped to the items in the selected catalog. However, in some cases, such as AutoCAD Plant 3D files, the properties are not mapped automatically. Therefore, to perform quantity takeoff from such files or from the files in which you want to change this mapping or select new property mapping, you can use the options available in the **Item Catalog** window.

You have an option to map the properties for individual catalog items or globally for all items in the catalog. The new property mapping will be valid for both 2D and 3D takeoffs. The following section explains the procedures of how to do this.

Prepared By Ka... Do Not ...

PROCEDURES REQUIRED WHILE WORKING WITH THE QUANTIFICATION MODULE

The following are the various procedures that are required while working with the **Quantification** module.

Procedure for Mapping Properties of Individual Catalog Items

The following is the procedure for mapping properties of individual items:

a. Open the **Item Catalog** window. From the left pane, select the item whose mapping properties you need to change.

b. From the right pane, select the **Item Map Rules** tab.

c. Double-click in the **Category** field of the takeoff property you want to edit and select the required category from the list.

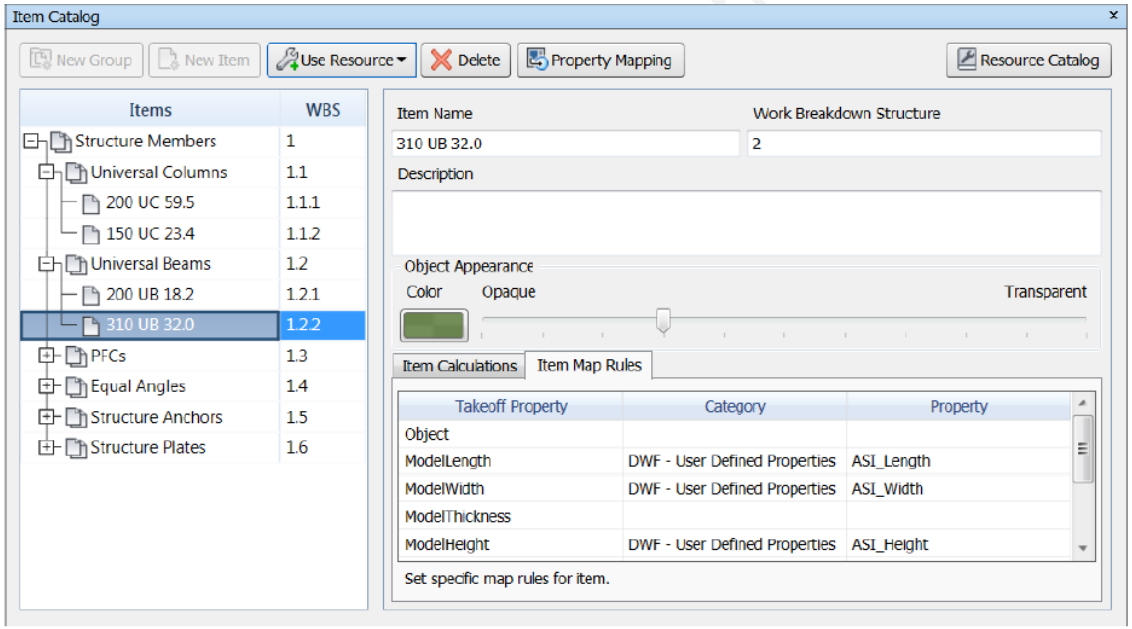
d. Double-click on the **Property** field to display the list of available properties for the selected category. From the list, select the property that you want to map to.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.9 Quantification Module

e. Repeat the same process for all the takeoff properties that you want to edit. The takeoffs performed henceforth will use the new mapping.

Figure 13 shows the **Item Catalog** window with the takeoff properties modified for the 310 UB 32.0 item.



www.dralavipour.com ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.9 Quantification Module

Procedure for Globally Mapping Takeoff Properties

The following is the procedure for globally mapping takeoff properties:

a. Open the **Item Catalog** window.

b. From the toolbar at the top, click **Property Mapping**; the **Property Mapping** dialog box will be displayed.

c. Click the **Add Mapping Rule** (+) button from the right of this dialog box; a new takeoff property row will be added.

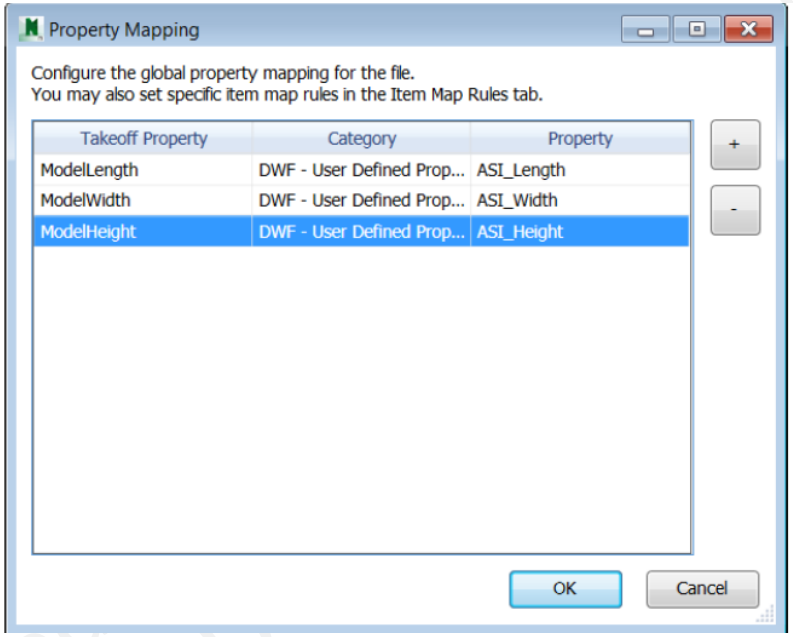
d. Double-click on **Object** in the row recently added and select the takeoff property that you want to edit.

e. Double-click in the **Category** field and select the required category from the list.

f. Double-click on the **Property** field to display the list of available properties for the selected category. From the list, select the property that you want to map to.

2.9 Quantification Module

g. Repeat the same process for all the takeoff properties that you want to edit. Figure 14 shows the **Property Mapping** dialog box with the modified takeoff properties.



h. Click **OK** in the dialog box. The takeoffs performed henceforth will use the new mapping.

Procedure for Adding Groups and Items to the Item Catalog

The following is the procedure for adding groups and items to the **Item Catalog**.

Adding a new Group to the Item Catalog

The following steps show you how to add a group to the item catalog.

a. Click on **Item Catalog** from the left of the **Selection Tree** and then dock this window.

b. In the left pane, click on the category under which you want to add the new group. If it is a blank catalog, skip to step c.

c. From the toolbar at the top in the **Item Catalog** window, click the **New Group** button; a new group is added in the window.

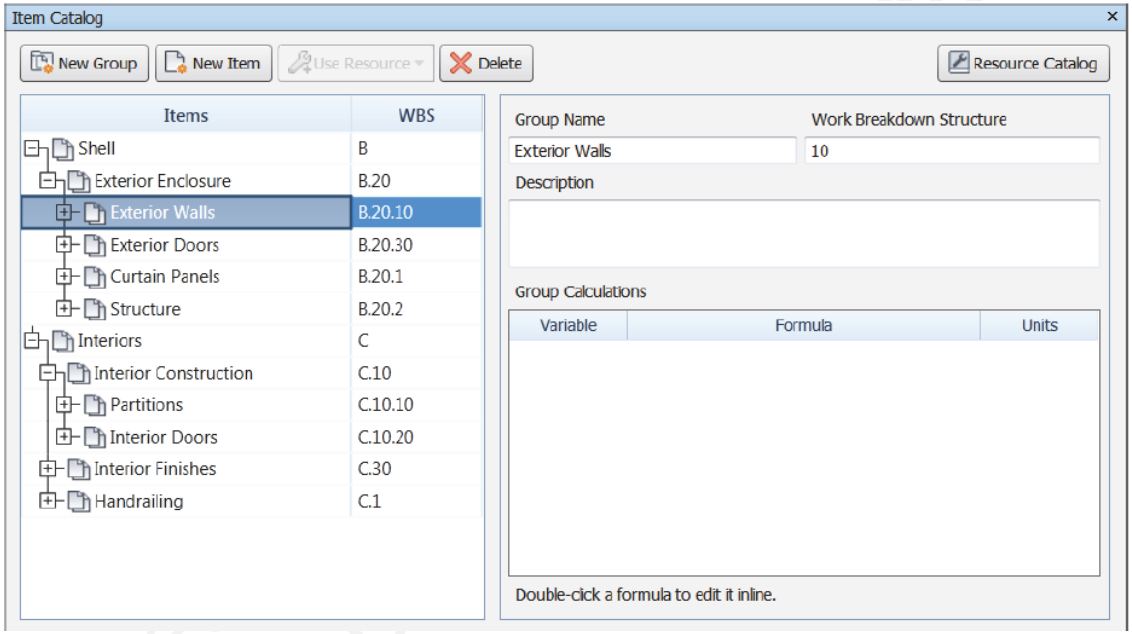
d. Rename the group to the required name.

Prepared By K.Kordani PMP, MSc Do Not

2.9 Quantification Module

e. In the right pane, specify the Work Breakdown Structure (WBS) for the group and add the description of the group.

Figure 15 shows the **Item Catalog** window with some groups and subgroups created.



***Tip**: The group icon is displayed as a double file icon in the left pane of the **Item Catalog** window.*

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.9 Quantification Module

Adding a new Item to the Item Catalog

The following steps show you how to add an item to the item catalog.

a. Click on **Item Catalog** from the left of the **Selection Tree** and then dock this window.

b. In the left pane, click on the group category under which you want to add the new item.

c. From the toolbar at the top in the **Item Catalog** window, click the **New Item** button; a new item is added in the window.

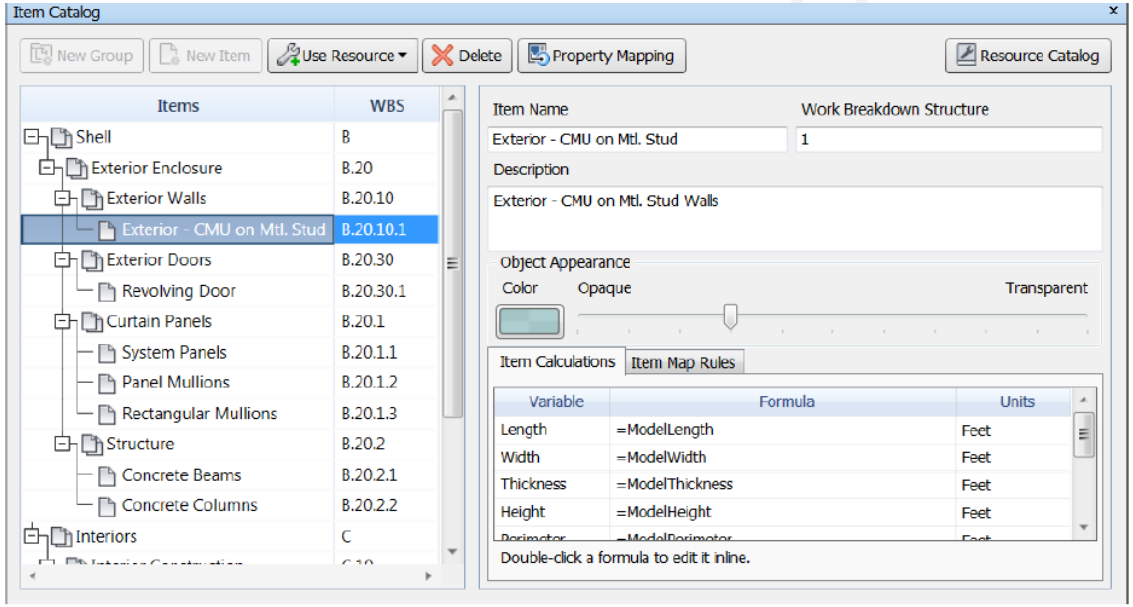
d. Rename the group to the required name.

e. In the right pane, specify the Work Breakdown Structure (WBS) for the group and add description of the group.

f. In the **Object Appearance** area of the right pane, specify the color and transparency for the takeoff objects that will be under the item you created.

2.9 Quantification Module

g. In the **Item Calculations** area, specify the formulas and calculations for the item that you have created.



***Tip**: The item icon is displayed as a single file icon in the left pane of the **Item Catalog** window.*

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

Procedure for Exporting Catalogs

If there are some items and categories that you often use during quantity takeoff, you can export the catalog with those items and categories and then use them in your new project or import them in the existing project. The following is the procedure to do this:

a. In the **Item Catalog** window, create the required group and items.

b. Return back to the **Quantification Workbook** window.

c. From the top right corner of the **Quantification Workbook** window, click the **Import/Export catalogs and export Quantities** flyout > **Export Catalog to XML**; the **Export Catalog to XML** dialog box will be displayed.

d. Browse and select the folder in which you want to save the catalog file.

e. Specify the name of the catalog and save the file.

Prepared By

2.9 Quantification Module

Procedure for Selecting a User-defined Catalog in a New Project

The following steps show you how to select a user-defined catalog in a new project.

a. Open the Autodesk Navisworks scene in which you want to perform quantity takeoff using the user-defined catalog.

b. From the **Quantification Workbook** window, click the **Project Setup** button; the **Quantification Setup Wizard** dialog box is displayed with the **Select Catalog** page active.

c. Select the **Browse to a catalog** radio button at the bottom of the dialog box; the **Browse** button is activated.

d. Click the **Browse** button; the **Open** dialog box is displayed.

e. Browse and select the XML file of the user-defined catalog that you want to use. Click **Open** in the dialog box to return to the **Quantification Setup Wizard** dialog box.

f. Click the **Next** button; the **Ready to Create Database** page is displayed.

g. Click **Finish**; the user-defined catalog is displayed in the **Quantification Workbook** window.

2.9 Quantification Module

Procedure for Importing a User-defined Catalog in an Existing Project

The following steps show you how to import a user-defined catalog in an existing project.

a. From the top right corner of the **Quantification Workbook** window of the existing project, click the **Import/export Catalogs and export Quantities** flyout > **Import Catalog**; the **Import Catalog** dialog box will be displayed.

b. Browse and select the user-defined catalog that you want to import. Click **Open**.

c. If the catalog you are importing has items with the same name as those in the current project, the **Import Catalog** dialog box will be displayed informing you about the duplicate names. You can click the **Replace** or **Keep** button, depending on whether you want to replace the existing items or not.

Prepared By Ka
Do Not

2.9 Quantification Module

Procedure for Performing Model Takeoffs

The following steps show you how to perform model quantity takeoffs.

a. Open the Autodesk Navisworks scene in which you want to perform the model quantity takeoff.

b. Dock the **Quantification Workbook** window.

c. From the left pane of the **Quantification Workbook** window, select the catalog item in which you want to place the takeoffs.

d. Using any selection method, select the model objects that you want to takeoff the quantity from.

e. From the toolbar at the top in the **Quantification Workbook** window, click **Model Takeoff > Takeoff to Selected Catalog Item**; the lower half of the right pane of the **Quantification Workbook** will show all the objects included in the takeoff and their respective values. The upper half of the right pane will show the item category and the sum of all values.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.9 Quantification Module

Procedure for Performing 2D Takeoffs

The following steps show you how to perform model quantity takeoffs.

a. Open the Autodesk Navisworks scene in which you want to perform the model quantity takeoff.

b. Using the **Project Browser** or the multisheet buttons on the status bar, go to the 2D sheet on which you want to perform the 2D takeoffs.

c. Dock the **Quantification Workbook** window.

d. From the left pane of the **Quantification Workbook** window, select the catalog item in which you want to place the takeoffs.

e. From the **2D Takeoff** toolbar, select the tool to create the linear or area takeoff markups. Alternatively, select the **Count** button if you are creating the count takeoffs.

f. Draw the required 2D markup. If need be, right-click to accept the markup; the right pane of the **Quantification Workbook** will show the length and perimeter or the area and perimeter values, depending on whether it was a length takeoff or the area takeoff.

2.9 Quantification Module

Procedure to Perform Virtual Takeoffs

As mentioned earlier, virtual takeoffs can be performed on objects that have geometry but do not have any native properties. It can also be performed on the objects that do not have any geometry. The following are the procedures to perform virtual quantity takeoffs on both these types of objects.

Virtual Quantity Takeoff on the Objects with no Properties

The following steps show you how to perform virtual takeoffs on the objects with no properties.

a. Open the Autodesk Navisworks scene in which you want to perform the virtual quantity takeoff.

b. Dock the **Quantification Workbook** window.

c. From the left pane of the **Quantification Workbook** window, select the catalog item in which you want to place the takeoffs.

d. Zoom to the area that shows the objects that you want to include in the virtual takeoff.

e. Select the objects to be included in the virtual takeoff.

2.9 Quantification Module

f. From the toolbar at the top in the **Quantification Workbook** window, click **Virtual Takeoff > Takeoff to Selected Catalog Item**; a new row is added in the takeoff pane of the **Quantification Workbook** window and a new viewpoint is added.

g. Use the measure tools to measure the objects in the graphics window.

h. Enter the measurements in the takeoff pane of the **Quantification Workbook** window to get the individual quantities.

i. Edit the formulas in the item pane to get the total quantities.

Prepared By Kambiz Kordani, MSc
Do Not Distribute

2.9 Quantification Module

Virtual Quantity Takeoff for Objects with No Geometry

The following steps show you how to perform virtual takeoffs on the objects with no geometry.

a. Open the Autodesk Navisworks scene in which you want to perform the virtual quantity takeoff.

b. Dock the **Quantification Workbook** window.

c. From the left pane of the **Quantification Workbook** window, select the catalog item in which you want to place the takeoffs.

d. Zoom to the area where you want to perform the virtual takeoff.

e. From the toolbar at the top in the **Quantification Workbook** window, click **Virtual Takeoff > Takeoff to Selected Catalog Item**; a new row is added in the takeoff pane of the **Quantification Workbook** window and a new viewpoint is added.

f. Enter the measurements of the virtual objects in the takeoff pane of the **Quantification Workbook** window to get the individual quantities.

g. Edit the formulas in the item pane to get the total quantities.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.9 Quantification Module

Procedure for Exporting the Quantities to Microsoft Excel

The following procedures show you how to export all the quantity takeoffs or selected quantity takeoffs to a Microsoft Excel spreadsheet.

Exporting all Quantities to Microsoft Excel

The following steps show you how to export all the quantities to a Microsoft Excel file.

a. From the top right corner of the **Quantification Workbook** window, click **Import/export catalogs and export Quantities > Export Quantities to Excel**; the **Export Quantities to Excel** dialog box will be displayed.

b. Browse and select the folder where you want to save the Microsoft Excel file.

c. Specify the name of the file and click **Save** in the dialog box.

Prepared By Ka... Do Not

2.9 Quantification Module

Exporting Selected Quantities to Microsoft Excel

The following steps show you how to export the selected quantities to a Microsoft Excel file.

a. In the navigation pane of the **Quantification Workbook** window, hold down the CTRL key or the SHIFT key and select the quantities that you want to export.

b. From the top right corner of the **Quantification Workbook** window, click **Import/export catalogs and export Quantities > Export Selected Quantities to Excel**; the **Export Quantities to Excel** dialog box will be displayed.

c. Browse and select the folder where you want to save the Microsoft Excel file.

d. Specify the name of the file and click **Save** in the dialog box.

Prepared By Kan... Do Not L...

GETTING AUTODESK NAVISWORKS READY TO PERFORM MODEL TAKEOFF FROM the AutoCAD PLANT 3D FILES

As mentioned earlier, while performing model takeoffs, some of the properties embedded inside the AutoCAD Plant 3D files are not automatically mapped to the items in the catalog. Therefore, while working with these files, you will first have to use the **Item Catalog** to map the required properties. The procedure for this was shown in the previous section.

GUID PROPERTY FOR QUANTITY TAKEOFF

It is important to mention here that only the objects that have GUID property listed in the **Item** tab of the **Properties** window can be selected in an AutoCAD Plant 3D file for the quantity takeoff.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

۲. مدلسازی اطلاعات ساخت با استفاده از نرم افزار

Navisworks

Kambiz Kordani PMP,MSc
Linkedin: @kordani-kambiz
Instagram: @acemi_social

2.10 Working with the Clash Detective Module

- Introduce you to the Clash Detective module of Autodesk Navisworks.
- Familiarize you with the Clash Detective window.
- Explain various types of clashes that can be performed.
- Teach you how to view various clash results.
- Teach you how to fix clashes using the Switchback option.
- Teach you to create a clash report.
- Teach you how to use rules to ignore certain objects during clash detection.
- Teach you how to perform a clash test with point cloud data.

Kambiz Kordani PMP, MSc
Do Not Distribute

www.dralavipour.com ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.10 Working with the Clash Detective Module

THE CLASH DETECTIVE MODULE

The **Clash Detective** module available in Autodesk Navisworks Manage is used to perform clashes between various items in the Autodesk Navisworks scene. These clashes can then be reviewed and reported to the concerned stakeholders. This is a very handy tool for the project managers that allows them to avoid any unexpected problems during the construction phase. For example, the project manager can perform clashes between the structural model and the elevator shafts to make sure the shafts are not cutting through any beams or columns in the structure.

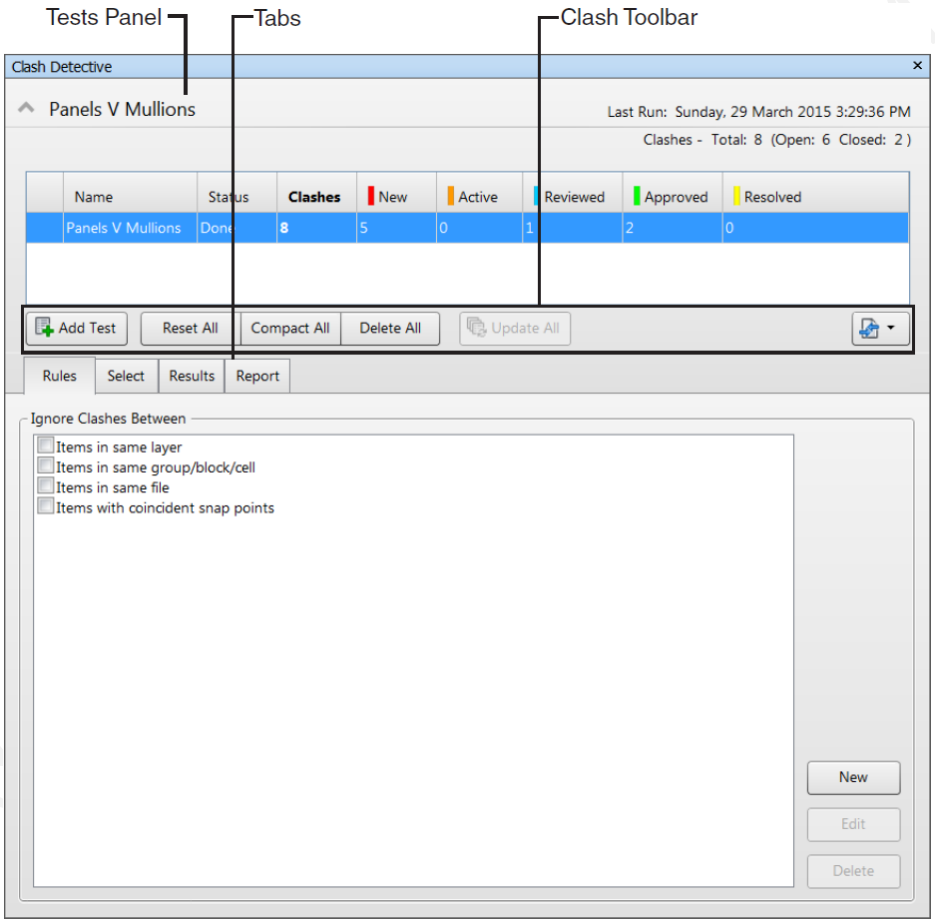
Hard: In this type of clash, only the objects that physically intersect will be considered as clashing.

Hard (Conservative): In this type of clash, even though the geometry triangles are not intersecting, the objects will be considered as clashing.

Clearance: In this type of clash, the objects will be considered as clashing if they come within the specified distance of each other.

Duplicates: This type of clash is performed to make sure there are no duplicate overlapping objects in the model.

2.10 Working with the Clash Detective Module



ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.10 Working with the Clash Detective Module

Tests Panel

The **Tests** panel shows all the information about all the clash tests available in the current Autodesk Navisworks scene. Note that this panel is blank unless you first add a test using the **Add Test** button on the **Clash** toolbar. Once you have performed the clash test, a brief summary about when the test was last performed and the number of open and closed clashes are displayed on the top left of this panel. The various columns displayed in this panel are discussed next.

Name

This column displays the name of the tests that you have added or performed in the current scene. If the model has altered since the last test was run, an error symbol will be displayed on the left of the name of the clash informing you that the model or the test settings have changed since the last run. In this case, you can rerun the test to update the results to reflect the current state of the model.

Status

This column displays the status of the clash test. If the test is performed, this column will display **Done**. If the clash is not yet performed, this column will display **New**.

Clashes

This column displays the total number of clashes found in the test. If the test is not performed yet, this column will display **0**.

2.10 Working with the Clash Detective Module

New
This column displays the total number of new clashes found when you run the test for the first time or the new clashes found when you rerun the test after modifying the model.

Active
This column displays the total number of the clashes that are found to be still active when you rerun the test after modifying the model.

Reviewed
This column displays the total number of clashes that have been reviewed using the **Status** column of the **Results** tab.

Approved
This column displays the total number of clashes that have been approved using the **Status** column of the **Results** tab.

Resolved
This column displays the total number of clashes that have been resolved by making modifications in the model and rerunning the clash test. You can also use the **Status** column of the **Results** tab to override the display of the clash as resolved.

2.10 Working with the Clash Detective Module

Clash Toolbar

This toolbar provides the following buttons:

Add Test

This button is used to add clash tests to the **Tests** panel. If there are no clash tests added, the **Tests** panel will be blank and all the options in the **Clash Detective** window will be grayed out.

Reset All

This button resets the status of all the clashes to **New**, which means that the tests are set to a state where they are not yet run.

Compact All

This button removes all the resolved clashes from the **Tests** panel for all the clash tests.

Update All

This button will only be active if there are changes made in the model and the clash tests needs an update or when you reset the clash tests using the **Reset All** button. By clicking the **Update All** button, the tests are rerun to show the results based on the current state of the model.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.10 Working with the Clash Detective Module

Import/Export Clash Tests

If you regularly perform clashes by searching objects in the scene or by selecting the objects based on their properties, you can export and import those clash setups using this flyout. Note that only the clash selections based on the search sets or properties of the objects can be exported and imported. When you click the **Export Clash Tests** option in this flyout, the **Export**... dialog box is displayed that allows you to save the clash test settings as a .XML file. You can specify the name and location of the clash test settings using this dialog box. To import a saved test, click the **Import Clash Tests** option from the flyout and use the **Import**... dialog box to select and import the clash test settings.

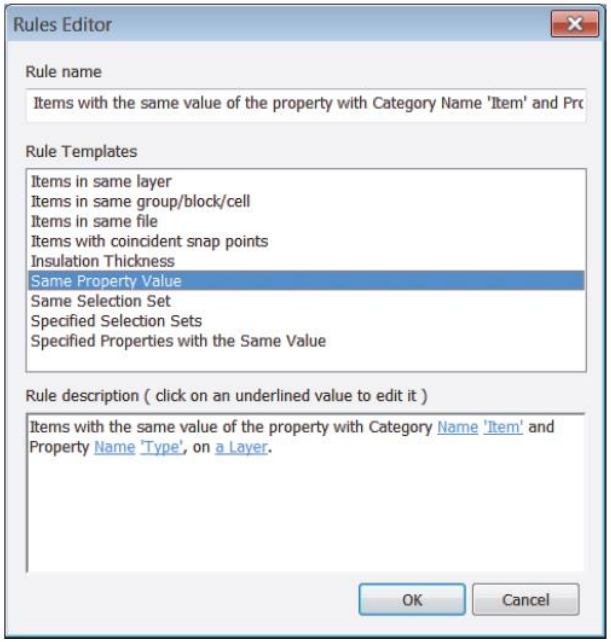
***Tip**: You can use the **Collapse** or **Expand** arrows on the top left of the **Tests** panel to hide or show this panel and the **Clash** toolbar.*

Prepared By Kambiz
Do Not Dis

2.10 Working with the Clash Detective Module

Rules Tab

This tab allows you to ignore objects matching the rules criteria from the clash tests. There are some default rules available in this tab, as shown in Figure 1. To ignore objects during clash test using any of these rules, select the tick box on the left of that rule. To create a new rule, click the **New** button in this tab to display the **Rules Editor** dialog box. Figure 2 shows a custom rule created using this dialog box.

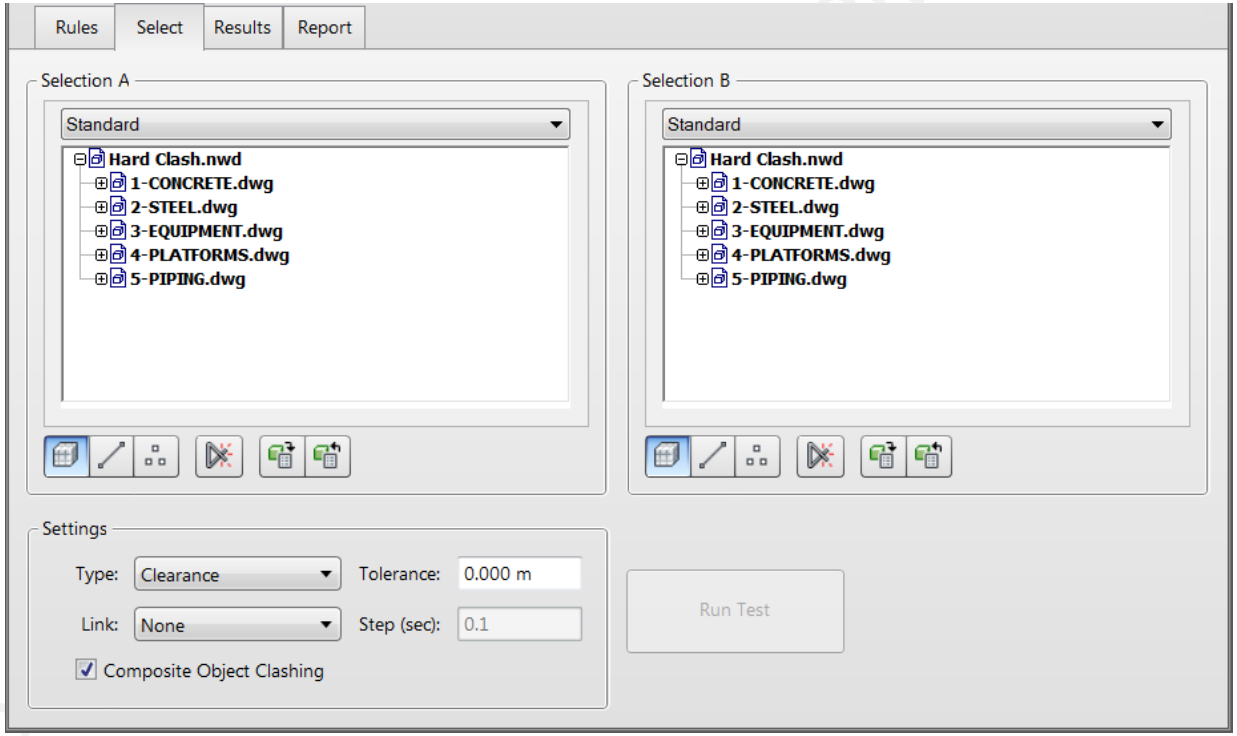


ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.10 Working with the Clash Detective Module

Select Tab

This tab allows you to select the objects between which you want to perform clash tests, as shown in Figure 3. The options available in this tab are discussed next.



ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.10 Working with the Clash Detective Module

Select Tab

Selection A Area

This area provides the options to make the first selection for the clash test. The list at the top in this area allows you to control the display the tree in this area. By default, **Standard** is selected from this list. As a result, the standard tree is displayed that you can expand and select the object in the first selection set. If you want to select saved search or selection sets as your first selection, from the list at the top in this area, select **Sets**. The toolbar available at the bottom of this area provides the following buttons:

Surfaces

Use this button if the objects in the first selection comprise of surfaces. It is important to mention here that the solid models created in the native CAD software are considered as surface objects in Autodesk Navisworks.

Lines

Use this button if the objects in the first selection are wireframe objects made up of lines, arc, circles, and so on.

Points

Use this button if the objects in the first selection are made up of points. This is generally used when you are performing clashes with the point cloud data.

2.10 Working with the Clash Detective Module

Select Tab

Self-Intersect
Use this button if you want to perform clash tests of the objects with themselves.

Use Current Selection
Use this button if the objects that you want in the first selection set are already selected in the graphics window. When you click this button, the objects that are selected in the graphics window are also highlighted in the tree in this area.

Select in Scene
This button is used to highlight in the graphics window the objects you have selected from the tree in this area.

Selection B Area

This area provides the options to make the second selection for the clash test. All these options are the same as those discussed in the **Selection A** area.

Settings Area

The options available in this area are discussed next.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.10 Working with the Clash Detective Module

Select Tab

Type
This list allows you to select the type of clash test you want to run. As mentioned earlier in this chapter, you can perform **Hard**, **Hard (Conservative)**, **Clearance**, and **Duplicates** test.

Tolerance
This box allows you to enter a tolerance value for the clashes. If the clash is within this tolerance value, it will be ignored.

Link
This list allows you to link a timeliner simulation or an animator scene to the clash test. The timeliner simulation is generally linked when the construction of the new facility begins after the demolition of an existing facility. In the Autodesk Navisworks scene, generally both the models are shown because of which there will be clashes between those two models. However, linking a timeliner animation that includes demolition of the first model before the new model is constructed ensures there will be no clash between them. The animator scenes are generally linked when you have moving objects in the scene and you want to make sure that during their movement, they do not clash with other objects in the scene.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.10 Working with the Clash Detective Module

Select Tab

Step
This box allows you to enter the step size of the timeliner simulation. Note that this box will only be available after you have linked a timeliner simulation to the clash test.

Composite Object Clashing
If this tick box is selected, the clashes with composite objects will be considered as a single clash. This will reduce the total number of clashes that are displayed.

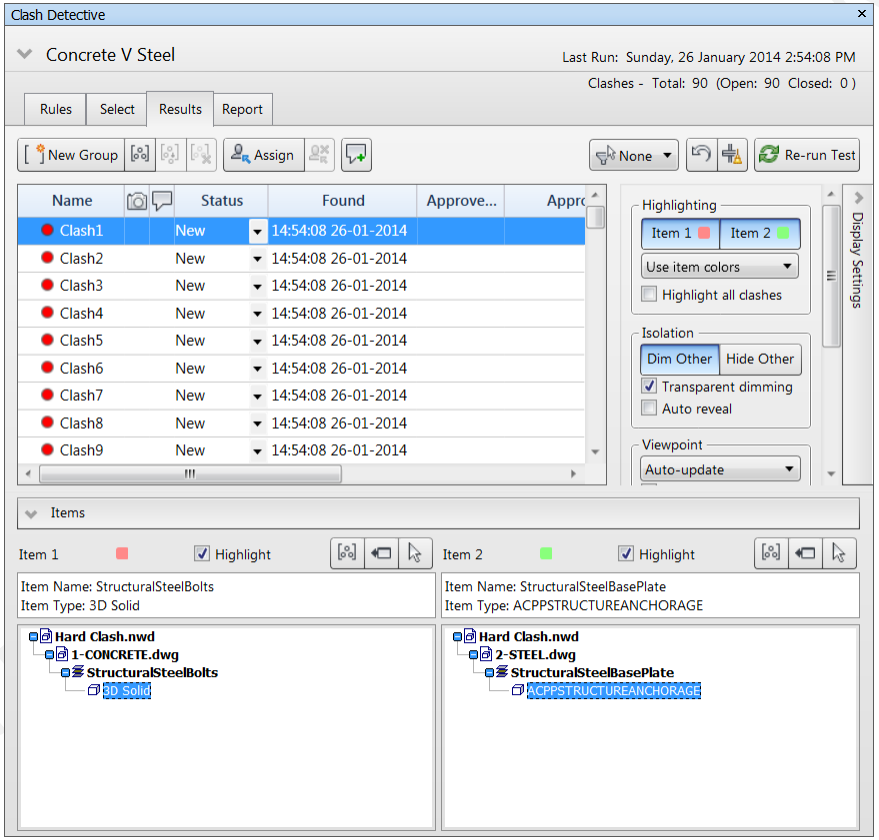
Run Test

After you have selected the two sets of objects for clash test and specified the clash settings, click this button to run the test. Once the test run is completed, you will be taken to the **Results** tab and the currently selected clash result will be displayed in the graphics window.

Prepared By Kar... Do Not D...

2.10 Working with the Clash Detective Module

Results Tab



Prepared by MSc

Results Tab

Results Toolbar

The **Results** toolbar is available at the top in the **Results** tab. The buttons available in this toolbar are discussed next.

New Group

If you have a large number of clashes displayed in the results tab, it is recommended to organize them in groups. Clicking this button creates a new group that you can rename to the required name. The clash results can then be dragged and dropped in the new group. You can also drag the groups in the **Results** area to move it up or down in the list.

Group Selected Clashes

This button will only be available when you have clashes selected from the **Results** area. Clicking this button will create a new group and will move the selected clashes in that group. Figure 5 shows the **Results** area with the clashes organized in the **Pipes** and **Supports** groups.

Prepared By
Do N
MSc

2.10 Working with the Clash Detective Module

Results Tab

Remove from Group
This button will only be available when you select a clash result that is placed in a group. Clicking this button will remove the selected clash result from that group.

Explode Group
This button will only be available when you select a group. Clicking this button will delete the selected group and move all the results outside.

Assign
This button is used to assign the responsibility of the selected result to someone in the team. When you click this button, the **Assign Clash** dialog box is displayed in which you can specify the name of the person who will be assigned the responsibility of this clash. You can also enter some notes about this clash in the dialog box, as shown in Figure 6. The name of the person to which the clash results have been assigned will be displayed in the **Assign To** column in the **Results** area. Also, the **Comments** column will display a number, indicating that there is some information added to the clash result. This information will be displayed in the **Comments** window when you select the clash result.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

Results Tab

Unassign

This button is used to unassign the responsibility of the selected result from the person to which it was assigned earlier. Note that when you select this clash result, the **Comments** window will show two comments. The first one will be about assigning the clash and the second one will be about unassigning.

Add Comment

Clicking this button displays the **Add Comment** dialog box in which you can enter the comment about the clash. The comment that you add will be displayed in the **Comments** window when you select this clash.

***Tip**: As mentioned in the earlier chapters, to go to the second line of text in the **Comments** dialog box, hold down the CTRL key and then press ENTER.*

Filter

The **Filter** flyout allows you to filter the clash results displayed in the **Results** area. By default, **None** is clicked in this flyout. Therefore, the results are not filtered. Click the **Exclusive** option to only show the clash results that involve all the currently selected objects. Click the **Inclusive** option to only show the clash results that involve at least one of the currently selected objects.

2.10 Working with the Clash Detective Module

Results Tab

Reset

Clicking this button will reset the clash test to **New** and will delete all the clash results. You will have to run the test again to view the clash results.

Compact

Clicking this button will delete all the resolved clashes from the **Results** area.

Re-run Test

Clicking this button will rerun the clash test. This is generally used when you have made changes in the model and you want to update the clash results.

Results Area

The **Results** area displays all the clash results in a tabular format, as shown in Figure 4. The information about the clashes are displayed in various columns in this area. You can click on any clash result in this area to highlight that clash in the graphics window. The clash results in the **Results** area can also be organized in groups for ease of understanding, as shown in Figure 5. Right-clicking on any of the columns in this area displays a shortcut menu from which you can click **Choose Columns** to add or remove columns that are displayed in this area.

2.10 Working with the Clash Detective Module

Results Tab

Display Settings Area

This is a collapsible area displayed on the right of the **Results** area. This area displays the options to control the visibility of the clashes in the graphics window, see Figure 7. These options are discussed next.

Highlighting Area

This area is used to control the highlighting of the first and second sets of clashing objects. By default, the **Item 1** and **Item 2** buttons in this area are clicked. As a result, both the clashing items are highlighted when you click a clash result. The highlight colors for the two items are defined in the **Options Editor dialog box > Tools > Clash Detective**. The list in this area allows you to specify whether the objects should be highlighted in the graphics window using the item colors specified in the **Options Editor** or using the color of the clash status displayed in the **Results** area. Selecting the **Highlight all clashes** will highlight all the clashes found in the current clash test.

Prepared By K... Do Not...

2.10 Working with the Clash Detective Module

Results Tab

Isolation Area

This area is used to control the visibility of the objects around the clashing objects. By default, the **Dim Other** button is clicked. As a result, all the objects other than the objects participating in the currently selected clash are displayed in gray. Clicking the **Hide Other** button will hide all the objects other than the ones participating in the currently selected clash. The **Transparent dimming** tick box ensures the dimmed objects are also displayed transparent so you can clearly see the clashing objects. Selecting the **Auto reveal** tick box will temporarily turn off the visibility of the objects interfering with your view and the clashing objects. This is generally used when the **Transparent dimming** tick box is not selected.

Viewpoint Area

This area is used to control the viewpoint that displays the clashes in the graphics window. Selecting the **Auto-update** option from the list in this area ensures that if you zoom or pan in the graphics window to view the clash from a different viewpoint, it is saved and restored when you click on the same clash next time. In this case, the **Saved Viewpoint** column in the **Results** area will show a viewpoint icon next to the result. Selecting the **Auto-load** option ensures that the changes made in the clash view by zooming or panning are not saved. Selecting the **Manual** option will not automatically take you to a clash view when you click on the clash result. You will have to manually zoom or pan to the clashing objects in the graphics window. The **Animate transition** tick box allows a view to smoothly transition from one clash point to the other instead of an abrupt change in the view. The **Focus on Clash** button is generally used to restore the original clash view if you have changed it by moving or panning.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.10 Working with the Clash Detective Module

Results Tab

Simulation Area

This area is used when you have linked a timeliner animation to the clash test. Selecting the **Show simulation** tick box in this area ensures that the timeliner simulation moves to the exact time when the selected clash occurs.

View in Context Area

This area is used to view the clashes on the screen in context of all the objects in the scene, the file containing the clashing objects, or the home view. You can select the option in whose context you want to view the clash and then click the **View** button. The view will zoom out to the selected context and then zoom back into the clash view.

Items Area

This is a collapsible area displayed at the bottom of the **Results** area. This area displays the items that are participating in the current clash. These items, along with their names and types are displayed in the tree view in the **Item 1** and **Item 2** panes, as shown in Figure 8.

The options available at the top of the **Item 1** and **Item 2** areas are the same and are discussed next.

Highlight

This tick box is used to highlight the first or second clashing item in the graphics window.

2.10 Working with the Clash Detective Module

Results Tab

Group Clashes Involving Items
Clicking this button will find all the clashes that the selected item is involved in and will group them together. You can rename the group to any desired name.

SwitchBack
Clicking this button will switch back to the native CAD software to edit the model. The view of the model, when it opens in the native CAD software, will match the view in the Autodesk Navisworks graphics window. It is important to mention here that the native CAD software has to be installed on the same machine and need to be ready for switch back. This will be discussed in detail later in this chapter.

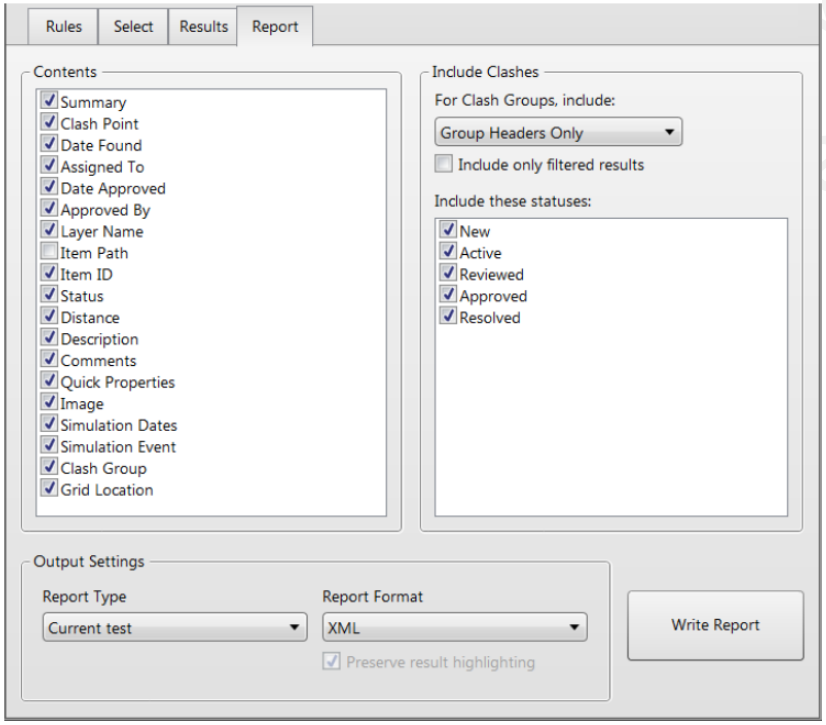
Select in Scene
Clicking this button will select and highlight the clashing item in the graphics window.

Prepared By Kar... Do Not ... MSc

2.10 Working with the Clash Detective Module

Report Tab

This options in this tab are used to write a clash report, as shown in Figure 9.



2.10 Working with the Clash Detective Module

Report Tab

Contents Area

This area lets you select the content you want to display in the clash report. You can select the tick boxes on the left of all the items that you want to display in the report.

Include Clashes Area

This area lets you select the type of clashes to be included in the report. These options are discussed next.

For Clash Groups, include List

This list allows you to select what items you want to include in the report for clash groups. Selecting the **Group Headers Only** option will only include the names of the groups in the report, along with the clashes that are not grouped. The clashes organized inside those groups will not be displayed in the report. Selecting the **Individual Clashes Only** option will include the clashes inside the groups in the report, along with the clashes that are not grouped. In this case, the group names will not be displayed in the clash report. As a result, you will not be able to know if the clashes were grouped. Selecting the **Everything** option will include the group names and the clashes organized in those groups, along with the clashes not grouped in the clash report.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.10 Working with the Clash Detective Module

Report Tab

Include only filtered results Tick Box

Selecting this tick box will ensure that only the results that were filtered using the **Filter** flyout on the **Results** tab are displayed in the clash report.

Include these statuses Area

This area shows the status of the various clashes that you can include in the clash report.

Output Settings Area

This area provides various options to specify the clash report output. These options are discussed next.

Report Type List

This list allows you to specify what results need to be included in the report. By default, the **Current test** option is selected. As a result, only the results of the currently selected clash test are included in the report. Selecting the **All tests (combined)** option will include the results of all the clash tests and will combine them together. Selecting the **All tests (separate)** option will write separate reports for all the clash tests. The names of the reports will be the same as the names of the clash tests.

2.10 Working with the Clash Detective Module

Report Tab

Report Format List

This list allows you to specify what format you want to write the report in. The formats available are **XML**, **HTML**, **HTML (Tabular)**, **Text**, and **As viewpoints**. Remember that selecting the **As viewpoints** option will create viewpoints in the **Saved Viewpoints** window for each clash result. These viewpoints will be organized inside the folder named the same as the clash test. If the clash results were organized inside groups, there will be subfolders in the **Saved Viewpoints** window containing those viewpoints, as shown in Figure 10.

Write Report Button

Clicking this button allows you to write the report based on the options selected in the **Report** tab. If you have selected the **Current test** or **All test (combined)** option from the **Report Type** list and you click this button, the **Save As** dialog box is displayed. You can use this dialog box to specify the name and location of the report file. However, if you have selected the **All test (separate)** option from the **Report Type** list, when you click this button, the **Browse For Folder** dialog box will be displayed using which you can specify the location of the report file. The names of the reports will be the same as the names of the clash tests.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

PROCEDURES FOR PERFORMING VARIOUS CLASH TEST RELATED ACTIVITIES

The following sections explain procedures for performing various clash test related activities.

Procedure for Performing Hard Clash Tests

The following is the procedure to perform a hard clash test between two sets of objects.

a. Open, append, or merge the files in which you want to perform clashes.

b. From the **Home** ribbon tab > **Tools** ribbon panel, click **Clash Detective** to display this window.

c. Dock the **Clash Detective** window on the left of the graphics window.

d. From the **Clash** toolbar, click the **Add Test** button to add a new clash test.

e. Rename the clash test to the desired name.

f. Click the **Select** tab and from the **Selection A** area, click the first set of objects.

g. From the **Selection B** area, click the second set of objects.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.10 Working with the Clash Detective Module

h. From the **Selection A** and **Selection B** areas, make sure the **Surfaces** buttons are clicked.

i. From the **Type** list in the **Settings** area, select **Hard**, if it is not already selected.

j. In the **Tolerance** edit box, enter the tolerance value up to which the clashes are to be ignored.

k. Click the **Run Test** button, the clashes will be performed and the results will be displayed in the **Results** tab.

Prepared By Kambiz Ko
Do Not Distribu

2.10 Working with the Clash Detective Module

Procedure for Interrogating the Clash Results

Once the clash is performed between two sets of objects, the results are displayed in the **Results** tab. The following is the procedure to interrogate the clash results.

a. In the **Clash Detective** window, click the **Results** tab, if it is not already active; the first clash will be highlighted and the clashing objects will be displayed in the graphics window.

b. In the **Results** area, scroll to the right and notice the clash distance and clash point in their respective columns.

c. From the **Display Settings** area on the left of the **Results** area, click the **Hide Other** button in the **Isolation** area; all the other objects in the view will be hidden.

d. From the same area, click the **Dim Other** button; the other objects will be displayed as dim.

e. Clear the **Transparent dimming** tick box from the same area; the other objects will no more be displayed as dim.

f. Click the **Auto reveal** button; the objects obscuring the clashing objects will be removed from the display.

2.10 Working with the Clash Detective Module

g. Select the **Transparent dimming** tick box and then clear the **Auto reveal** tick box.

h. Scroll down in the **Display Settings** area and make sure **All** is selected from the list in the **View in Context** area.

i. From the same area, click the **View** button; the camera will zoom out to the extents of all the objects in the scene and will zoom back to the clash, showing you the clash in context of all the objects in the scene.

j. Perform steps b to k for some other clashes shown in the **Results** area.

Prepared By Kambiz K... Do Not Distrib...

2.10 Working with the Clash Detective Module

Procedure for Writing the Clash Report

The clashes found in the Autodesk Navisworks model can be written in a report. The following steps show you the procedure of writing the clash report.

a. In the **Clash Detective** window, click the **Report** tab.

b. From the **Contents** area, select the tick boxes of all the content you want to include in the report.

c. From the **Include Clashes** area, click the tick boxes of all the statuses you want to include in the report.

d. From the **Report Type** list in the **Output Settings** area, select the report type that you want to write. If you have multiple clash tests performed, you can select the option to create a combined report for all the tests or separate reports for all the tests.

e. From the **Report Format** list, select the format in which you want to write the report.

f. Click the **Write Report** button.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.10 Working with the Clash Detective Module

g. If you selected the report format other than **As viewpoints** from the **Report Format** list and **Current test** or **All tests (combined)** from the **Report Type** list, the **Save As** dialog box will be displayed using which you can specify the name and the location of the report. If you selected **All tests (separate)** from the **Report Type** list, the **Browse For Folder** dialog box will be displayed using which you can specify the folder in which all the test reports will be saved. In this case, the reports will be named the same as the clash names.

h. If you selected **As viewpoints** from the **Report Format** list, the viewpoints will be saved in the **Saved Viewpoints** window inside a folder with the same name as the clash test.

Prepared By Kambiz K... MSc
Do Not Distrib...

2.10 Working with the Clash Detective Module

Procedure for Resolving the Clashes Using the SwitchBack Functionality

As mentioned earlier, the switchback functionality allows you to switchback to the native CAD software from the NWF or NWD files, if you are working with the native files. This is a very useful functionality to modify the CAD model in the native CAD software to resolve clashes. However, for this functionality to work, you need to have the CAD software installed on the same computer as Autodesk Navisworks and also get it ready for switchback.

The following procedures show how to use the switchback functionality to resolve clashes with various CAD software.

Prepared By Kambiz ... MSc
Do Not Distri...

2.10 Working with the Clash Detective Module

Using the Switchback Functionality to Resolve Clashes with AutoCAD or AutoCAD-based Software

Remember that the switchback functionality is only available in AutoCAD 2004 or later versions. The following steps show you how to use the switchback functionality to resolve the clashes in AutoCAD or AutoCAD-based software, such as AutoCAD Plant 3D, AutoCAD Mechanical, and so on.

a. Start AutoCAD or AutoCAD-based program.

b. Type a command **NWLOAD** and press ENTER; the command line shows **Navisworks ready**.

***Note**: If you regularly work with the switchback functionality, you can add the option to automatically load the Autodesk Navisworks exporter plug-in in AutoCAD. This can be done by adding the required version of the Autodesk Navisworks exporter in the **Startup Suite** of the **Load/Unload Applications** dialog box displayed using the **APPLOAD** command in AutoCAD. The exporter is generally available in the **<Install Drive>:\Program Files\Common Files\Autodesk Shared\Navisworks\20XX\nwexportdbx20XX** folder where XX is the release number.*

2.10 Working with the Clash Detective Module

*Tip: When you type the **NWLOAD** command and get an error message about the unknown command, this means that Autodesk Navisworks was installed before installing AutoCAD or AutoCAD-based software. This can be resolved by loading the Autodesk Navisworks exporter utility using the **APPLOAD** command. As mentioned earlier, the exporter is generally available in the **<Install Drive>:\Program Files\Common Files\Autodesk Shared\Navisworks\20XX\nwexportdbx20XX** folder where XX is the release number.*

c. Return to the Autodesk Navisworks window and use switchback on the DWG file; the model will be opened in AutoCAD or AutoCAD-based software zoomed to the same view as that in Autodesk Navisworks.

d. Make the modification in the model, as required, and then save the file.

e. Return to the Autodesk Navisworks window and then click the **Refresh** button on the **Quick Access Toolbar**; the changes you made in the DWG file will be reflected in the Autodesk Navisworks scene.

f. In the **Results** tab of the **Clash Detective** window, click the **Re-run Test** button to rerun the clash test.

2.10 Working with the Clash Detective Module

Using the Switchback Functionality to Resolve Clashes with Microstation V7/V8 or Software Based on them

The following steps show you how to use the switchback functionality to resolve the clashes in Microstation V7/V8 or the software based on them.

a. Start Microstation V7/V8 or the program based on them.

b. Click **Utilities** > **Key In** to display the **Key-In** dialog box.

c. In the dialog box, type MDL LOAD NWEXPORT10 and then exit the dialog box.

***Note**: If you regularly work with the switchback functionality with Microstation, it is better to add NWEXPORT10 to the MDL plug-ins.*

d. Return to the Autodesk Navisworks window and use switchback on the Microstation file; the model will be loaded in Microstation.

e. Make the required changes in the model in Microstation and save the file.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.10 Working with the Clash Detective Module

f. Return to the Autodesk Navisworks window and then click the **Refresh** button on the **Quick Access Toolbar**; the changes you made in the Microstation will be reflected in the Autodesk Navisworks scene.

g. In the **Results** tab of the **Clash Detective** window, click the **Re-run Test** button to rerun the clash test.

Prepared By Kambiz Kordani
Do Not Distribute

2.10 Working with the Clash Detective Module

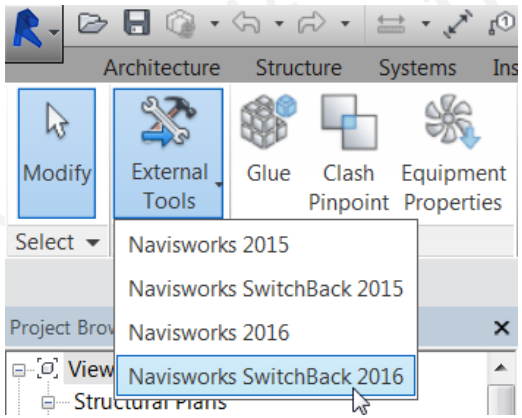
Using the Switchback Functionality to Resolve Clashes with Autodesk Revit

Remember that the switchback functionality only works with Autodesk Revit 2012 or later. The following steps show you how to use the switchback functionality to resolve clashes in Autodesk Revit.

a. Start Autodesk Revit and then start a new project or open an existing project.

b. Click the **Add-Ins** ribbon tab.

c. From the **External** ribbon panel, click **External Tools** flyout > **Navisworks Switchback 2015**, as shown in Figure 11.



2.10 Working with the Clash Detective Module

d. Return to the Autodesk Navisworks window and use switchback on the Autodesk Revit file; the model will be loaded in Autodesk Revit.

e. Make the required changes in the model in Autodesk Revit and save the file.

f. Return to the Autodesk Navisworks window and then click the **Refresh** button on the **Quick Access Toolbar**; the changes you made in the Autodesk Revit will be reflected in the Autodesk Navisworks scene.

g. In the **Results** tab of the **Clash Detective** window, click the **Re-run Test** button to rerun the clash test.

Prepared By Kambiz ... Do Not Distri...

2.10 Working with the Clash Detective Module

Using the Switchback Functionality to Resolve Clashes with Autodesk Inventor

Autodesk Inventor does not need any special setup to get it ready for switchback. It just needs to be started alongside Autodesk Navisworks. However, it is recommended that the project is changed to the project of the design you are working on in Autodesk Navisworks. The following steps show how to use the switchback functionality to resolve clashes with Autodesk Inventor.

a. Start Autodesk Inventor and change the project to the project of the file opened in Autodesk Navisworks.

b. Return to the Autodesk Navisworks window and use switchback on the Autodesk Inventor file; the model will be loaded in Autodesk Inventor.

c. Make the required changes in the model in Autodesk Inventor and save the file.

d. Return to the Autodesk Navisworks window and then click the **Refresh** button on the **Quick Access Toolbar**; the changes you made in the Autodesk Inventor will be reflected in the Autodesk Navisworks scene.

e. In the **Results** tab of the **Clash Detective** window, click the **Re-run Test** button to rerun the clash test.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.10 Working with the Clash Detective Module

Procedure for Linking a TimeLiner simulation or an Animator Scene to the Clash Test

As mentioned earlier, you can link a timeliner simulation or an animator scene to the clash test. The following are the procedures to do this.

Linking the TimeLiner Simulation to the Clash Test

The following steps show how to link a timeliner simulation to the clash test.

a. In the **Select** tab of the **Clash Detective** window, select the two sets of objects for performing the clash test. Note that these objects need to be included in the timeliner simulation as well.

b. From the **Link** list in the **Settings** area, select the timeliner simulation that you want to link to the clash test.

c. In the **Step (sec)** box, make sure the value is **0.1**.

***Tip**: It is better to have the time step value to 0.1. The reason is if the value is large, the clashes between the time steps will not be picked during the test.*

d. Click **Run Test**; the clash test will be run and the results will be displayed in the **Results** tab.

2.10 Working with the Clash Detective Module

e. Dock the **TimeLiner** window below the graphics window and then click the **Simulate** tab.

f. In the **Results** tab of the **Clash Detective** window, scroll down in the **Display Settings** area and make sure the **Show simulation** tick box is selected in the **Simulation** area.

g. From the **Results** area, click any clash; the view in the graphics window changes to the clash view and also the simulation bar in the **Simulate** tab of the **TimeLiner** window moves to the date when that clash was detected.

h. Repeat step g for some other clashes and notice how the simulation bar in the **Simulate** tab of the **TimeLiner** window moves to the dates of those clashes.

Prepared By Kambiz ... MSc
Do Not Distri...

2.10 Working with the Clash Detective Module

Linking the Animator Scene to the Clash Test

The following steps show how to link an animator scene to the clash test.

a. In the **Select** tab of the **Clash Detective** window, select the two sets of objects for performing the clash test. Note that these objects need to be included in the animator scene animation as well.

b. From the **Link** list in the **Settings** area, select the animator scene that you want to link to the clash test.

c. In the **Step (sec)** box, make sure the value is **0.1**.

***Tip**: It is better to have the time step value to 0.1. The reason is if the value is large, the clashes between the time steps will not be picked during the test.*

d. Click **Run Test**; the clash test will be run and the results will be displayed in the **Results** tab.

e. Dock the **Animator** window below the graphics window.

f. In the **Results** tab of the **Clash Detective** window, scroll down in the **Display Settings** area and make sure the **Show simulation** tick box is selected in the **Simulation** area.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.10 Working with the Clash Detective Module

g. From the **Results** area, click any clash; the view in the graphics window changes to the clash view and also the simulation bar in the **Animator** window moves to the time duration when that clash was detected.

h. Repeat step g for other clashes and notice how the simulation bar in the **Animator** window moves to the time duration of those clashes.

Prepared By Kambiz Kordani MSc
Do Not Distribute

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.10 Working with the Clash Detective Module

Procedure for Exporting and Importing Clash Tests

The following procedures show how you can setup a clash test using search sets or using the properties selection and then export it for the use in other files.

Exporting and Importing Clash Tests Based on Search Sets

The following steps show you how to setup a clash test based on search sets and then export and import them.

a. In the **Select** tab of the **Clash Detective** window, select **Sets** from the **Standard** list in the **Selection A** area.

b. Select the first search set to perform the clash test.

c. From the **Standard** list in the **Selection B** area, select **Sets**.

d. Select the second set to perform the clash test.

e. Specify the remaining clash options and click the **Run Test** button; the clash information will be displayed in the **Tests** panel at the top in the **Clash Detective** window.

ذکر مطالب فوق تنها در صورت ذکر منبع بلامانع است.

2.10 Working with the Clash Detective Module

f. From the **Clash** toolbar, click the **Import/Export Clash Tests** flyout and click **Export Clash Tests**; the **Export** dialog box will be displayed.

g. Specify the name and location of the clash setup file and then exit the dialog box.

h. Close the current file and then open the file into which you want to import the clash test setup.

i. From the **Clash** toolbar, click **Import Clash Tests** to display the **Import** dialog box.

j. Browse to the folder where you saved the clash test setup and then double-click on the file; the clash test will be displayed in the **Tests** panel and the objects in the search sets will be selected in the **Select** tab.

k. From the **Select** tab, click **Run Test**; the clash test will be performed based on the imported setup.

Prepared By

2.10 Working with the Clash Detective Module

Exporting and Importing Clash Tests with the Objects Selected Based on the Properties

The following steps show you how to setup a clash test based on objects selected using their properties and then export and import them.

a. In the **Select** tab of the **Clash Detective** window, select **Properties** from the **Standard** list in the **Selection A** area.

b. Select the properties that you want to have in the first set of objects to perform the clash test.

c. From the **Standard** list in the **Selection B** area, select **Properties**.

d. Select the properties that you want to have in the second set of objects to perform the clash test.

e. Specify the remaining clash options and click the **Run Test** button; the clash information will be displayed in the **Tests** panel at the top in the **Clash Detective** window.

f. From the **Clash** toolbar, click the **Import/Export Clash Tests** flyout and click **Export Clash Tests**; the **Export**... dialog box will be displayed.

2.10 Working with the Clash Detective Module

g. Specify the name and location of the clash setup file and then exit the dialog box.

h. Close the current file and then open the file that has objects with properties matching the ones you defined in the exported clash setup.

i. From the **Clash** toolbar, click **Import Clash Tests** to display the **Import** dialog box.

j. Browse to the folder where you saved the clash test setup and then double-click on the file; the clash test will be displayed in the **Tests** panel and the objects in the search sets will be selected in the **Select** tab.

k. From the **Select** tab, click **Run Test**; the clash test will be performed based on the imported setup.

Prepared By Kam... Do Not D...