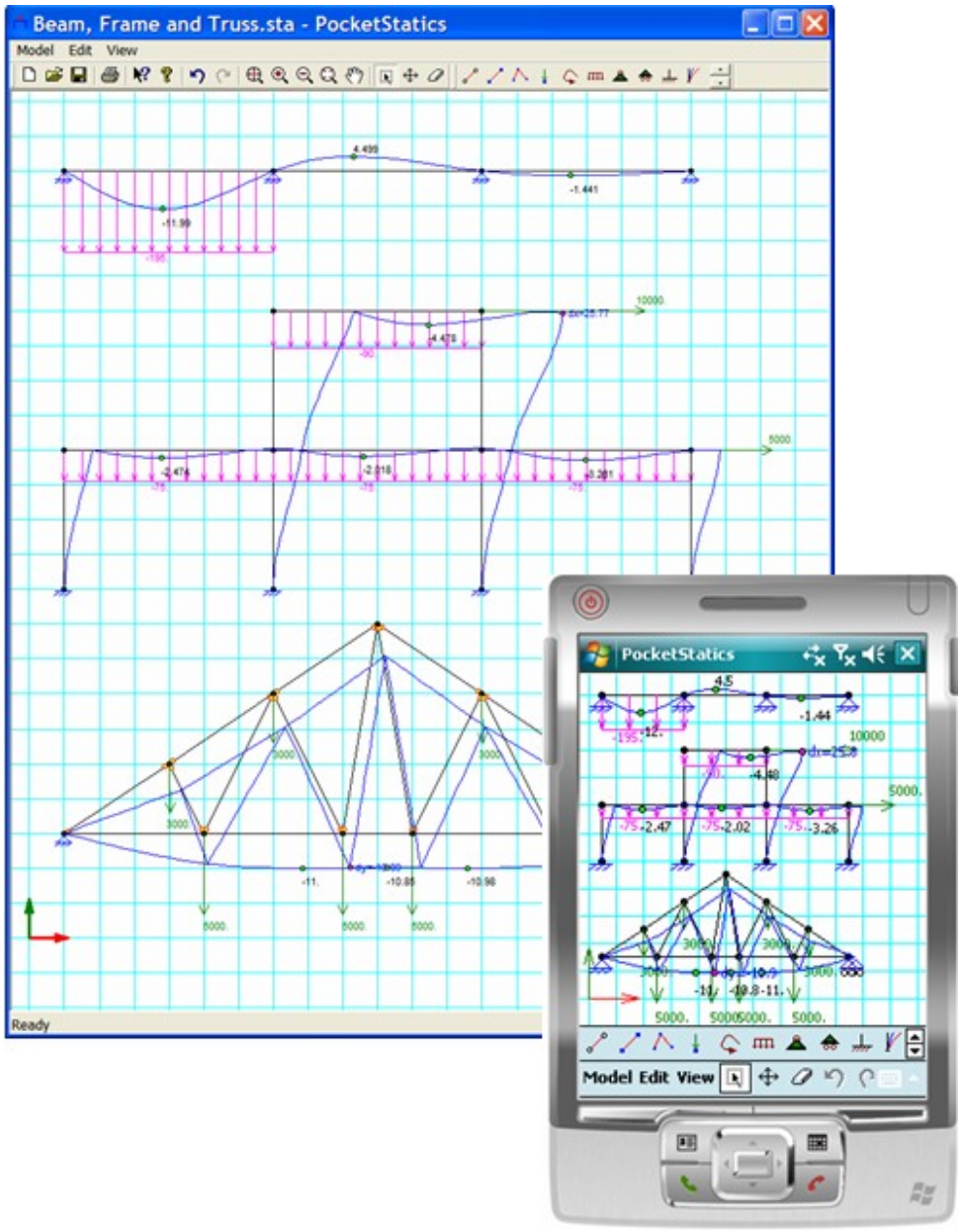
PocketStatics 2.0 Help Manual



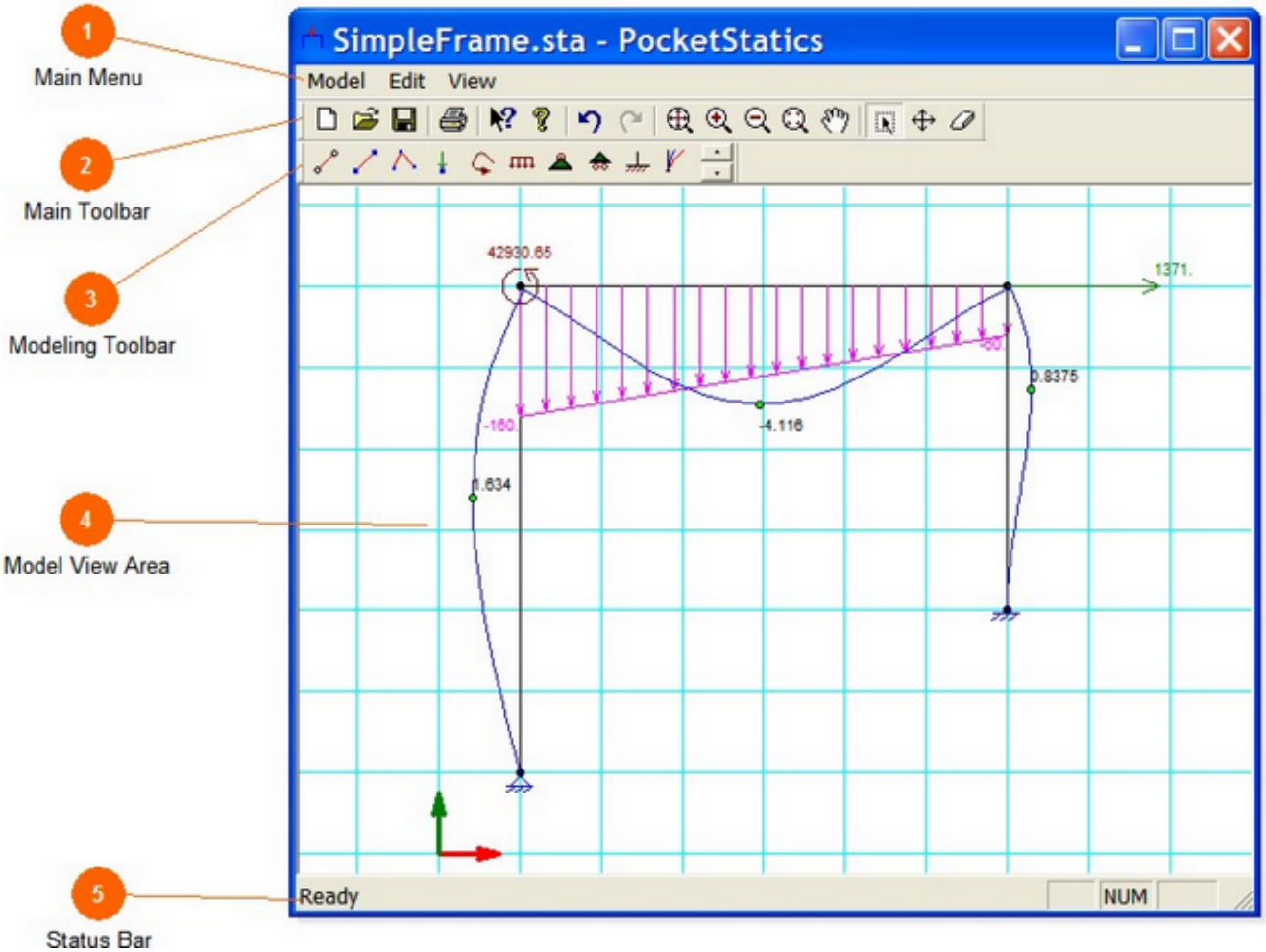
© Earthquake Solutions, 2007

Table of Contents

Welcome to PocketStatics 5
Overview of PocketStatics 7
Analysis Capabilities 9
User Interface Capabilities 10
Using the PocketPC Stylus 12
Ordering PocketStatics 13
Main Toolbar 14
Overview 14
Select 15
Move 16
Erase 17
Modeling Toolbar 18
Modeling Toolbar Overview 18
Draw Truss 20
Draw Frame 21
Draw Multiple Frames 22
Node Force 23
Node Moment 24
Element Distsributed Load 25
Pinned Support 26
Roller Support 27
Fixed Support 28
Autoscale 29
Increase/Decrease Scale 30
Model Menu 31
Clear Model 31
New Model 32
Open... 33
Save 34
Save As... 35
Print... 36
Print Preview 37
Print Setup... 38
About PocketStatics... 39
PocketStatics Help 40
Register... 41
Exit 42
Edit Menu 43
Undo 43
Redo 44
Element Properties... 45
Element Load... 46
Node Coordinates... 47
Node Supports... 48
Node Moment... 49
Node Forces... 50
Grid Dimensions... 51

Options **52**
Options Overview 52
Model View Options 53
Diagram View Options 54
Scale Values Options 55
View Menu **56**
Support Reactions 56
Axial Force Diagram 57
Shear Diagram 58
Moment Diagram 59
Deformed Shape 60
Zoom and Pan **61**
Zoom & Pan Overview 61
Zoom All 62
Zoom In 63
Zoom Out 64
Zoom Dynamic 65
Pan View 66
Context Menus **67**
Overview 67
Element Context Menu 68
Node Context Menu 69
Node Force Context Menu 70
Diagram Context Menu 71
Node Moment Context Menu 72
Shortcuts **73**
List of Shortcuts 73

Welcome to PocketStatics



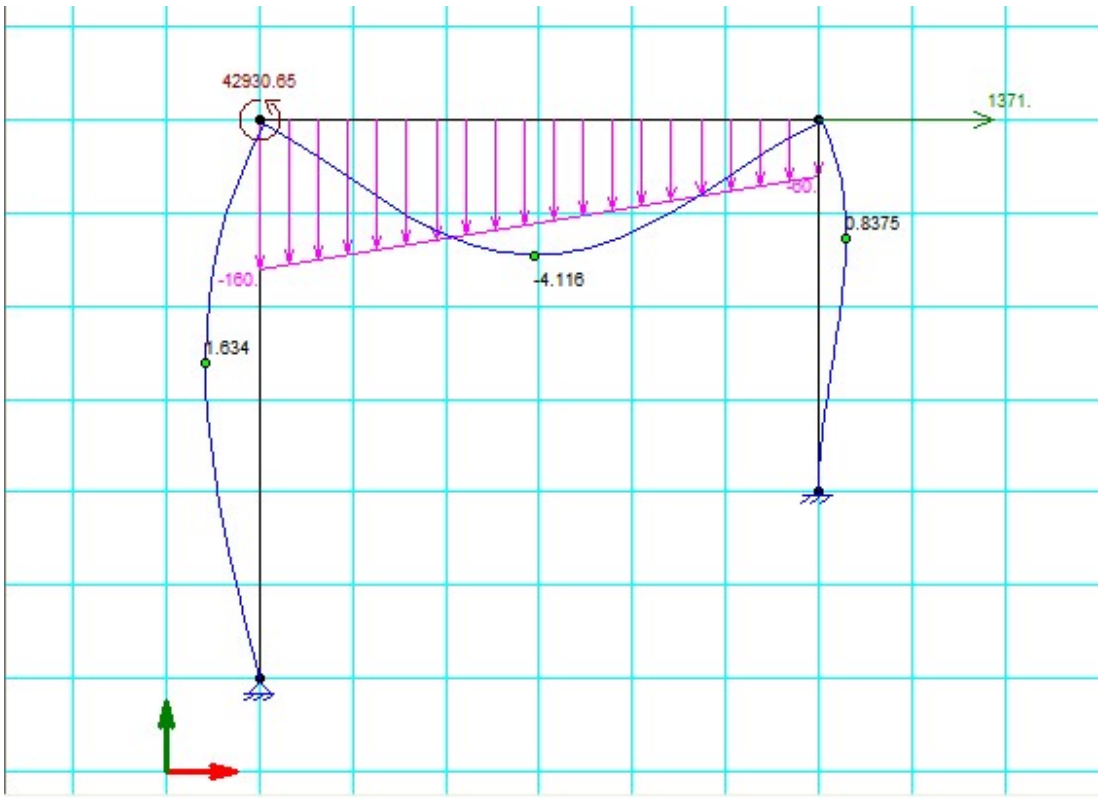
Includes the Model Menu, Edit Menu and View Menu.

Includes standard file and printing buttons, undo/redo, zooming/panning buttons, in addition to basic model tools.

3 [Modeling Toolbar](#)

Includes model building tools.

4 **Model View Area**



5 **Status Bar**

Provides hints and feedback to user.

-o-

Overview of PocketStatics

PocketStatics is a structural analysis program for the PocketPC (Windows Mobile) and Microsoft Windows (Desktop, Laptop or Tablet PC) Operating Systems.

PocketStatics is very interactive, and simulates to a great degree working with an engineering pad and a pencil. Toolbars are very intuitive, and the structure can be simply sketched on the screen using the stylus. The program automatically computes and displays results whenever any changes are made to the structure.

PocketStatics may be the easiest and most intuitive structural analysis program that you will ever use.

Use PocketStatics can be used to solve most statics problems including:

- statically determinate and indeterminate trusses and frames with any number of elements and degrees of freedom,
- simple columns,
- single or continuous beams,
- 2D frames such as moment frames, braced frames,
- 2D trusses
- any 2D structure including buildings and bridges

PocketStatics comes in two editions, one for PocketPC PDA's and one for computers and laptops running Windows or TabletPC.

This makes it extremely useful for:

- civil engineers,
- structural engineers and designers,
- mechanical designers,
- field engineers,
- construction engineers,
- site inspectors,
- students,
- instructors and others.

It is ideal for use in meetings, at construction sites, and if your phone is PocketPC enabled, you can use it wherever you can take your phone.

-o-

Analysis Capabilities

PocketStatics provides analysis features that allow the efficient analysis of almost any two-dimensional frame or truss.

Whether you need to analyze a single beam or column, multiple continuous beams, or a complex frame or truss, PocketStatics provides the capability to perform that analysis.

The user interface is very easy to use and very interactive and responsive which makes ideal for use in meetings, at construction sites, as well at your desk in addition to any other location. This allows you to quickly test ideas and document and analyze your structure whenever you want.

Analysis Capabilities

- Analysis of 2D frames and trusses
- Element distributed loads. Can be trapezoidal (non-uniform)
- Node loads (forces and moments)
- Element end moment releases
- Define frame element properties (modulus of elasticity, area, moment of inertia)
- Compute node deformations, reactions
- Compute deformations, moments, shears, axial forces along element
- Translational and rotational degrees of freedom can be individually fixed for each node.

-o-

User Interface Capabilities

Main User Interface Features

- Highly intuitive interface which allows the use the Pocket PC stylus (PocketPC version) or mouse (Windows version) to easily and quickly sketch your structure
- Unlimited Undo/Redo capability
- Saving and opening of model files
- Ability to share models between the PocketPC and Windows versions, just synchronize your file using ActiveSync!
- Help system on both the PocketPC and Windows version
- Immediate display of deformed shape, reactions, and moment, shear and axial force diagrams
- Powerful zooming and panning capabilities

Detailed User Interface Features

- Interactive definition of structure geometry, boundary condition and loads without having to type any values
- Common boundary conditions (pin, roller, fixity) can be easily assigned through toolbar buttons
- Further refinement of properties and load values can be easily accomplished using simple dialog boxes
- Continuous analysis of the structure and display of results as you make modifications
- Continuous feedback on the stability of the structure
- Ability to show/hide:
  - Nodes
  - Node Labels
  - Node Forces
  - Node Moments
  - Frame Elements
  - Frame Labels
  - Frame End Releases
  - Frame Distributed Loads
- Display of deformed shape, moment diagram, shear diagram, axial force diagram and reactions
- The program displays the exact deformed shape and force diagrams (moment, shear, axial):

- The point with maximum or minimum deformation or force is highlighted
- The value of the maximum deformation or force is also shown which makes it easy to obtain results very quickly.
- Control what values will be shown on diagrams (you can refine how many values you see in order to optimize your view especially on smaller screens)

-o-

Using the PocketPC Stylus

The PocketPC stylus is used similar to the mouse on a computer.
Tapping the stylus on the screen is equivalent to pressing the mouse left-button on a Windows PC.
Since a stylus does not have a right-button, a right-button click is typically simulated on the PocketPC by pressing and holding the stylus against the screen for a brief period (Tap and Hold).

Since this documentation relates to both the PC and PocketPC versions of PocketStatics, in the following pages, both mouse and stylus actions are described.
For example, when a right click is required, it is described as Right-Click/Tap-and-Hold.

-o-

Ordering PocketStatics

Ordering PocketStatics is easy. Just go to http://eqsols.com/orderpocketstatics.aspx or visit http://www.eqsols.com and then proceed to the PocketStatics order page.

Payments are processed through Paypal. All major credit cards are accepted.

-o-

Main Toolbar

Main Toolbar

The main toolbar contains the Select , Move and Erase buttons.

The main toolbar also includes the Undo and Redo buttons (also available on the Edit Menu):

Undo : Reverses the last action (See Undo under the Edit Menu).

Redo : Repeats a action that was undone using Undo (See Redo under the Edit Menu).

On the Windows version of PocketStatics, the main toolbar also contains the following toolbar buttons which can also be accessed from the menu:

New : Creates a new model file

Open : Opens an existing model file

Save : Saves current model to file

Print : Displays the Print dialog (prints current view)

Context Help : Activates Context Help mode. Clicking on another toolbar or menu item brings the help for that item.

About PocketStatics : Shows the About PocketStatics Dialog.

Zoom All : Automatically sets the screen zoom to fit the whole model.

Zoom In : Zooms into the view by a fixed amount (25%).

Zoom Out : Zooms out of the view by a fixed amount (25%).

Zoom Dynamic : Provides dynamic zooming (in and out).

Pan View : Provides dynamic panning (moving) of the view.

-o-

Select

Main Toolbar -> Select

Changes current mode to *Select Mode*. While in *Select Mode*, it is possible to select elements and nodes by clicking/tapping on them or by selection using a selection rectangle.

-o-

Move

Main Toolbar -> Move

While in *Move Mode*, it is possible to move one or more selected nodes. Simply click/tap on one of the selected nodes and drag to the new location. All selected nodes will move by the same distance.

-o-

Erase

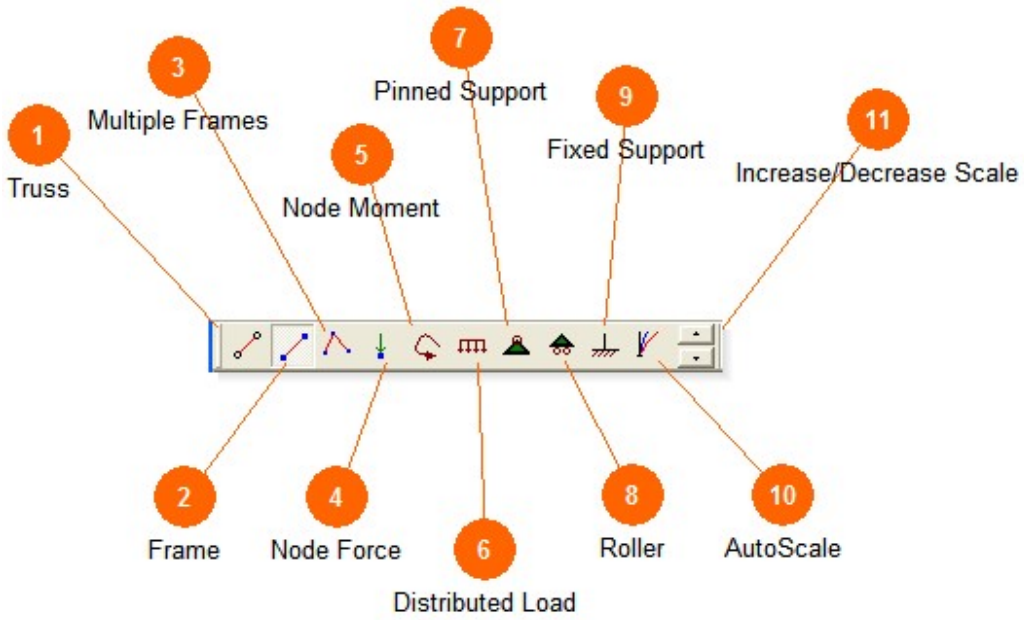
Main Toolbar -> Erase

changes current mode to *Erase Mode*. While in *Erase Mode*, elements, loads and boundary conditions can be erased by clicking/tapping on or by dragging across the objects to be erased.

-o-

Modeling Toolbar

Modeling Toolbar Overview



1 **Truss**

Use to draw a truss element

2 **Frame**

Use to draw a frame element

3 **Multiple Frames**

Use to draw multiple frame elements

4 **Node Force**

Use to draw a node force

5 **Node Moment**

Use to draw a node moment

6 **Distributed Load**

Used to draw a distributed load

7 **Pinned Support**

Assigns a pinned support to a node

8 **Roller**

Assigns a roller support to a node

9 **Fixed Support**

Assigns a fixed support to a node

10 **AutoScale**

Set scale automatically

11 **Increase/Decrease Scale**

Increase/Decrease scale

-o-

Draw Truss

Modeling Toolbar -> Draw Truss

Use this tool to draw truss elements.

Truss elements are frame elements that have moment releases on both ends.
To draw a truss element:

1. Select the Truss tool
2. Click/Tap to select the first point of the truss; if no node exists at this location, it will be created.
3. Click/Tap to select the second point of the truss; if no node exists at this location, it will be created.

See also: Draw Frame, Element Context Menu, Element Properties

-o-

Draw Frame

Modeling Toolbar -> Draw Frame

Use this tool to draw frame elements.

To draw a frame element:
1. Select the frame tool
2. Click/Tap to select the first point of the frame; if no node exists at this location, it will be created.
3. Click/Tap to select the second point of the frame; if no node exists at this location, it will be created.

See also: Draw Truss, Draw Multiple Frames, Element Context Menu, Element Properties

-o-

Draw Multiple Frames

Modeling Toolbar -> Draw Multiple Frames

Use this tool to draw multiple frame elements.

To draw multiple frame element:
1. Select the frame tool
2. Click/Tap to select the first point of the frame; if no node exists at this location, it will be created.
3. Click/Tap to select the second point of the frame; if no node exists at this location, it will be created.
4. Continue selecting points to add frames. Frames will be connected to each other.

See also: Draw Frame, Element Context Menu, Element Properties

-o-

Node Force

Modeling Toolbar -> Node Force

 Use this tool to add a concentrated force to a node.

To add a node force, click/tap on a node and drag to determine the direction and magnitude of the force.

To modify an existing node force, click/tap on the node force and drag.

See also: Node Forces (Edit Menu), Node Context Menu, Node Force Context Menu

-o-

Node Moment

Modeling Toolbar -> Node Moment

Use this tool to add a concentrated moment to a node.

To add a node moment, click/tap on a node and drag to determine the direction and magnitude of the moment.

To modify an existing node moment, click/tap on the node moment and drag.

See Also: Node Moment (Edit Menu), Node Context Menu, Node Moment Context Menu

-o-

Element Distsributed Load

Modeling Toolbar -> Element Distributed Load

Assigns a distributed load to an element.

After selecting the tool, click/tap on the element and drag to draw the distributed load.

To modify an existing distributed element load, click/tap on the distributed load and drag. Note that if the distributed load will be varied uniformly along the element length except if the load is selected near the ends of the element. In this case, the load value will be varied only at the end resulting in a trapezoidal shape.

See Also: Element Load (Edit Menu), Element Context Menu

-o-

Pinned Support

Modeling Toolbar -> Pinned Support

Assigns a roller support to a node.

After selecting the tool, click/tap on the nodes that wish to assign the roller support. If the node already has a pinned support, the support will be removed.

See Also: Roller Support, Fixed Support, Node Supports (Edit Menu), Node Context Menu

-o-

Roller Support

Modeling Toolbar -> Roller Support

Assigns a roller support to a node.

After selecting the tool, click/tap on the nodes that wish to assign the roller support. If the node already has a roller support, the support will be removed.

See Also: Pinned Support, Fixed Support, Node Supports (Edit Menu), Node Context Menu

-o-

Fixed Support

Modeling Toolbar -> Fixed Support

Assigns a fixed support to a node.

After selecting the tool, click/tap on the nodes that wish to assign the fixed support. If the node already has a fixed support, the support will be removed.

See Also: Pinned Support, Roller Support, Node Supports (Edit Menu), Node Context Menu

-o-

Autoscale

Modeling Toolbar -> Autoscale

Click to automatically set the scale of the current diagram.

The diagram can be a deformed diagram, moment diagram, shear diagram or axial force diagram.

-o-

Increase/Decrease Scale

Modeling Toolbar -> Increase/Decrease Scale

Press the up/down buttons to increase/decrease the scale of the current diagram.

The diagram can be a deformed diagram, moment diagram, shear diagram or axial force diagram.

-o-

Model Menu

Clear Model

Model Menu -> Clear Model

Resets the current the model. Erases all objects in the model including all nodes, elements and loads.

-o-

New Model

Model Menu -> New

Creates a new model file.

Shortcut (Windows only): Ctrl + N

-o-

Open...

Model Menu -> Open

Opens an existing model file. The file to be opened may be any file saved by any version of PocketStatics. The Windows version of PocketPC can open files saved by the PocketPC version and vice versa.

Shortcut (Windows only): Ctrl + O

-o-

Save

Model Menu -> Save

Saves the current file.

Shortcut (Windows only): Ctrl + S

-o-

Save As...

Model Menu -> Save As

Saves the current file to a new a file name.

-o-

Print...

Model Menu -> Print

Displays the print dialog.

Note: Printing is only available in the Windows version
Shortcut: Ctrl + P
See Also: Print Preview, Print Setup

-o-

Print Preview

Model Menu -> Print Preview

Shows a print preview of the current model.

Note: Printing is only available in the Windows version
See Also: Print, Print Setup

-o-

Print Setup...

Model Menu -> Print Setup

Brings up the printer setup dialog.
Use this dialog to specify printing properties (example Landscape or Portrait).

Note: Printing is only available in the Windows version
See Also: Print, Print Preview

-o-

About PocketStatics...

Model Menu -> About PocketStatics

Shows the *About PocketStatics* dialog.

-o-

PocketStatics Help

Model Menu -> Help Topics

Shows Help Contents for PocketStatics.
Help can also be shown by pressing F1 (Windows only).
In addition, context help is also supported. Context help can be activated by pressing SHIFT+F1 or by pressing the context menu button on the Main Toolbar (Windows only).
Example, press SHIFT+F1 (Windows only), and then press on a toolbar button or menu item, the help topic for the selected item will then be displayed.

-o-

Register...

Model Menu -> Register

Brings up the dialog for entering the registration code for PocketStatics.
PocketStatics will run in trial-mode for 30-days. After the 30-day period is over, PocketStatics will not function anymore unless a registration code is obtained and entered.
Ordering PocketStatics is easy, just go to http://eqsols.com/orderpocketstatics.aspx or visit http://www.eqsols.com and then proceed to the PocketStatics order page.

-o-

Exit

Model Menu -> Exit

Exit PocketStatics.
If your file is not saved, you will be prompted to save.

Shortcut (Windows only): Ctrl + Q

-o-

Edit Menu

Undo

Edit Menu -> Undo

Reverses the last action. The action can be re-performed using the Redo command.

PocketStatics keeps an unlimited list of undo/redo actions. This allows the user to undo all actions in a given session if needed.
This feature is very useful in recovering from errors and mistakes, and gives the user the freedom to experiment with the model, knowing that all changes can be easily undone.

All actions that are associated with modifying the model such as adding and removing elements, loads and supports, and setting element, node and load properties are undoable and are included in the Undo/Redo history. In short, any action that modifies the model can be undone/redone.

Things that are not included in the Undo history include changes in Options and in the current view (Zooming and Panning).

The Undo and Redo actions can also available through buttons on the main toolbar.

Shortcut (Windows only): Ctrl + Z
See also: Redo

-o-

Redo

Edit Menu -> Redo

Repeats a action that was undone using "Undo".

Shortcut (Windows only): Ctrl + Y
See also: Undo

-o-

Element Properties...

Edit Menu -> Element Properties

Shows a dialog for modifying element properties for currently selected elements. Properties that can be modified are:

- Area (A)
- Modulus of Elasticity (E)
- Moment of Inertia (I)
- Moment Release at end I
- Moment Release at end J

Note: This dialog can be also be displayed for a single element by double-clicking/tapping on an element or by right-clicking/tapping-and-holding on an element and selecting "Element Properties" from the context menu.

See also: Element Context Menu

-o-

Element Load...

Edit Menu -> Element Load

Modifies distributed load for selected elements.
The distributed load can be uniform or may vary linearly over the length of the element.

Note: This dialog can be also be displayed for a single element by double-clicking/tapping on the element's distributed load or by right-clicking/tapping-and-holding on an element and selecting "Element Properties" from the context menu.

See also: Element Context Menu

-o-

Node Coordinates...

Edit Menu -> Node Coordinates

Modifies the node coordinates of a single selected node.

See also: Move Node (on Main Toolbar)

Note: This dialog can also be shown for a single node by double-clicking/tapping on the node or by right-clicking/tapping-and-holding on the node and selecting "Node Coordinates" from the context menu.

See also: Node Context Menu

-o-

Node Supports...

Edit Menu -> Node Supports

Modifies the node supports of one or more selected nodes. If multiple nodes are selected and the boundary conditions for those nodes are not the same, the fixity conditions for directions where it is not the same will appear as grayed.

See also: Pin Support, Roller Support, Fixed Support (on Modeling Toolbar)

Note: This dialog can also be shown for a single node by double-clicking/tapping on the node support or by right-clicking/tapping-and-holding on the node and selecting "Node Supports" from the context menu.

See also: Node Context Menu, Pinned Support, Roller Support, Fixed Support

-o-

Node Moment...

Edit Menu -> Node Moment

Change the applied moment to a single selected node.

Note: This dialog can also be shown for a single node by double-clicking/tapping on the node moment or by right-clicking/tapping-and-holding on the node and selecting "Node Moments" from the context menu.

See also: Node Moment button (on Modeling Toolbar), Node Context Menu, Node Moment Context Menu

-o-

Node Forces...

Edit Menu -> Node Forces

Modifies the forces applied to a single selected node. If a node with multiple forces is selected, a drop-down is shown at the top of the dialog from which one can select the node force to be modified.

It is possible to specify node forces using:
1. Magnitude and Angle (degrees)
2. Magnitude and Aspect Ratio (horizontal:vertical)
3. X and Y force components

Note: This dialog can also be shown for a single node by double-clicking/tapping on the node force (modifies a single force) or by right-clicking/tapping-and-holding on the node and selecting "Edit Node Forces" from the context menu (modifies all forces for this node).

See also: Node Force button (on Modeling Toolbar), Combine Node Forces (Node Context Menu), Node Force Context Menu

-o-

Grid Dimensions...

Edit Menu -> Grid Dimensions

Shows the dialog for modifying the grid size and the screen limits.
When drawing new elements, the new point locations will always snap to the grid size (or to existing nodes). It is possible to later modify the coordinates of a node so they do not fall on the grid.

-o-

Options

Options

Edit Menu -> Options

Brings up the Options Dialog. Accessible from the Edit Menu.

The options dialog has three tabs:

1. Model View Tab
2. Diagram View Tab
3. Scale Values Tab

-o-

Model View Options

Edit Menu -> Options -> Model View Tab

Controls items to show/hide:

Node Options

- Show/Hide Node
- Show/Hide Node Label
- Show/Hide Node Supports
- Show/Hide Node Loads

Element Options

- Show/Hide Frame Elements
- Show/Hide Element Labels
- Show/Hide Element Moment Releases
- Show/Hide Element Distributed Loads

-o-

Diagram View Options

Edit Menu -> Options -> Diagram View Tab

Select for each type of diagram the number of peak values to show.

The peak view settings can be set for the following diagrams:

- Moment Diagram
- Shear Diagram
- Axial Diagram
- Reactions
- Deformed Shape
- Node Deformations

The typical options for each diagram type are:

- All Peaks
- Peak Maximum and Minimum
- Peak absolute value
- None (Hide)

-o-

Scale Values Options

Edit Menu -> Options -> Scale Values Tab

The Scale Values Tab shows the settings for scaling diagrams and forces. The size of different diagrams may be controlled by setting scale values in this tab. The values generally relate the diagram units to the length units on the screen.

Scale values are available for the following:

- Deformed shape
- Moment Diagram
- Shear Diagram
- Axial Diagram
- Node Forces
- Node Moments
- Distributed Loads

-o-

View Menu

Support Reactions

View Menu -> Support Reactions

Displays the support reactions.

Shortcut (Windows only): Ctrl + R

-o-

Axial Force Diagram

View Menu -> Axial Force Diagram

Displays the axial force diagram.

Shortcut (Windows only): Ctrl + A

-o-

Shear Diagram

View Menu -> Shear Diagram

Displays the shear diagram.

Shortcut (Windows only): Ctrl + H

-o-

Moment Diagram

View Menu -> Moment Diagram

Displays the moment diagram.

Shortcut (Windows only): Ctrl + M

-o-

Deformed Shape

View Menu -> Deformed Shape

Displays the deformed shape.

Shortcut (Windows only): Ctrl + D

-o-

Zoom and Pan

Zoom & Pan Overview

View Menu -> Zoome & Pan

Accessible from the View Menu.
Includes five tools:

1. Zoom All
2. Zoom In
3. Zoom Out
4. Zoom Dynamic
5. Pan View

In addition, the mouse scroll wheel can be used to dynamically zoom in and out of the screen.

-o-

Zoom All

View Menu -> Zoome & Pan -> Zoom All

Automatically sets the screen zoom to fit the whole model.

Shortcut (Windows only): A

-o-

Zoom In

View Menu -> Zoome & Pan -> Zoom In

Zooms into the view by a fixed amount (25%).

Shortcut (Windows only): + (plus sign)

-o-

Zoom Out

View Menu -> Zoome & Pan -> Zoom Out

Zooms out of the view by a fixed amount (25%).

Shortcut (Windows only): - (minus sign)

-o-

Zoom Dynamic

View Menu -> Zoome & Pan -> Zoom Dynamic

Provides dynamic zooming (in and out).

To use this feature:
1. Select "Zoom Dynamic" from the "View->Zoom and Pan" menu
2. click/tap anywhere in the view (Zooming will occur relative to this point)
3. To Zoom In, drag the stylus towards to the top-right corner of window
4. To Zoom Out, drag towards to the bottom-left corner of window

Tip (Windows only): Use the mouse scroll wheel to dynamically zoom in and out of the screen.

Shortcut (Windows only): Z

-o-

Pan View

View Menu -> Zoome & Pan -> Pan View

Provides dynamic panning (moving) of the view.

To use this feature:
1. Select "Pan View" from the "View->Zoom and Pan" menu
2. click/tap anywhere in the view (Panning will occur relative to this point)
3. Drag in any direction to pan the view in that direction

Tip (Windows only): A faster way to pan is to hold the middle mouse button and drag.

Shortcut (Windows only): P

-o-

Context Menus

Context Menus

Context menus appear when right-clicking/tapping-and-holding on an part of the model. A context menu contains commands relevant to the part that is clicked/tapped (element, node, force, moment, or background) and may also depend on the currently selected tool. The following context menus are available:

- Element Context Menu
- Node Context Menu
- Node Force Context Menu
- Node Moment Context Menu
- Diagram Context Menu

-o-

Element Context Menu

The Element Context Menu can be displayed by right-clicking/tapping-and-holding on an element or its distributed load.
It contains the following items:

1. Element Properties: Displays the Element Properties Dialog (Also accessible from the Edit Menu)
2. Element Load: Displays the Element Load Dialog (Also accessible from the Edit Menu)
3. Delete Element: Deletes the selected elements
4. Delete Element Load: Deletes the loads for selected elements

Note: The first two items above behave slightly differently when triggered from the Element Context Menu compared to the Edit Menu. When triggered from the Edit Menu, the Element Properties and Element Load dialogs modify properties and loads for all selected elements. However, when selected from the Context Menu, only the properties and load for the element under the cursor/stylus are modified even if multiple elements are already selected.

-o-

Node Context Menu

The Node Context Menu can be displayed by right-clicking/tapping-and-holding on a Node.
It contains the following items:

1. Node Coordinates: Displays the Node Coordinates dialog (Also accessible from the Edit Menu)
2. Node Supports: Displays the Node Supports dialog (Also accessible from the Edit Menu)
3. Node Moment: Displays the Node Moment dialog (Also accessible from the Edit Menu)
4. Node Forces: Displays the Node Forces dialog (Also accessible from the Edit Menu)
5. Combine Node Forces: Combines all forces for this node into one resultant force

Note: The second item above behaves slightly differently when triggered from the Node Context Menu compared to the Edit Menu. When triggered from the Edit Menu, the Node Supports dialog modify node supports for all selected nodes. However, when selected from the Context Menu, only the supports for the node under the cursor/stylus are modified even if multiple nodes are already selected.

-o-

Node Force Context Menu

The Node Force Context Menu can be displayed by right-clicking/tapping-and-holding on a Node Force.
It contains the following items:

1. Modify Force: Displays the Node Forces dialog (Also accessible from the Edit Menu)
2. Delete: Deletes the node force

-o-

Diagram Context Menu

When the cursor/stylus is clicked/tapped on an empty area of the scree, the Diagram context Menu is shown.
The Diagram Context Menu contains the following items:

1. Support Reactions: Displays the support reactions.
2. Axial Force Diagram: Displays the axial force diagram.
3. Shear Diagram: Displays the shear diagram.
4. Moment Diagram: Displays the moment diagram.
5. Deformed Shape: Displays the deformed shape.

The above items are also available from the View Menu.

-o-

Node Moment Context Menu

The Node Moment Context Menu can be displayed by right-clicking/tapping-and-holding on a Node Moment.
It contains the following items:

3. Modify Moment: Displays the Node Moment dialog (Also accessible from the Edit Menu)
4. Delete: Deletes the node moment

-o-

Shortcuts

List of Shortcuts

(Windows Version Only)

Action	Shortcut
Model -> New	Ctrl + N
Model -> Open	Ctrl + O
Model -> Save	Ctrl + S
Model -> Print	Ctrl + P
Model -> Exit	Ctrl + Q
Edit -> Undo	Ctrl + Z
Edit -> Redo	Ctrl + Y
View -> Support Reactions	Ctrl + R
View -> Axial Force Diagram	Ctrl + A
View -> Shear Diagram	Ctrl + V
View -> Moment Diagram	Ctrl + M
View -> Deformed Shape	Ctrl + D
View -> Zoom -> Zoom All	A
View -> Zoom -> Zoom Out	-
View -> Zoom -> Zoom In	+
View -> Zoom -> Zoom Dynamic	Z
View -> Zoom -> Pan View	P

-o-

Index

- A -

About PocketStatics... 39
Analysis Capabilities 9
Autoscale 29
Axial Force Diagram 57

- C -

Clear Model 31

- D -

Deformed Shape 60
Diagram Context Menu 71
Diagram View Options 54
Draw Frame 21
Draw Multiple Frames 22
Draw Truss 20

- E -

Element Context Menu 68
Element Distsributed Load 25
Element Load... 46
Element Properties... 45
Erase 17
Exit 42

- F -

Fixed Support 28

- G -

Grid Dimensions... 51

- I -

Increase/Decrease Scale 30

- L -

List of Shortcuts 73

- M -

Model View Options 53
Modeling Toolbar Overview
Moment Diagram 59
Move 16

- N -

New Model 32
New Model 32
Node Context Menu 69
Node Coordinates... 47
Node Force 23
Node Force Context Menu 70
Node Forces... 50
Node Moment 24
Node Moment Context Menu 72
Node Moment... 49
Node Supports... 48

- O -

Open... 33
Options Overview 52
Ordering PocketStatics 13
Overview 14, 18, 67
Overview 14, 18, 67
Overview of PocketStatics 7

- P -

Pan View 66
Pinned Support 26
PocketStatics Help 40
Print Preview 37
Print Setup... 38
Print... 36

- R -

Redo 44
Redo 44
Register... 41
Roller Support 27

- S -

Save 34
Save 34
Save As... 35
Scale Values Options 55
Select 15
Shear Diagram 58
Support Reactions 56

- U -

Undo 43
Undo 43
User Interface Capabilities 10
Using the PocketPC Stylus 12

- W -

Welcome to PocketStatics 5

- Z -

Zoom & Pan Overview
Zoom All 62
Zoom Dynamic 65
Zoom In 63
Zoom Out 64

© Earthquake Solutions, 2007
www.EQSOLS.com