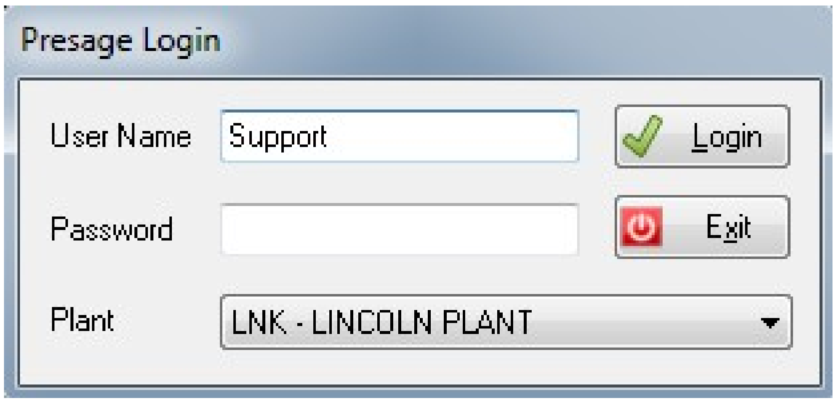
Table of Contents

QuickStart

The QuickStart section of the Presage User's Manual provides a quick overview of the many valuable functions Presage performs. Click *Help* on the top of the screen, then select *Manual* from the drop-down list to access the *Presage User's Manual*. Detailed instructions and descriptions of the features in Presage follow the *QuickStart* section.

QuickStart Logging In

The first screen on Presage is the Login box.



Presage will automatically insert the Windows user name into the user name field. Contact Presage support representative at 1 (800) 309-1704 to set up first user name and password. Then, click **Login** or hit the *Enter* key to login to Presage.

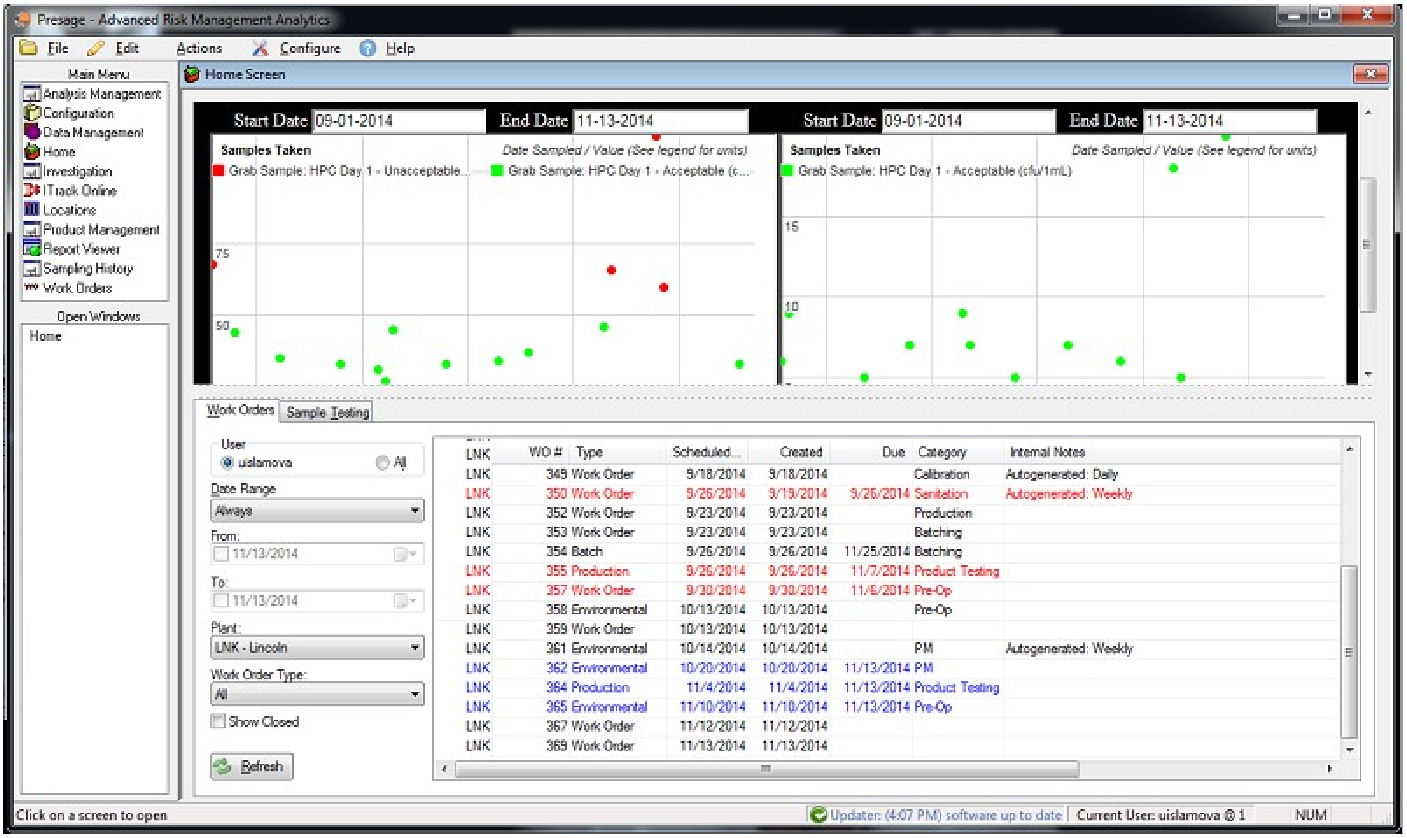
Tip: Password is case sensitive, but the username is NOT case sensitive.

Note: Additional users can be created and modified by following the instructions in the *Configuration Screen: Users Tab*.

QuickStart Main Menu

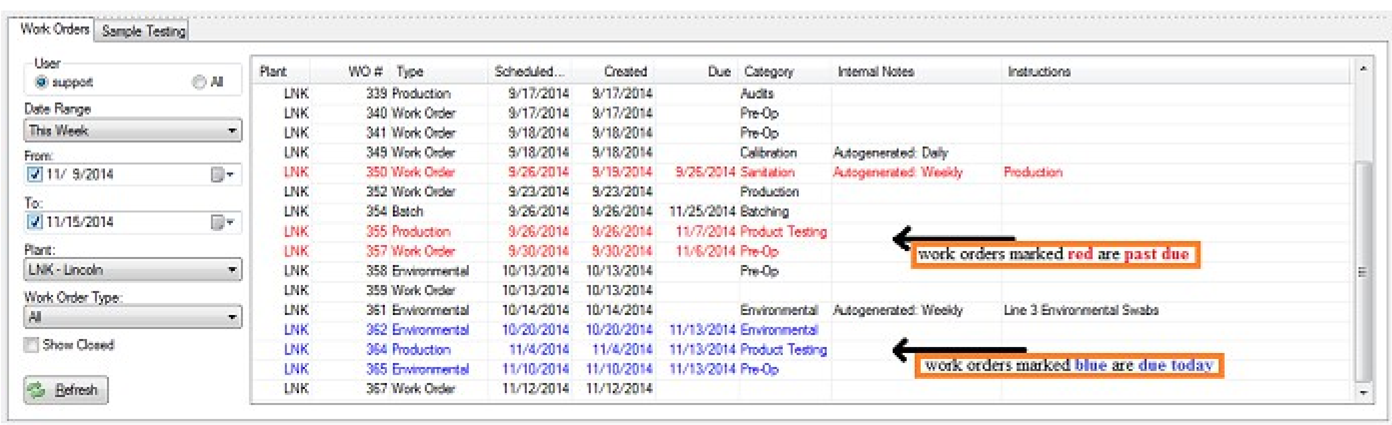
The Main Menu is the command center. The different screens in Presage are available in the *Main Menu*, located on the left side of the screen.

Below the *Main Menu* is *Open Windows*. Every time a new screen is opened from the *Main Menu*, it will show up under *Open Windows*. With this list, it is easy to toggle back and forth between opened screens without losing changes.

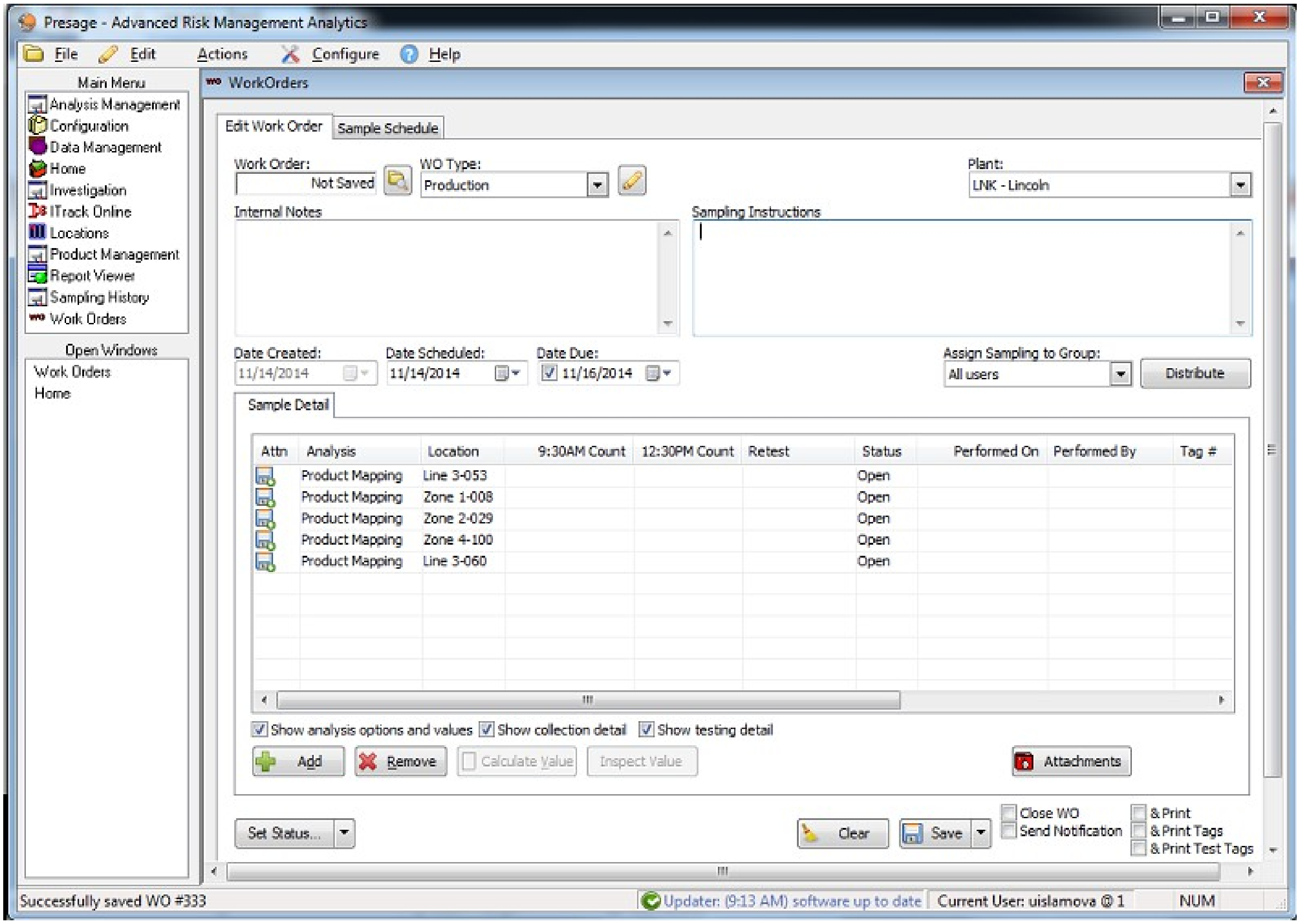


QuickStart Entering Analysis Results

Pending work orders are listed in the *Home* screen. To open the work order, double-click on it. Search for other work orders using the search panel under the same tab.



The work order will open up under the *Work Order Screen: Edit Work Orders Tab*. Enter analyses results under the option name columns. Depending on the analysis options assigned in the *Analyses Tab*, results are entered by checking a check box, typing in, or selecting from a drop-down list.



Click **Save**. There is an option to print and/or close the work order when saving, check the box(es) when appropriate.

Home Screen

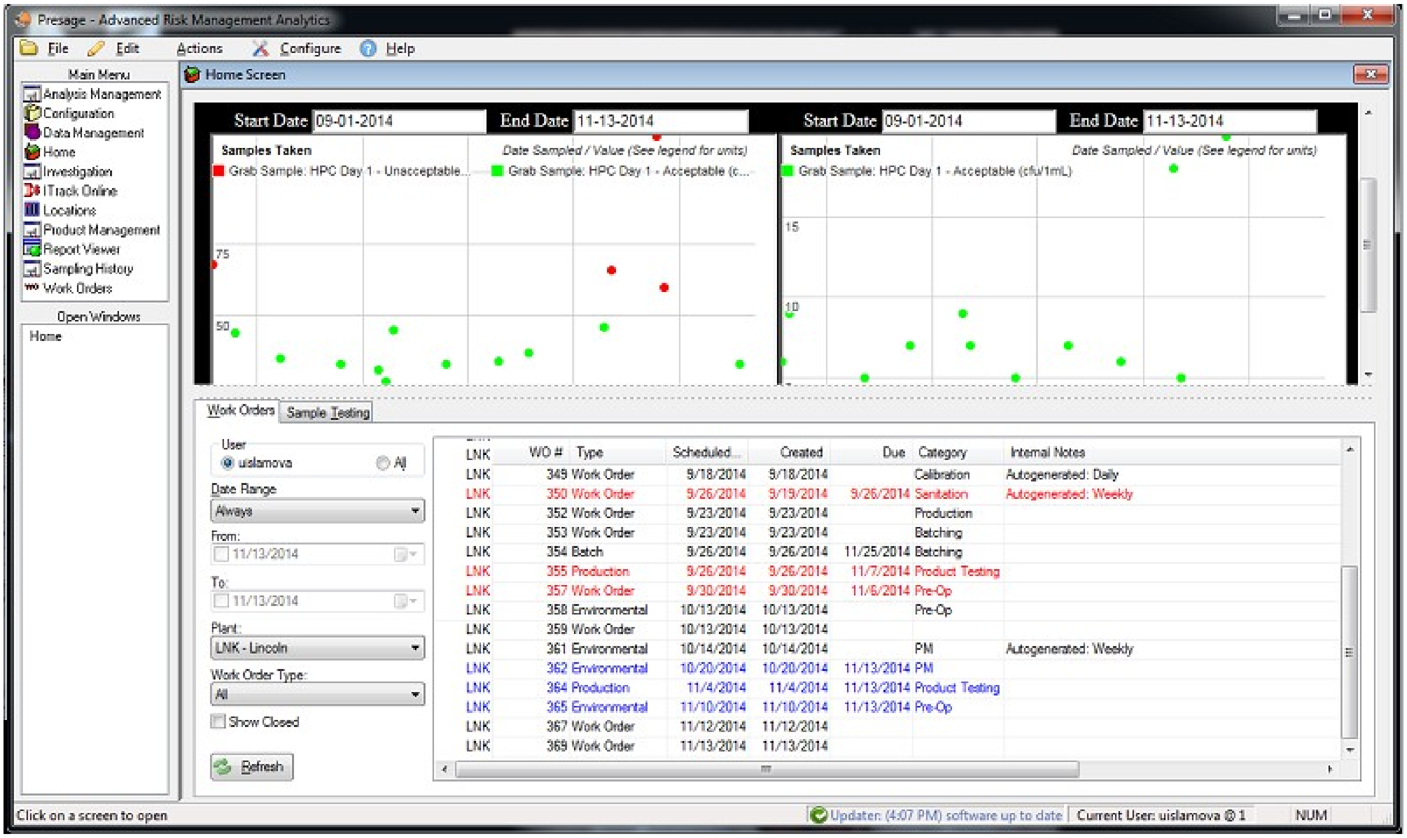
The *Main Menu* is the command center. All of the different screens for Presage are available in the *Main Menu*, located on the upper left side of the screen.

Below the *Main Menu* is *Open Windows*. Every time a new screen is opened from the *Main Menu*, it will show up under the *Open Windows*. With this list, it is easy to toggle back and forth between opened screens without losing changes.

Users can easily switch from one open window to another by clicking on it or by using the **Ctrl** and **Tab** keys on the keyboard.

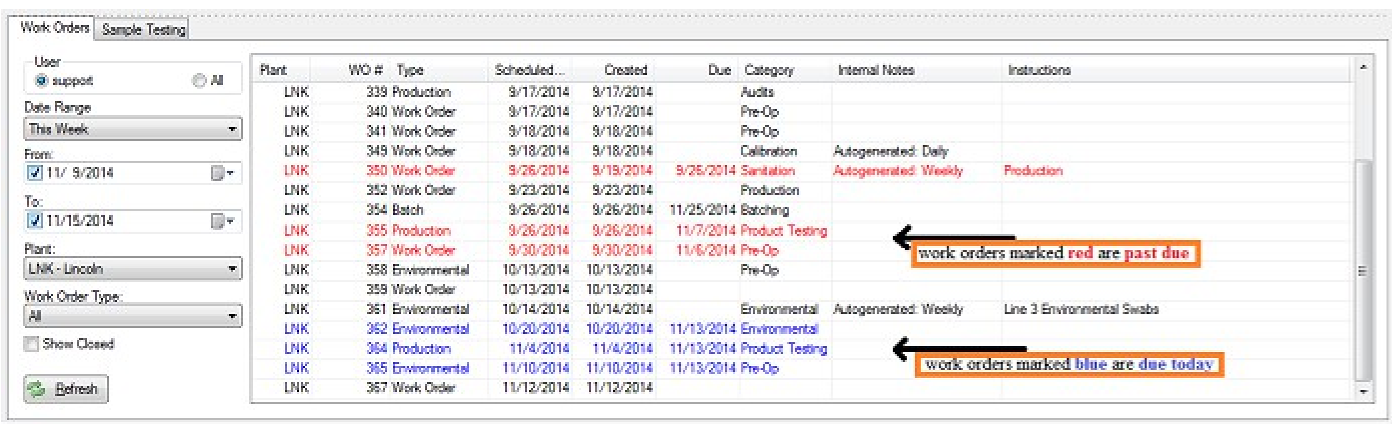
The top box where the Presage logo is a separate browser. There is an option to set a page with often-viewed graphs for easy monitoring. There is also an option to set any other browser page in here.

The bottom box is the work order queue. The panel on the left is a way to filter work orders, both open and closed.



Work Orders Tab

The *Work Order* tab on the *Home Screen* lists all pending work orders. Work orders are scheduled to a specific plant, date, and Group. This list also specifies the date the work order was created, due date and the unique work order number. There is also a place for notes and instructions next to each work order which can be editted in the *Work Order* screen.



User

There is a menu to the left of the display panel in the *Work Orders* tab. *User* is the login user.

Date Range

Date Range drop-down list shows time periods during which work orders were assigned. For example, selecting "Last Month" will list all work orders created in the previous month. There is an option to set the date range to *Custom*. For this option, users can set a date range by specifying the start and end date. To the right of the date box is a calendar icon for easier date selection.

We recommend to set the *Date Range* to *Always* to ensure all open work order are visible. It is best to close work orders once all data has been entered so this Work Order Queue is always up to date and only shows work orders awaiting to be filled, otherwise the list gets very long and it is difficult to find the work order user is looking for.

Colorization

The Work Order Queue is colorized based on *Due Date*. Red work orders are those with a expired due dates. Blue work orders are those with due dates expiring within 24 hours. Black work orders do not have a due date assigned.

Plants

Users assigned to multiple plants can select the plant from the drop-down to narrow down the work order list to one plant. Users assigned to only one plant will not have to worry about this step as the plant will default to the plant specified in the login page.

Work Order Type

The list of work orders can be filtered by *Work Order Type* which is right below the *Plant* drop-down.

Show Closed

By checking the *Show Closed* box, users can view all closed in addition to the open work orders within the set date range.

Refresh

Once a work order is created, open the *Home Screen* and click *Refresh* to see the new work order.

Log Out

On top panel in the *Home Screen*, click *File* then *Logout* once you are finished using the program. User is automatically logged out once the program is closed or if *Exit* is selected from the *File* drop-down.

Sample Testing Tab

Sample Testing tab is next to the *Work Orders* tab in the *Home Screen*. Its purpose and function is very similar to that of the *Work Order* tab. This screen lists the testing samples which have to be pulled out of the testing mode. For example, it lists when samples have to be pulled out of the incubator.

No actions have to be taken in this screen. Once the samples are scanned to indicate when they have been pulled out, these samples are taken off of this list.

Items in this list are color-coded:

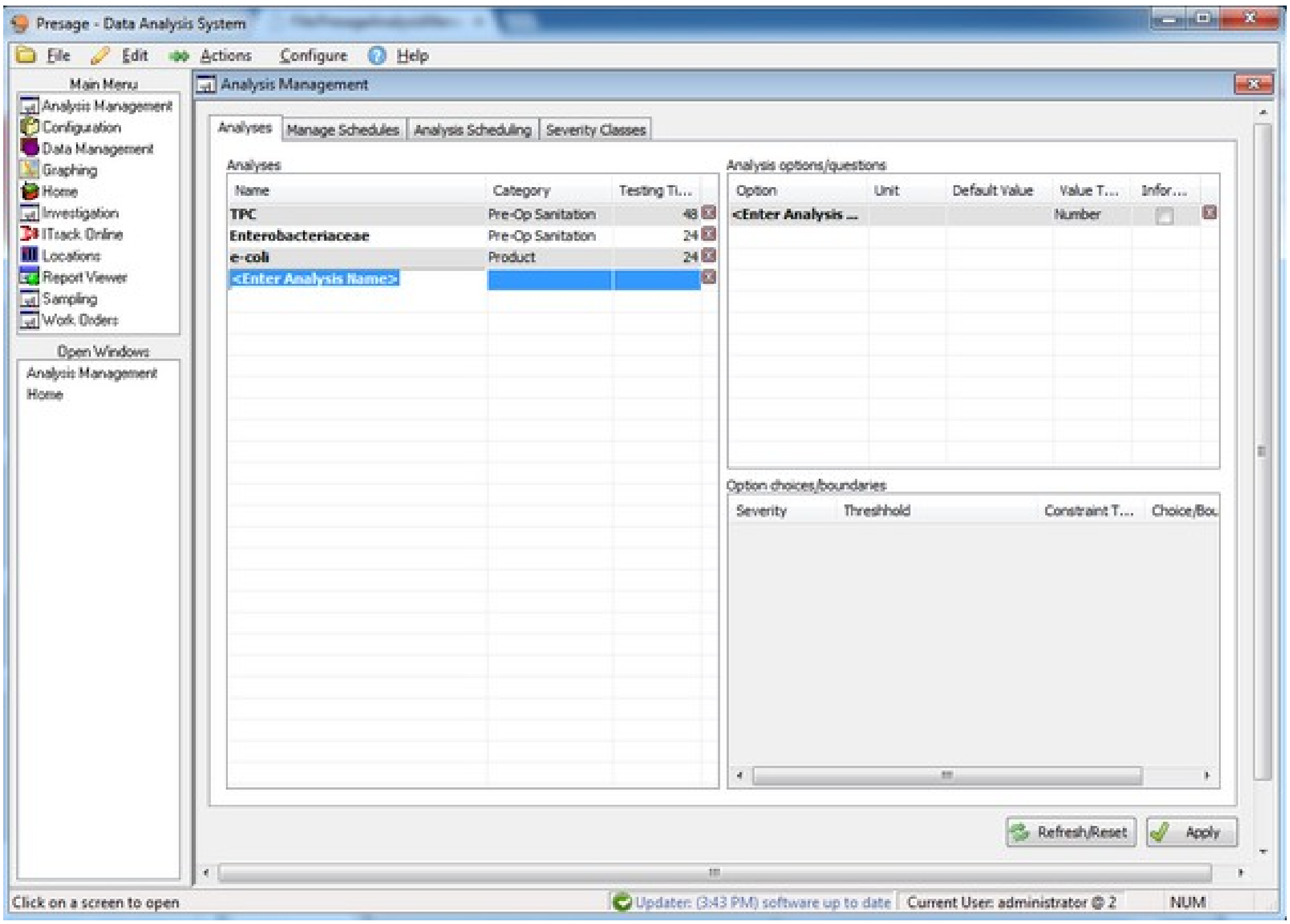
Dark grey if its been tested, Black if testing has not begun yet, Red if it should have been finished already but isn't, Blue if it should be finished 'soon' (currently in the next 24 hours), otherwise Green.

Analyses Tab

To create a new analysis, click on the box *<Enter Analysis Name>*. Enter the analysis name on the box provided. In the next column, type in the category name. This column is used to group analyses.

Testing Time is for lab use. Type in the number hours the sample must rest in the incubator.

Keep the analysis name selected before going to the next section, *Analysis options/questions*.



Under *Option*, enter what is being tested for; for example: plate count, sample area, detect-ability, etc.

When appropriate, enter the unit in the next column; for example: cfu, ppm, etc.

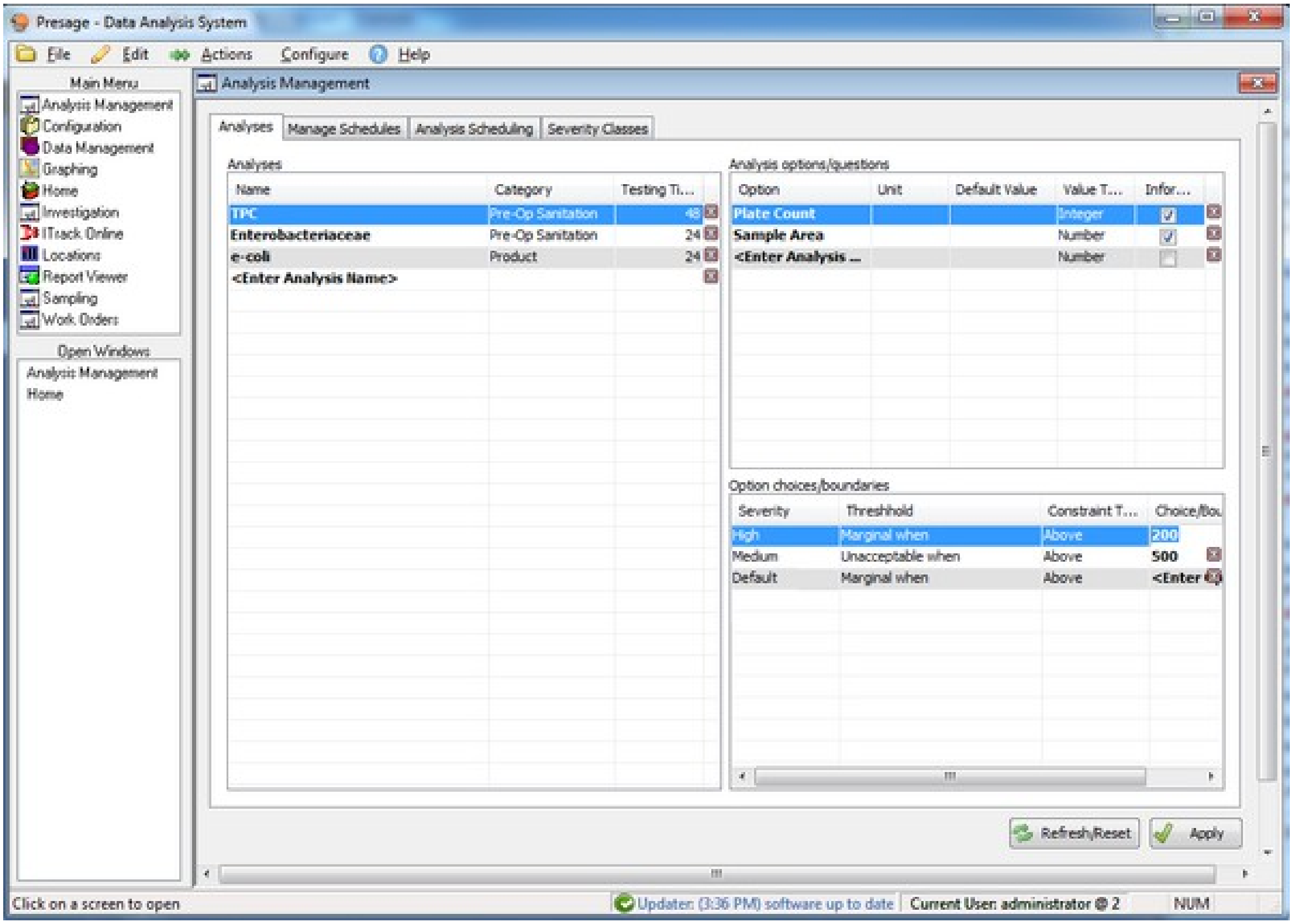
The **Informational** box is for options which don't need to be graphed. Check if appropriate.

Then choose the *Value Type* from the drop-down list provided. For example, plate count is measured in number, and qualitative analyses are measured in boolean: positive or negative. To create a drop-down list of various options to select from, choose *Choice*.

Keep analysis name and analysis option selected to go on to the next section: Option choices/boundaries. Set thresholds for each option separately.

First, select the severity from the drop-down list. To set up severity classes, refer to *Analysis Management: Severity Classes*.

For *Threshold*, select an option from the drop-down list. Marginal values are those which are acceptable, but are nearing the unacceptable boundary. Unacceptable values represent failure. Invalid values are those which have no value or validity.



Constraint Type sets the minimum and maximum boundary value. Choose the appropriate *Choice/Boundary Value*, or fill it in manually. For example, to set 10 cfu as a maximum acceptable limit for an analysis, choose *Unacceptable when* under *Thresholds*; *Above*under *Constraint Type*; and type in *10* under *Choice/Boundary*.

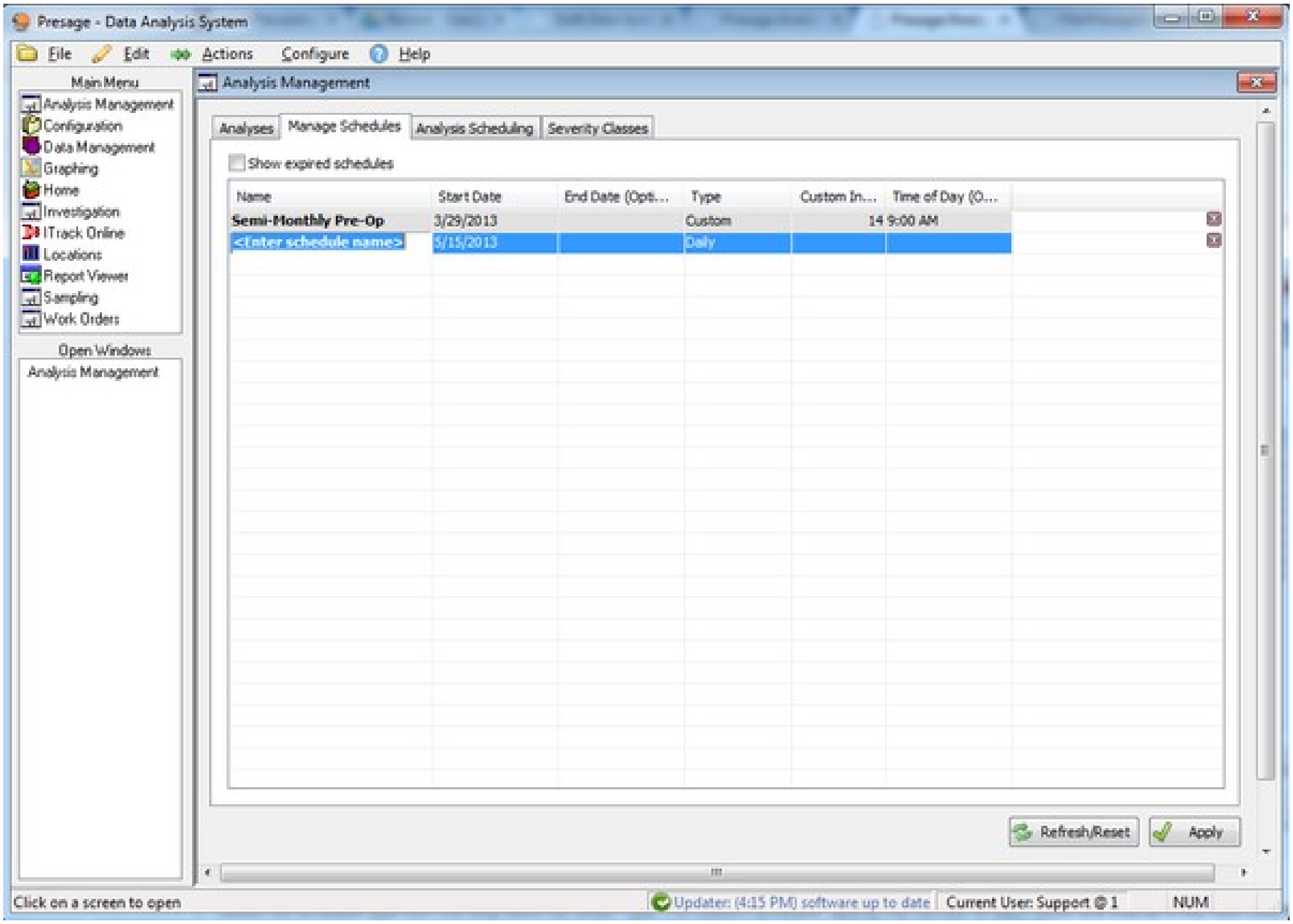
Click **Apply** to save changes.

Manage Schedules Tab

This is where schedule intervals are defined. Choose a name for the schedule that will best identify the interval (i.e. Mondays, Weekly, By Week's End, etc.). Make sure to give each schedule a unique name that best identifies it. Click on *<Enter Schedule Name>* for the cursor to appear; type in the new schedule title.

Enter a start date, usually today's date. End Date is optional. If the end date is not set, the schedule will continue indefinitely.

Select the schedule type from the drop-down list under Type. For custom schedule, specify the number of days within one period interval in the next column. In the final column, specify the time of day (optional) the test is scheduled for.

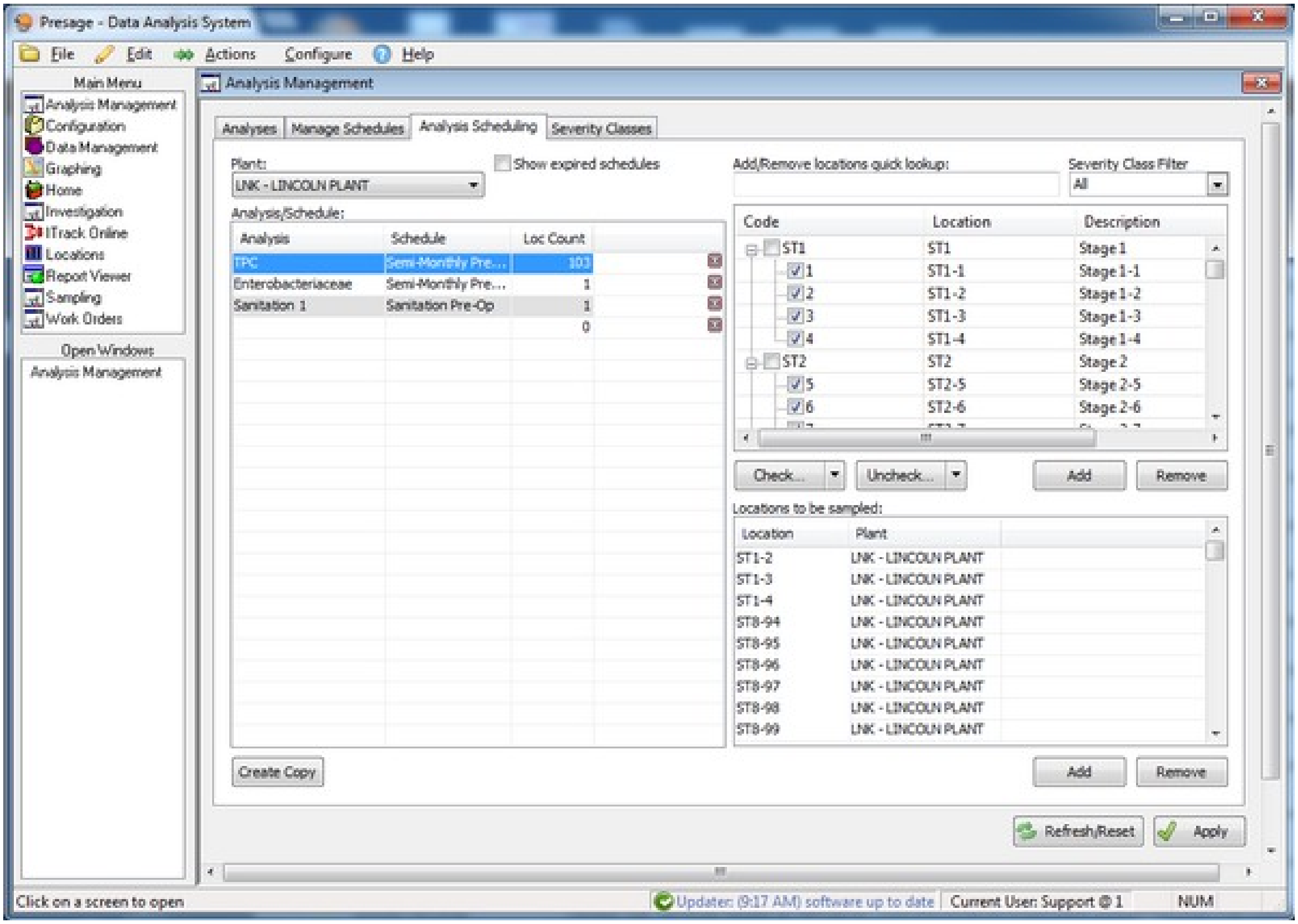


To delete a schedule, click on the corresponding red box with white X button on the far right of each line. Click **Apply** to save changes.

Analysis Scheduling Tab

Use this screen to build an analysis schedule for each plant. First select the plant from the drop-down list.

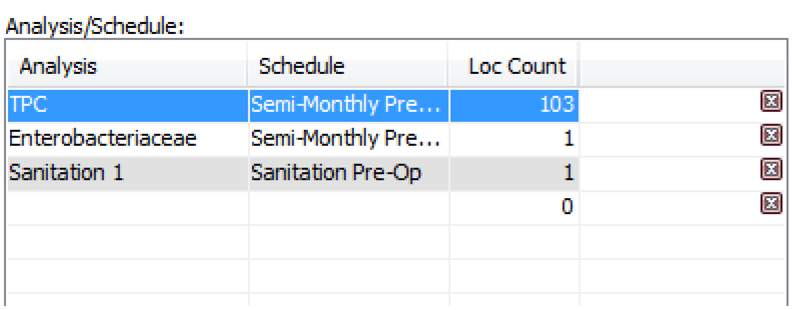
Check the box next to *Show expired schedules* to see all expired, in addition to current, schedules.



Under the column title *Analysis*, select the analysis type from the drop-down list. This list includes all analyses created under *Analysis Management: Analyses*.

Select the schedule period interval from the drop-down list under Schedule. This list is created under *Analysis Management: Manage Schedules*.

The *Location Count* is automatically filled out when the locations are selected in the next step.



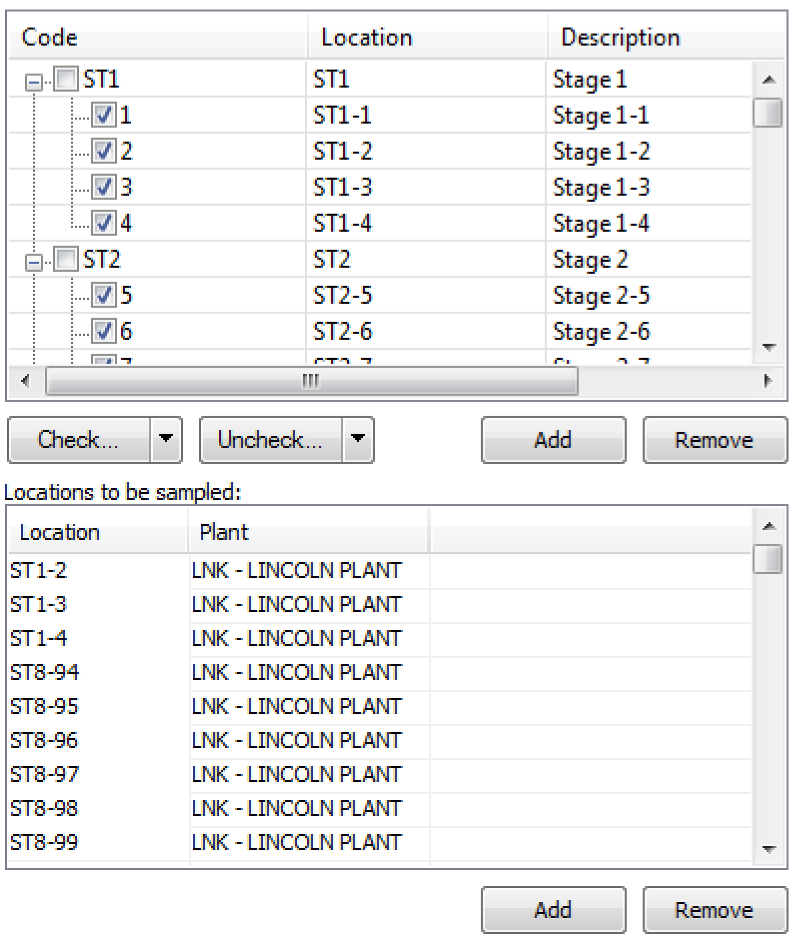
Analysis/Schedule:

Analysis	Schedule	Loc Count
TPC	Semi-Monthly Pre…	103
Enterobacteriaceae	Semi-Monthly Pre…	1
Sanitation 1	Sanitation Pre-Op	1
		0

The right side of the screen is for choosing locations to perform the analyses on. Select one analysis under *Analysis/Schedule:*. Then check the box next to the locations which need to be sampled.

For a quick look-up, type in the first letter or number of the display code for the location. The corresponding locations will show up in the display below. The list can also be filtered by severity class. Choose the severity class from the drop-down; the list in the display will narrow down to locations with the severity class specified.

Use the **Check...** button to select groups of locations. Use the **Uncheck...** button to deselect groups of locations. Click **Add** (on the middle of the screen, not the one on the lower portion of the screen) to add the selected locations to the list of locations to be sampled. Click **Remove** to take out locations from the list.

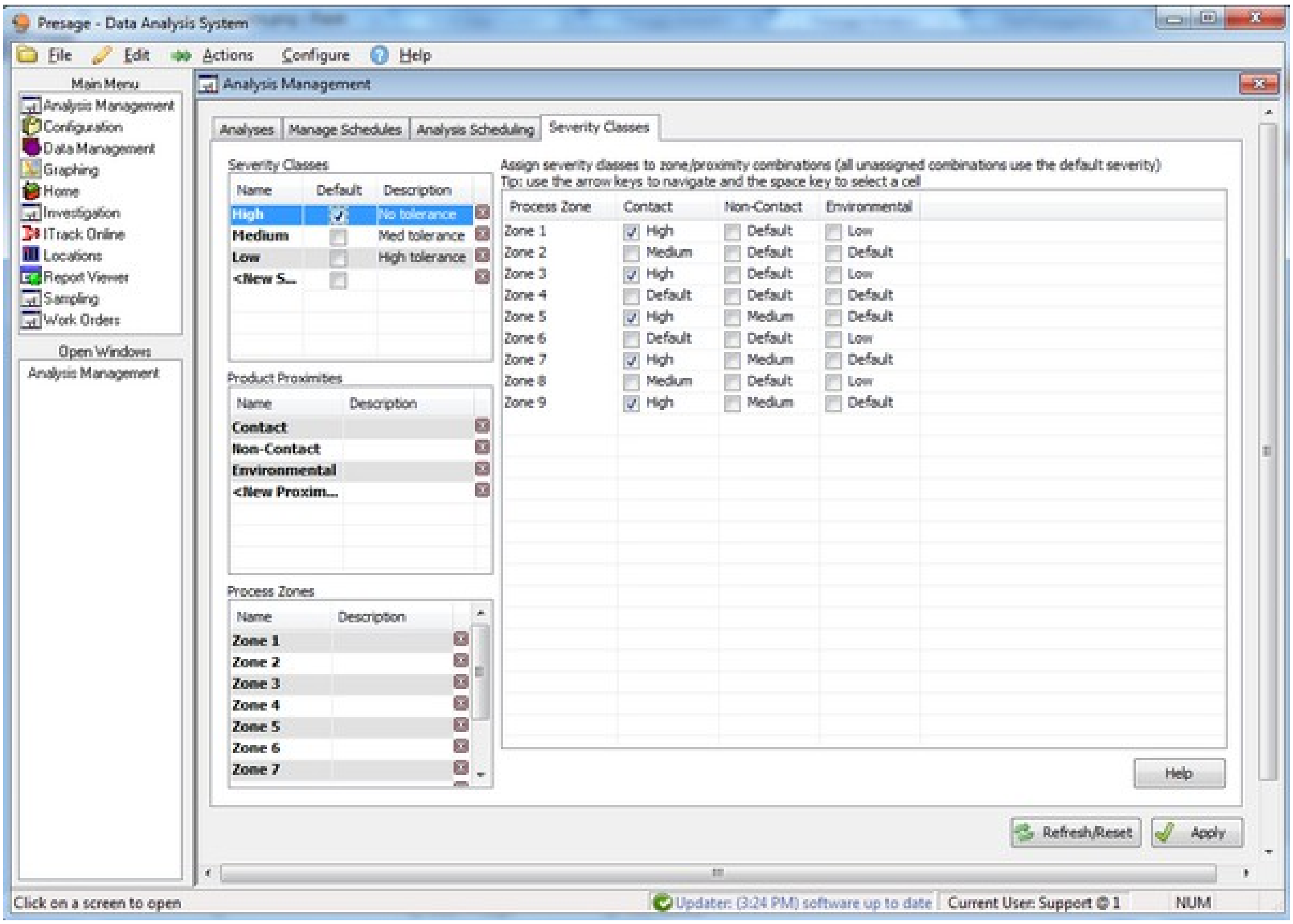


To add locations one at a time, use the **Add** button on the lower portion of the screen under *Locations to be sampled:*. To remove a location, select a location from the list and click **Remove**. To remove multiple locations, hold down the *Ctrl* button on keyboard while selecting multiple locations, then click **Remove**.

Click **Apply** to save any changes and use the **Refresh/Reset** button to remove any unsaved changes.

Severity Classes Tab

This screen is for assigning severity classes to zone/proximity combinations.



First give a unique names to all severity classes. Severity classes are based on limits which are set based on product proximities and process zones. Examples are High, Medium, and Low; or 10, 100, 500. Use terminology or numbers which best fit your system.

Click on the box under *Default* (may have to expand the column to see the full column title), to set a default. Default severity class is usually the one used the most often.

Product proximity refers to contact or non-contact surfaces. To add new product proximities, click on *<New Proximity>*. Type in the new proximity title. To remove product proximity, click on the red box with a white X to the right of each line. To rename, select the appropriate row and type in the new proximity name.

Process zones are different operational zones such as Chilling or Packaging. To add new zones, click on *<New Zone>* (It may be necessary to scroll down to see this row.) To delete a zone, click on the red box with white X to the right of each line. To rename, select the appropriate row and type in the new zone name.

To assign severity classes to zone/proximity combinations, click on one severity class under *Severity Classes* on the left display to highlight it, then check the boxes where the selected severity class is appropriate on the right display.

Process Zone	Contact	Non-Contact	Environmental
Zone 1	☑ High	☐ Default	☐ Low
Zone 2	☐ Medium	☐ Default	☐ Default
Zone 3	☑ High	☐ Default	☐ Low
Zone 4	☐ Default	☐ Default	☐ Default
Zone 5	☑ High	☐ Medium	☐ Default
Zone 6	☐ Default	☐ Default	☐ Low
Zone 7	☑ High	☐ Medium	☐ Default
Zone 8	☐ Medium	☐ Default	☐ Low
Zone 9	☑ High	☐ Medium	☐ Default

Now, click the next severity class to highlight it, then check the boxes where this severity class is appropriate on the right display. Not all boxes have to be checked.

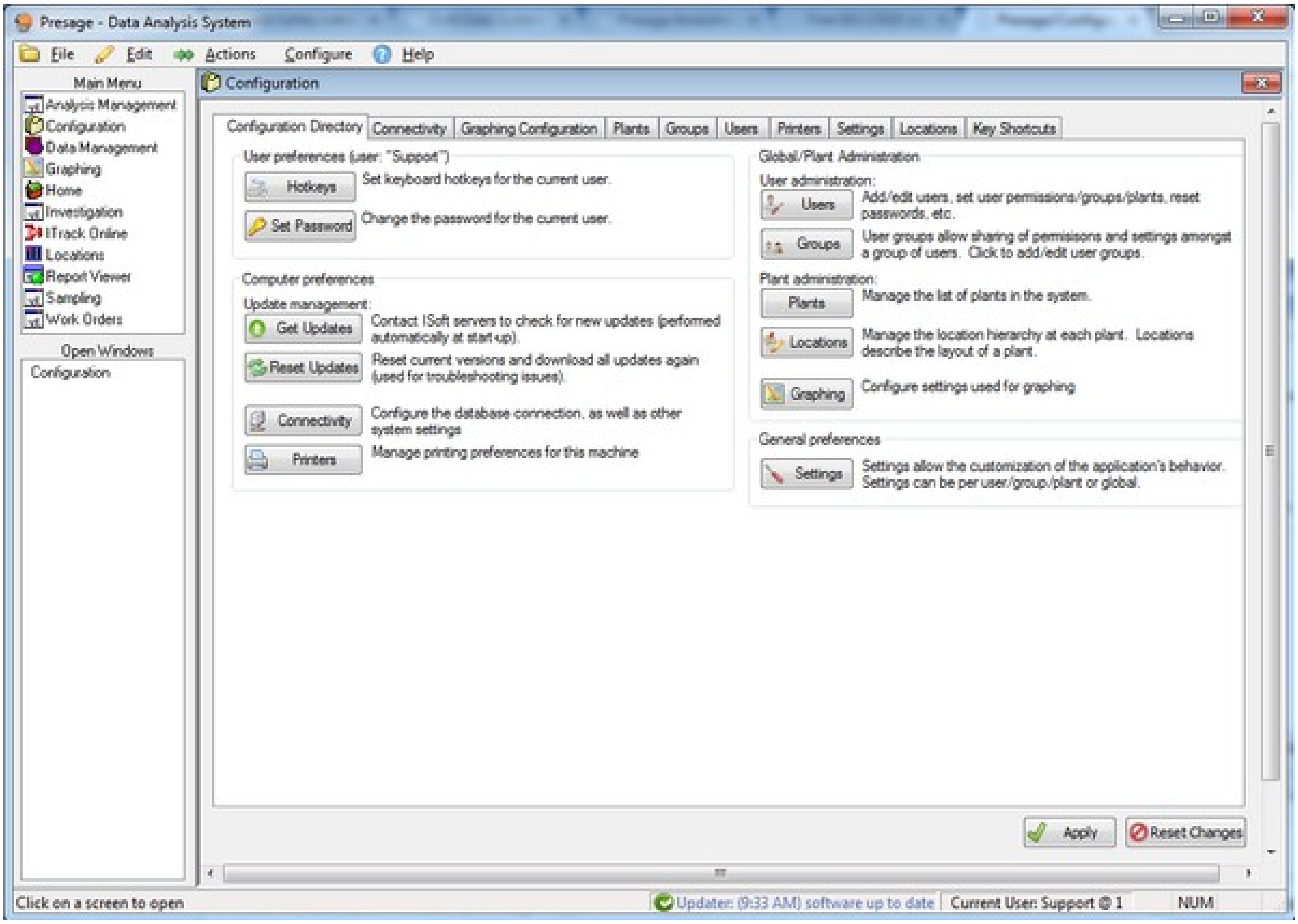
Click **Apply** to save the changes.

Configuration Screen

The Configuration Screen is used to configure a variety of different functions within Presage.

Configuration Directory Tab

This screen is used as a reference. The buttons here are shortcuts to the rest of the tabs in the *Configuration Screen*.

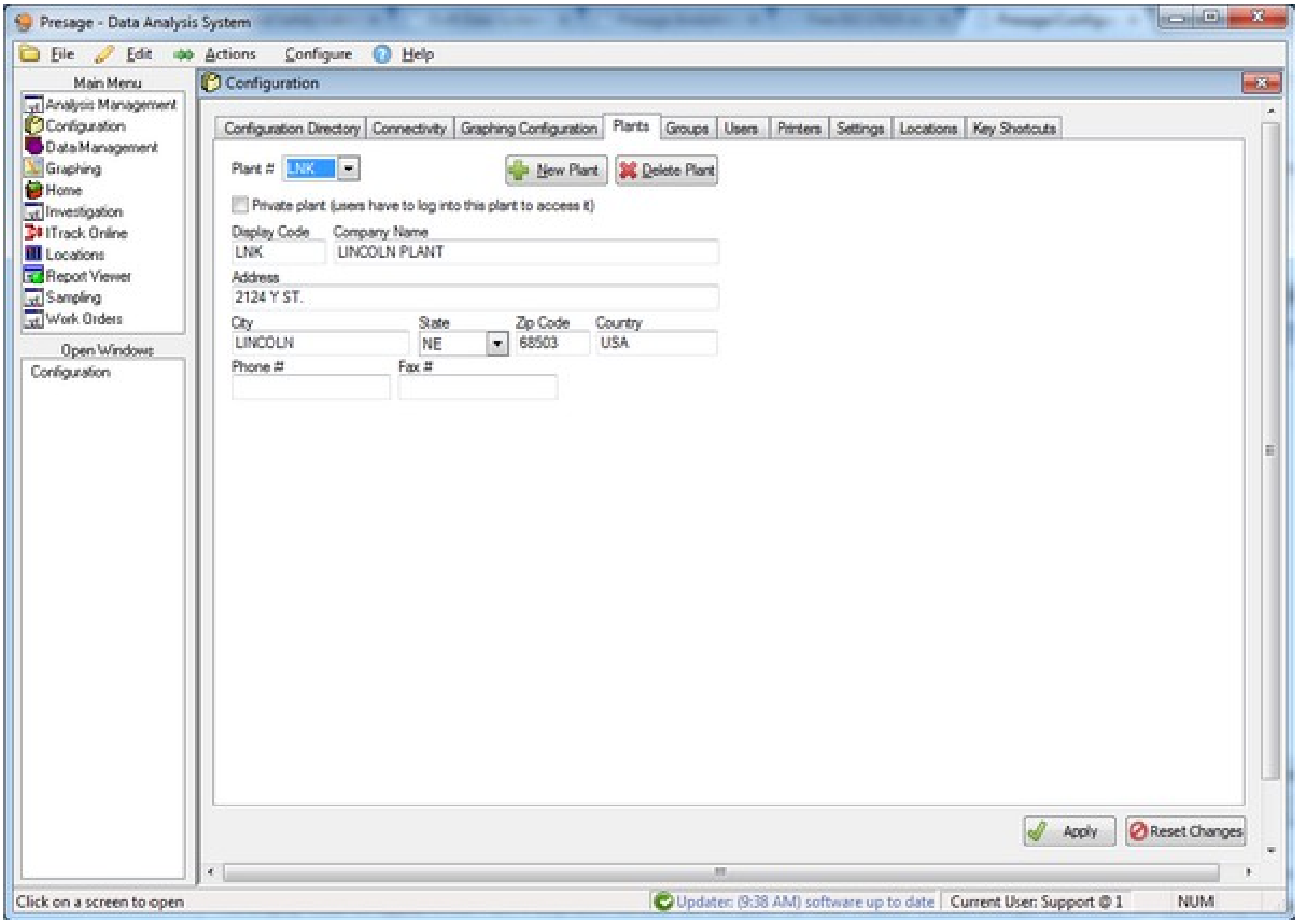


Connectivity Tab

This tab will usually be filled out by Presage Support. When appropriate, fill out the information in the space provided and click **Apply** to save the changes.

Plants Tab

Use *Plants* tab to add and delete plants.



To add a plant, click **New Plant**. For *Plan #*, select *New* from the drop-down list. If the new plant is private, check the box provided. With this option, users must log in to have access to the new plant.

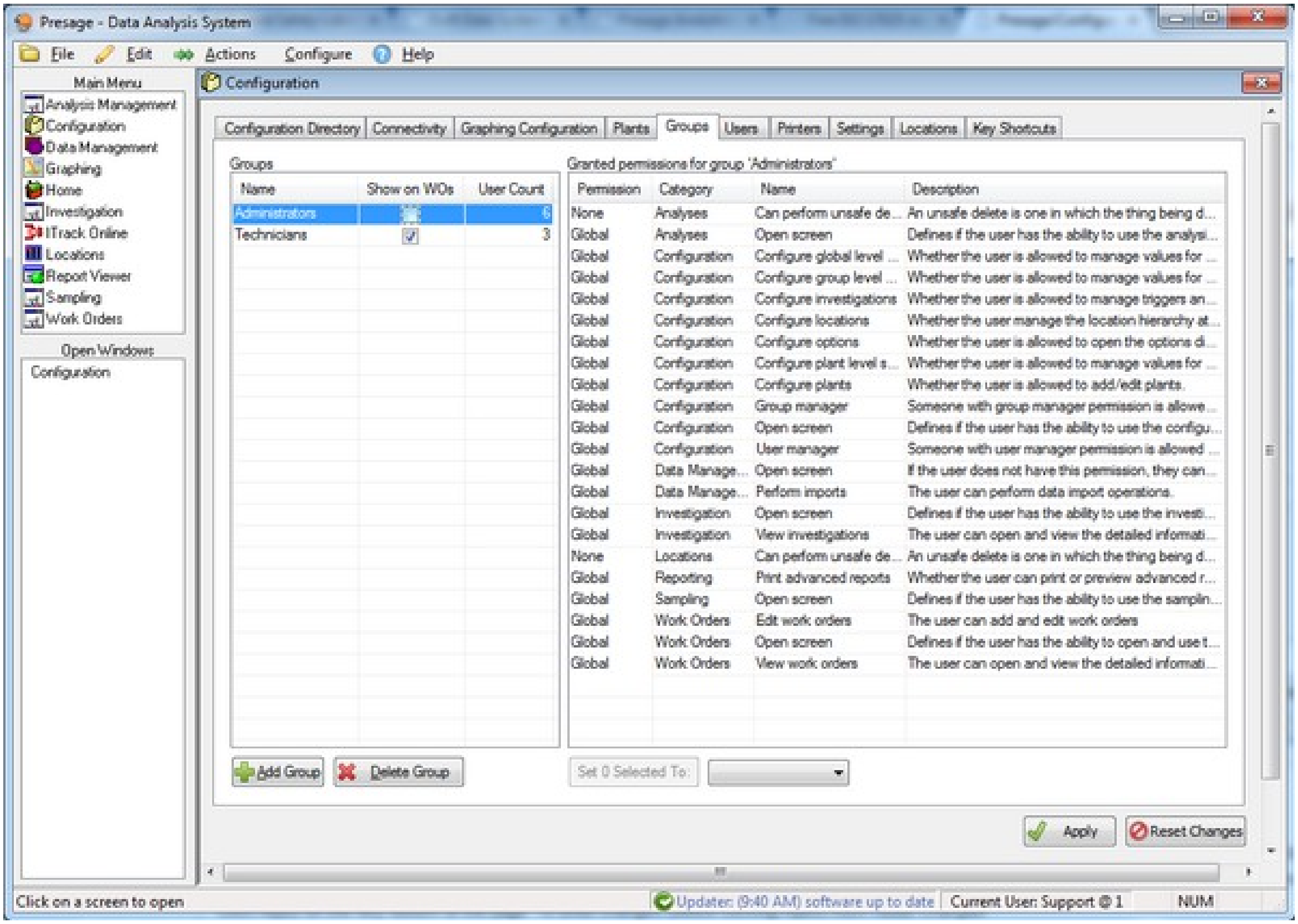
The *Display Code* will appear in the *Plant #* drop-down list after the new plant has been created. Choose a unique code which will best identify the plant.

Company Name refers to the business name a particular plant is associated with. Fill in all information in the spaces provided.

To delete a plant, choose the plant from the *Plant #* drop-down list then click **Delete Plant**.

Groups Tab

Use Groups tab to add, delete, and set permissions to groups.



To add a group, click **Add Group**. Type in group name in the pop-up box, then click **OK**. The new group will show up under *Groups list*.

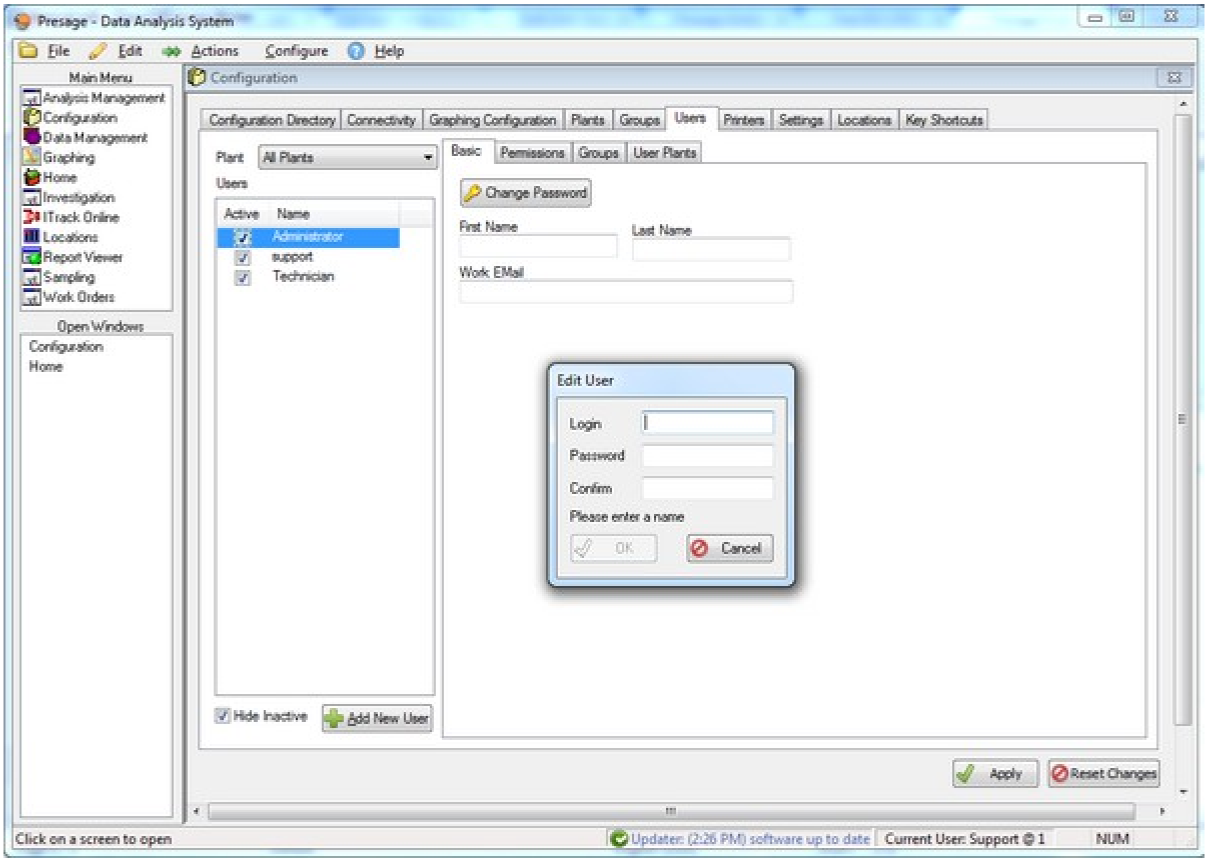
To delete a group, select the group name, then click **Delete Group**.

To set permissions to a group, select the group name, then select the permission setting from the drop-down list under *Grant* next to each category. If one permission level is appropriate for multiple categories, then select those categories while holding down the *Ctrl* key and select permission setting from the drop-down list next to **Set # Selected To:** on the lower portion of the screen. Click updated **Set # Selected To:**.

Click **Apply** on the very bottom of the page. To undo changes (before clicking **Apply**) click **Reset Changes**.

Users Tab

To add a new user, click **Add User**. Enter login name and password, and re-enter password to confirm in the pop-up box. Click **OK**.

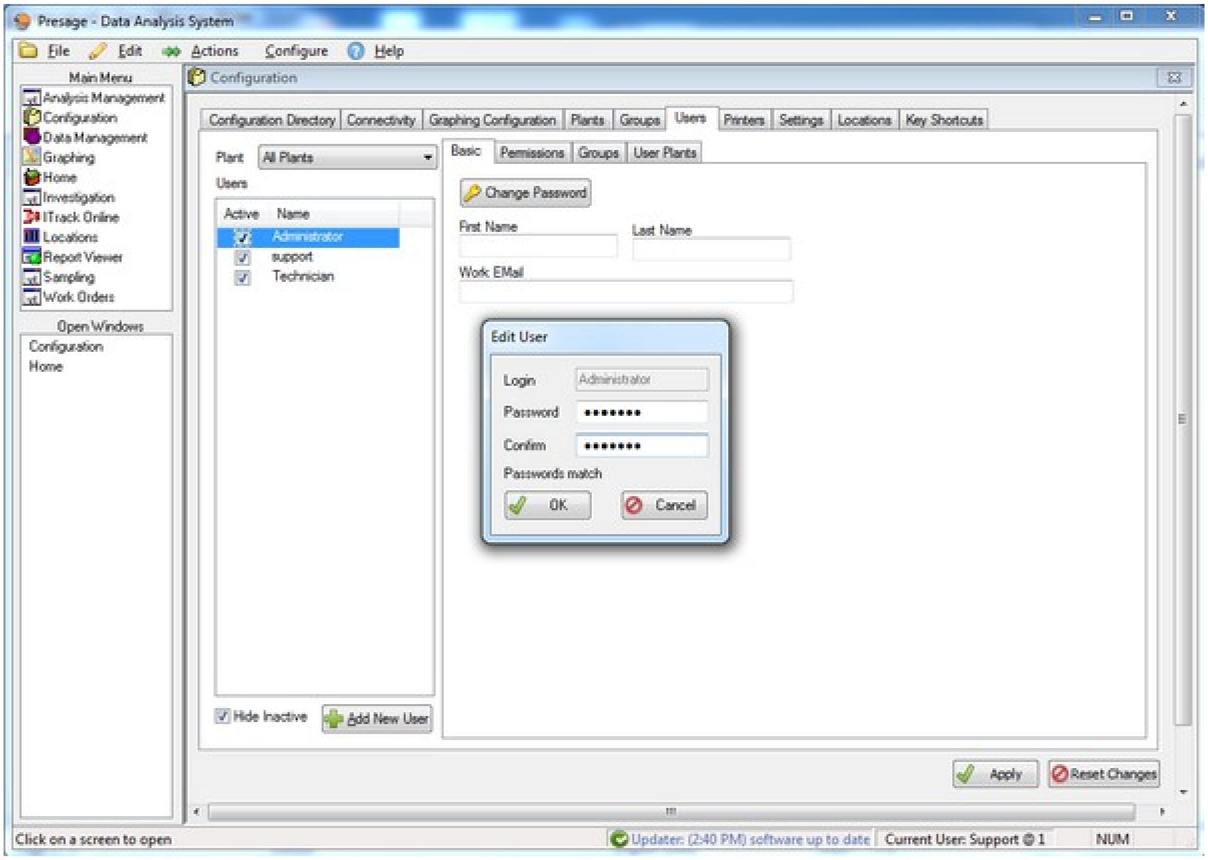


To inactivate users, simply uncheck the box next to the users' names. Check *Hide Inactive* box to take inactive users out of the users list. This option does not delete users' names from the list, only hides them.

Users tab has four sub-tabs: Basic, Permissions, Groups, and User Plants.

Basic Sub-Tab

To change login password, go to the *Basic* tab and click **Change Password**. Enter password. Reenter password in the next box to confirm. Click **OK**.



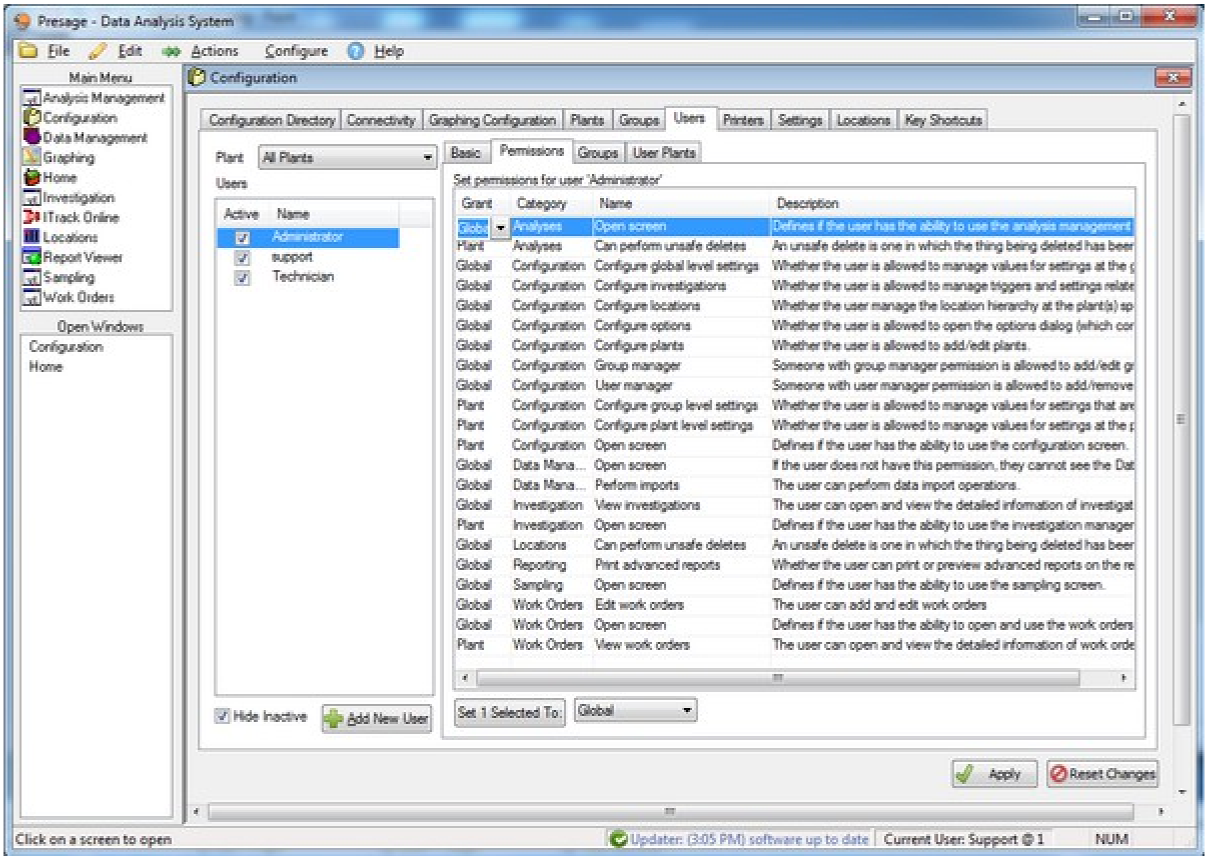
Permissions Sub-Tab

To set or change permissions for a user, click on *Permissions* sub-tab.

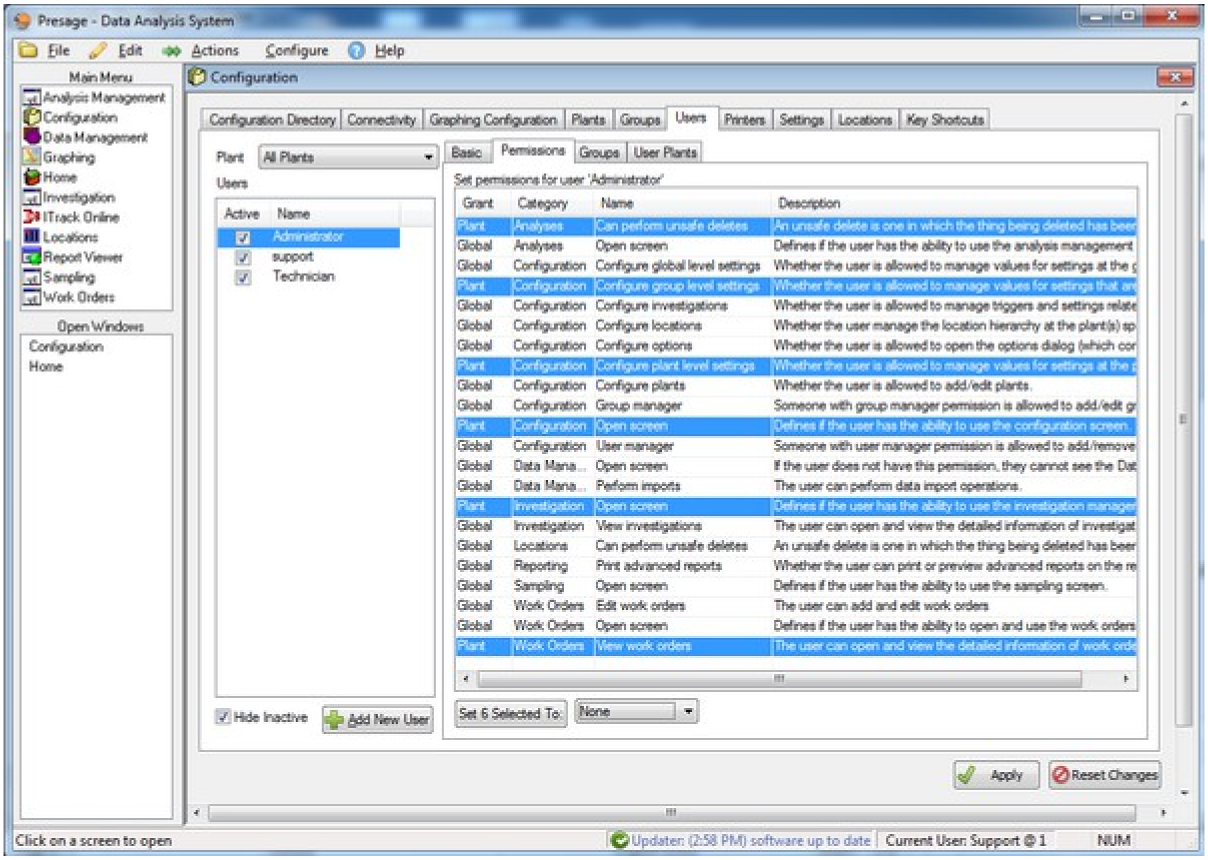
Click on the user's name to highlight it.

To change permission settings individually for each category, click on each row in the *Grant* column, a drop-down list of permission settings will occur. Make the desired selection.

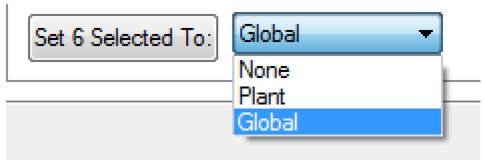
Click **Reset Changes** to undo changes. Click **Apply** to save changes.



To change permissions for multiple categories at once, hold down the Ctrl key and select desired rows.



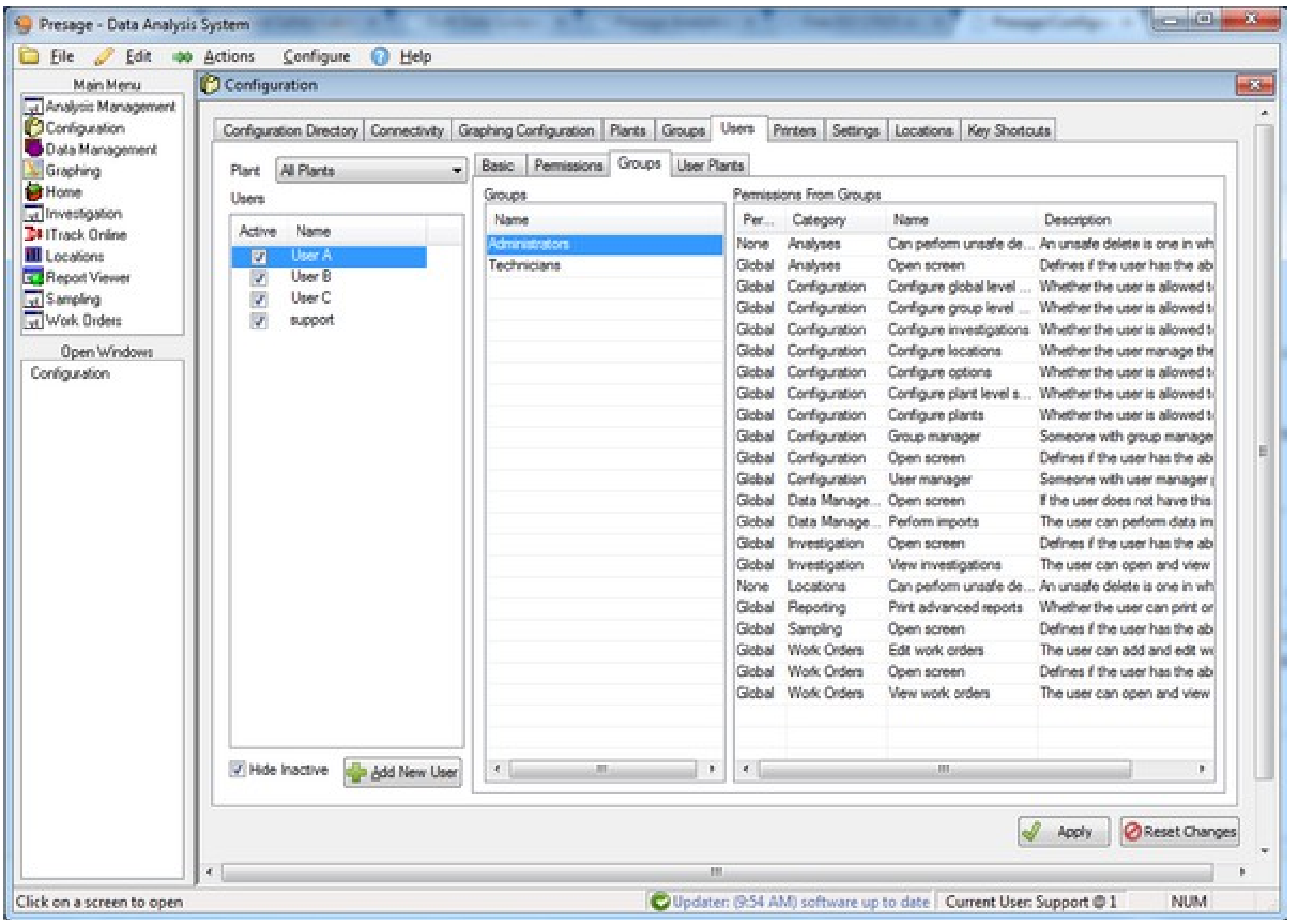
The first button directly below the permissions list will update the number of rows selected. Click on the second button, a drop-down list with permission settings will appear. Make the desired selection, then click the updated **Set # Selected To:** button to left of it.



Click **Reset Changes** to undo changes. Click **Apply** to save changes.

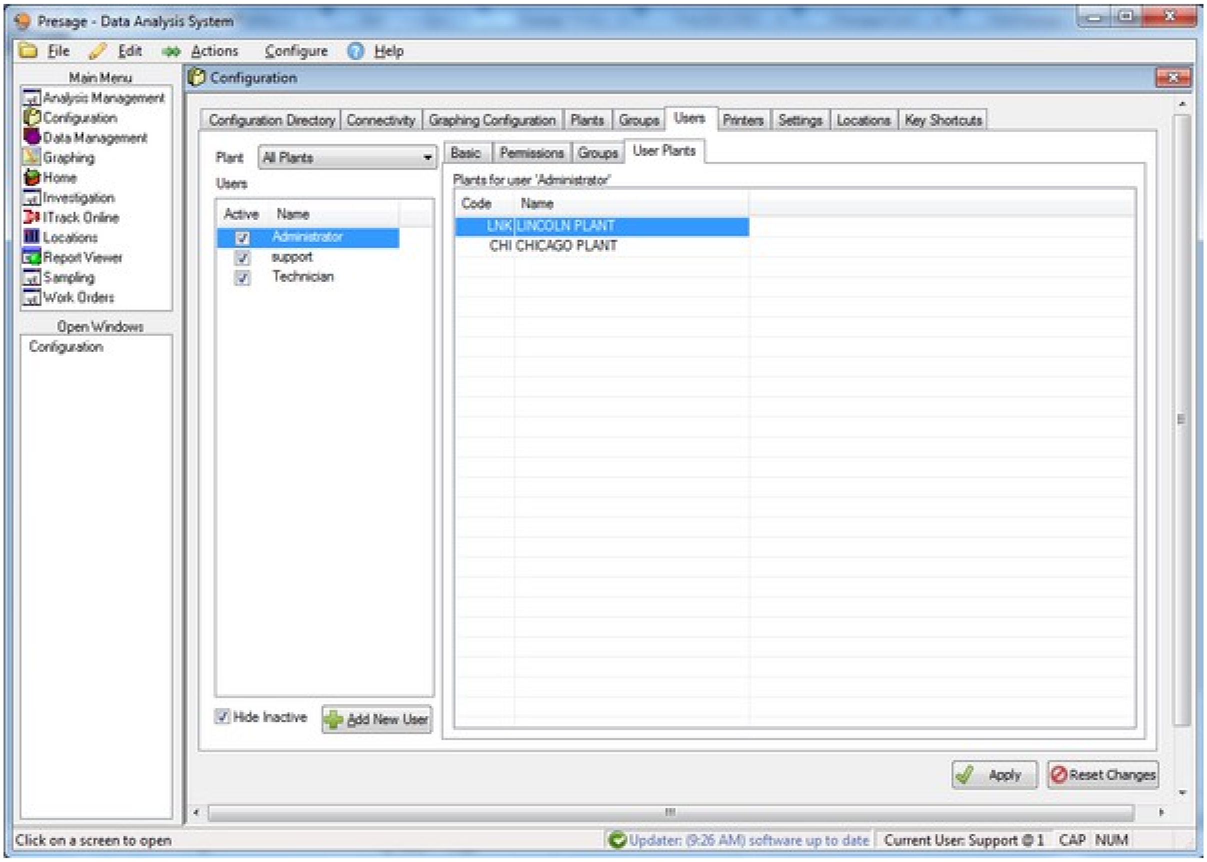
Groups Sub-Tab

To assign users to groups, simply click on the user name, then select the group name(s). Click **Apply**.



User Plants Sub-Tab

To assign users to plants, simply click on the user name, then select the plant name(s). Click **Apply**.



Printers Tab

Report Type drop-down list includes reports with various dimensions used by the plant. Select one.

Printer Name is the printer name (identification code) used to print the report. Select the appropriate printer used for the report dimensions selected.

The *Driver* and *Port* information will be filled automatically.

Click **Apply**. To undo any changes made before clicking **Apply**, use the **Reset Changes** button.

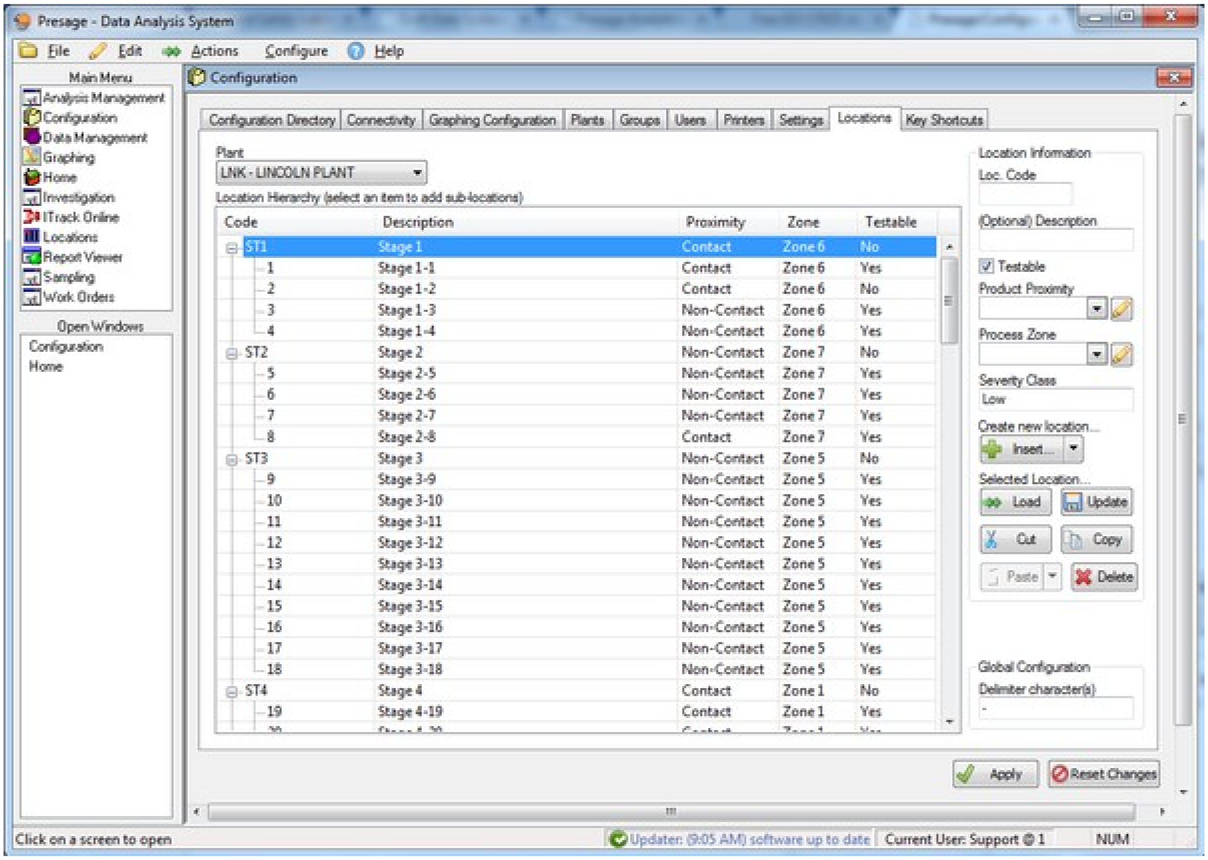
If there are multiple report types, follow the steps above to set appropriate Printer Name to other reports. Click **Apply** after each setting.

Settings Tab

This screen will be used initially to customize the software to the needs of the company. If need be, simply make changes by typing them into the *Value* column.

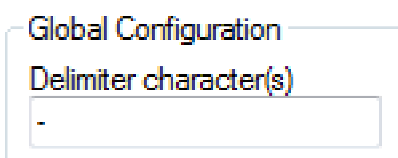
Locations Tab

To add locations or make changes to the location hierarchy, first select the appropriate plant from the *Plant* drop-down list.



Highlight an existing location from the *Location Hierarchy*, fill out the *Location Information* for the new location. Fill out the *Loc. Code* and *Description*, check the box if the new sight is testable, and select the appropriate product proximity and process zone from the drop-down lists. The *Severity Class* is filled automatically based on the process zone selected. Click on the arrow next to**Insert** and specify where in the hierarchy the new location belongs: parent or child of the location highlighted.

Note: Severity classes are assigned to product proximity and process zone combinations in the *Analysis Management: Severity Classes* screen.



Use **Load** button to bring up the *Location Information* of the location selected from the *Location Hierarchy*.

Use **Cut**, **Copy**, and **Paste** buttons to adjust the *Location Hierarchy*. Select a location, click **Cut**. Click on the arrow next to the **Paste** button, specify if the cut location needs to be pasted below or above the location selected in the *Location Hierarchy*.

To delete a location, select a location from the *Location Hierarchy* then click **Delete**.

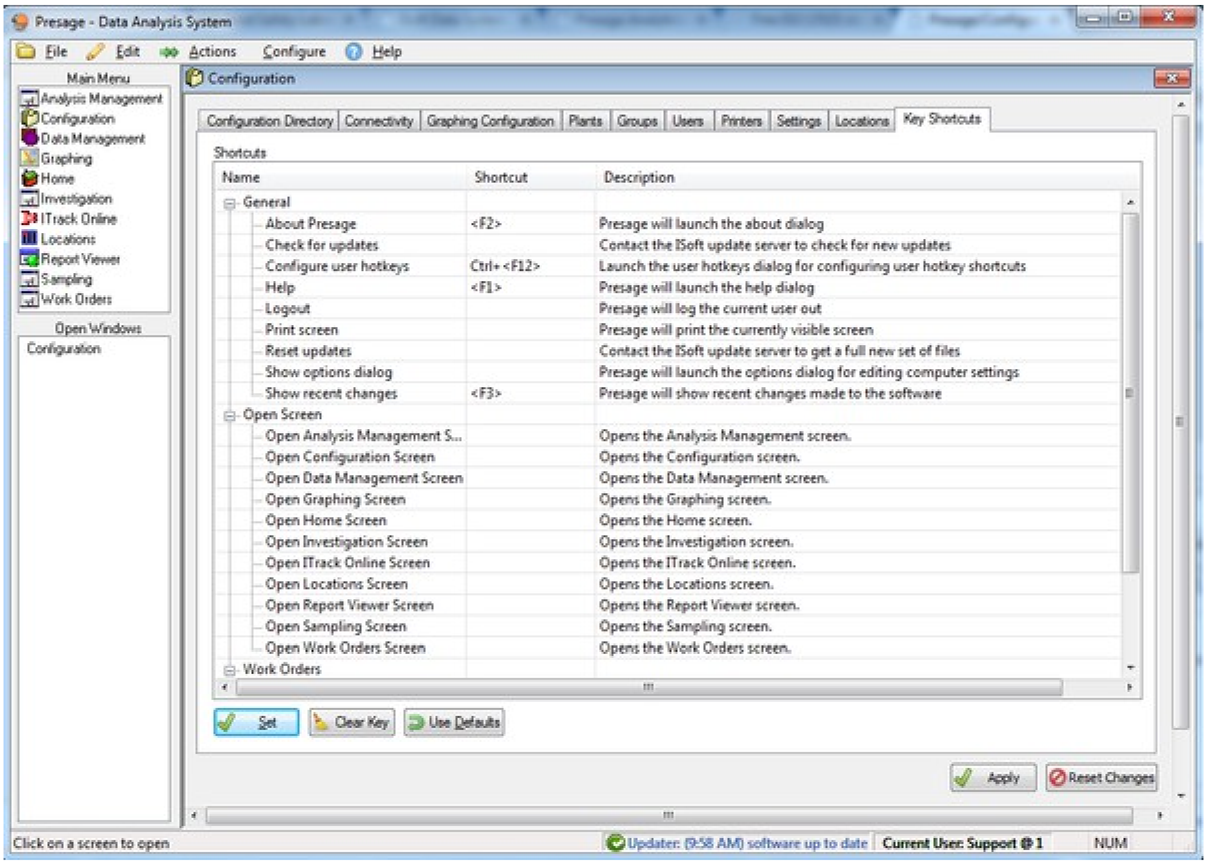
Click on **Update** to see the changes under the *Location Hierarchy*. This option does not save the changes made.

Click **Apply** to save changes. To undo a change (before clicking *Apply*) use the **Reset Changes** button.

The pencil buttons next to the *Product Proximity* and *Process Zone* are shortcuts to the *Severity Classes* tab under the *Analysis Management* window. Changes to the product proximity, process zones, and severity class combinations can be made in this screen.

Key Shortcuts Tab

This page is for reference and to create, change, and delete shortcuts. For example, setting F5 key as a shortcut for opening the *Analysis Management* screen enables the user to press the F5 key (while the Presage Software is open) to open the screen instead of using the mouse to click on the *Analysis Management* under the *Main Menu*. Any key can be used as a shortcut.



To set a shortcut, select one of the options under the column *Name* and click **Set**. A new pop-up window comes up. Press the key to be used as a shortcut, then click **OK**.

To delete a shortcut, select the one of the options under the column *Name* and click **Clear Key**.

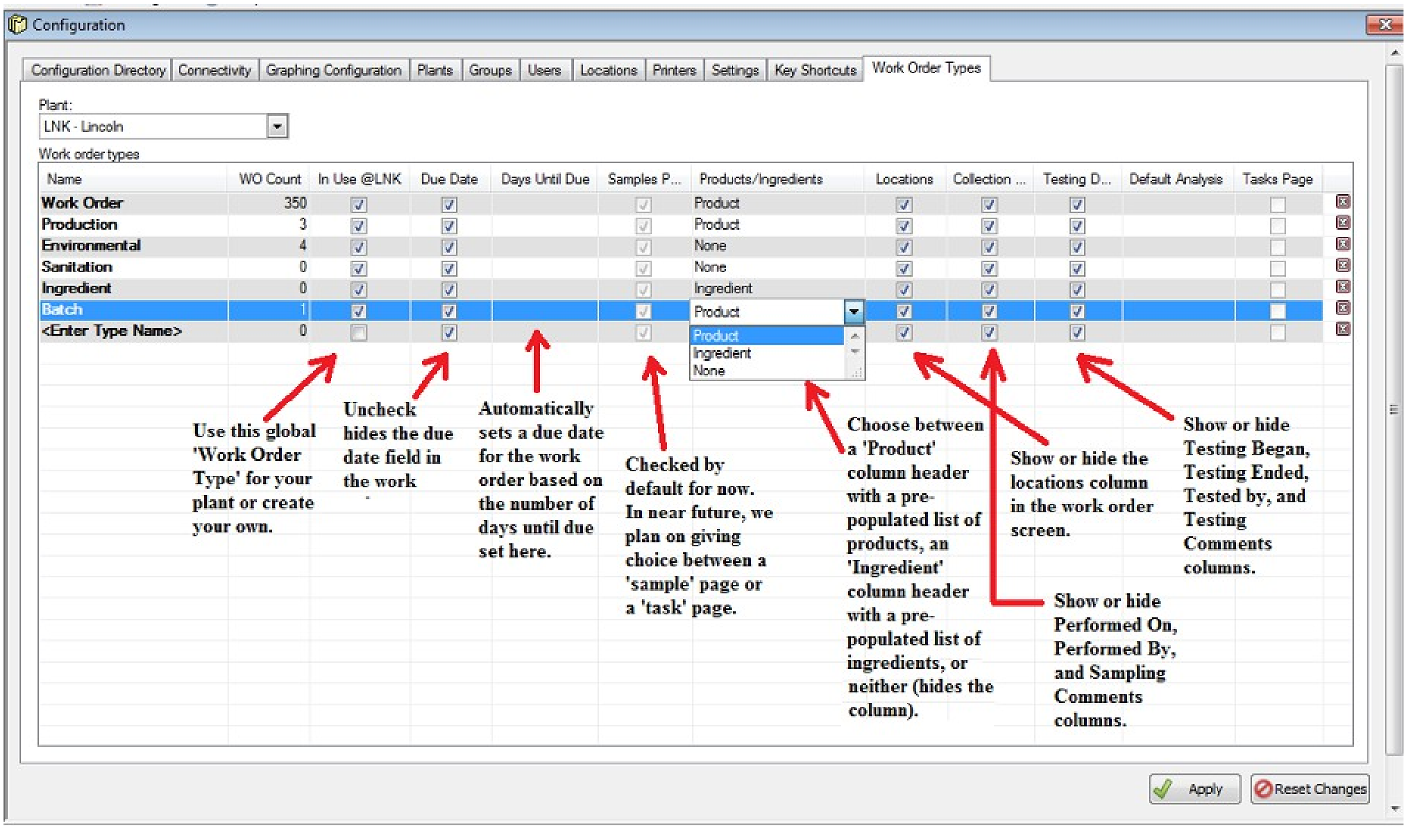
To use default shortcuts, select one of the options under *Name* and click **Use Defaults**.

For any changes made, click **Apply** to save. To undo changes (before clicking **Apply**), use the **Reset Changes** button.

Work Order Types Tab

Work order types can be created, edited, and assigned to a plant in this screen.

First pick a plant from the *Plant:* drop-down if your user is assigned to multiple plants.



To create a new work order type, click on the field with *<Enter Type Name>*. Type in what you would like to name the work order type.

The *WO Count* column is informational and nothing needs to be done to it.

Check the box under *In Use @(Plant Code)* to make this work order type visible to that plant.

Check the box under *Due Date* if this work order type will require a due date. *Days Until Due* is where you can specify a number for the number of days the due date will be after the date the work order is created. When a number is specified here, the Due Date is automatically filled in the *Work Order* screen for this type of work order.

Check "Samples Page *if this work order type will be for data collection (versus a task management).*

Choose to show the *Product* column OR *Ingredient* column OR neither. Selecting the *Product* will make that column appear in the *Work Order* screen with a pre-filled drop-down list of all products. Selecting *Ingredient* will do the same but with a list of all ingredients. Selecting *None* will hide the column.

Check *Locations* if you need to specify a location for the data collection.

Check *Collection Data* if you would like to see *Performed On*, *Performed By*, and *Sampling Comments* columns in the *Work Order* screen.

Check *Testing Data* if you would like to see *Testing Began*, *Testing Ended*, *Tested By*, and *Testing Comments* columns in the *Work Order* screen.

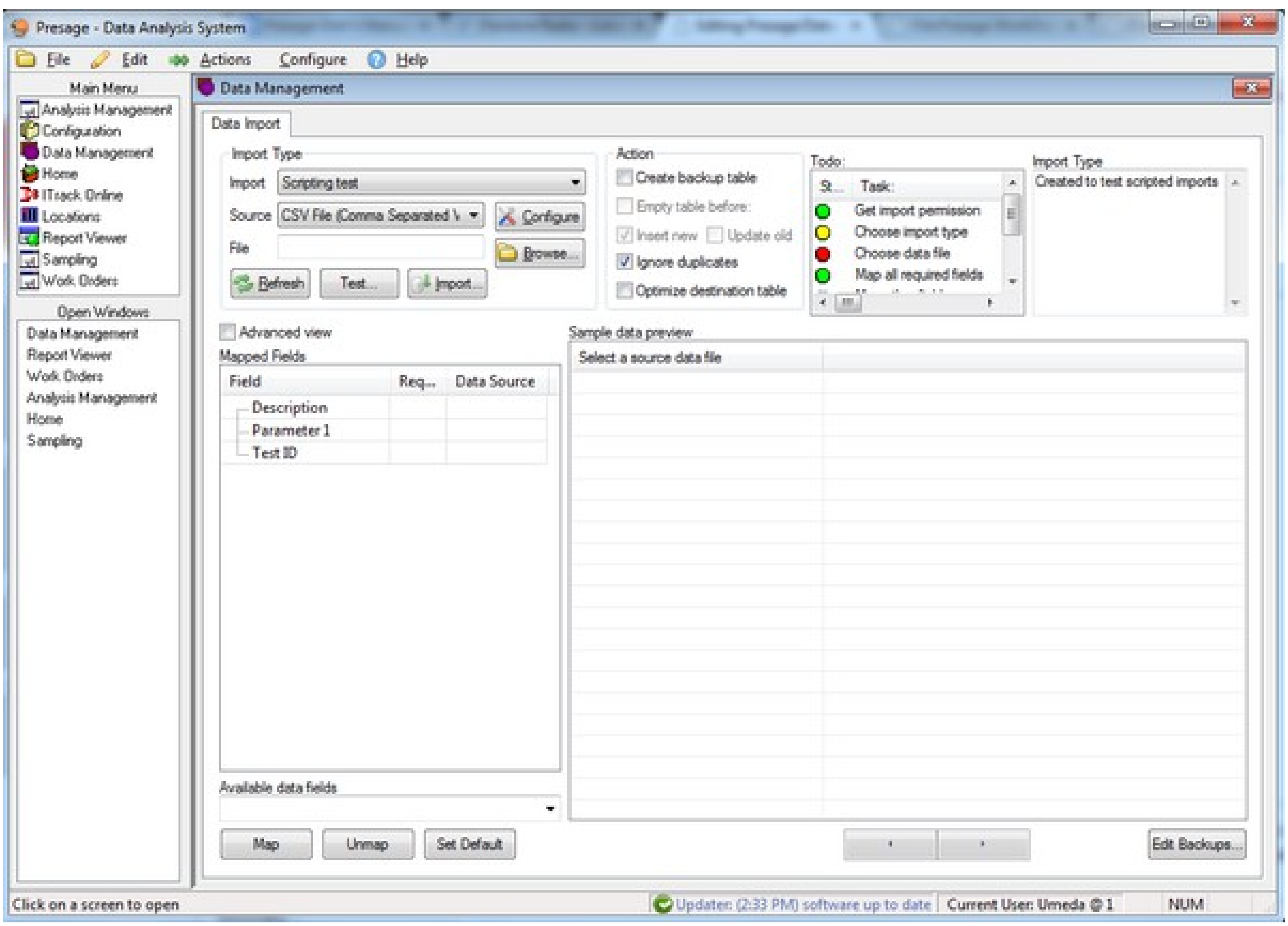
Select an analysis from the *Default Analysis* drop-down list to auto-select this analysis when creating new samples in the *Work Order* screen.

Check *Tasks Page* if this work order type will be for task management (versus sample data collection).

To activate an existing global work order type to a certain plant, simply select the work order type then check the box under *In Use @(Plant Code)*.

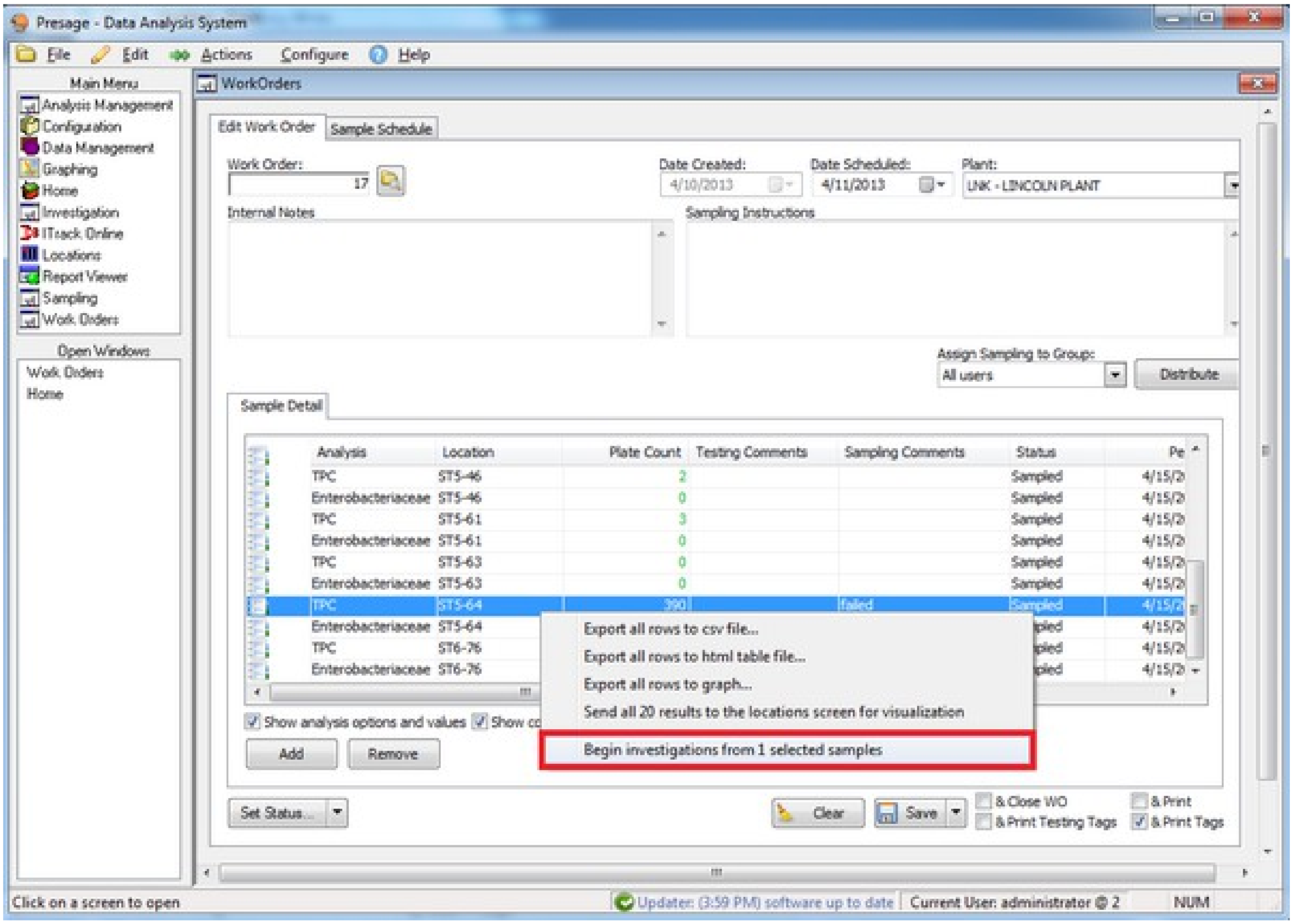
Data Management Screen

This screen is used to import data into Presage.

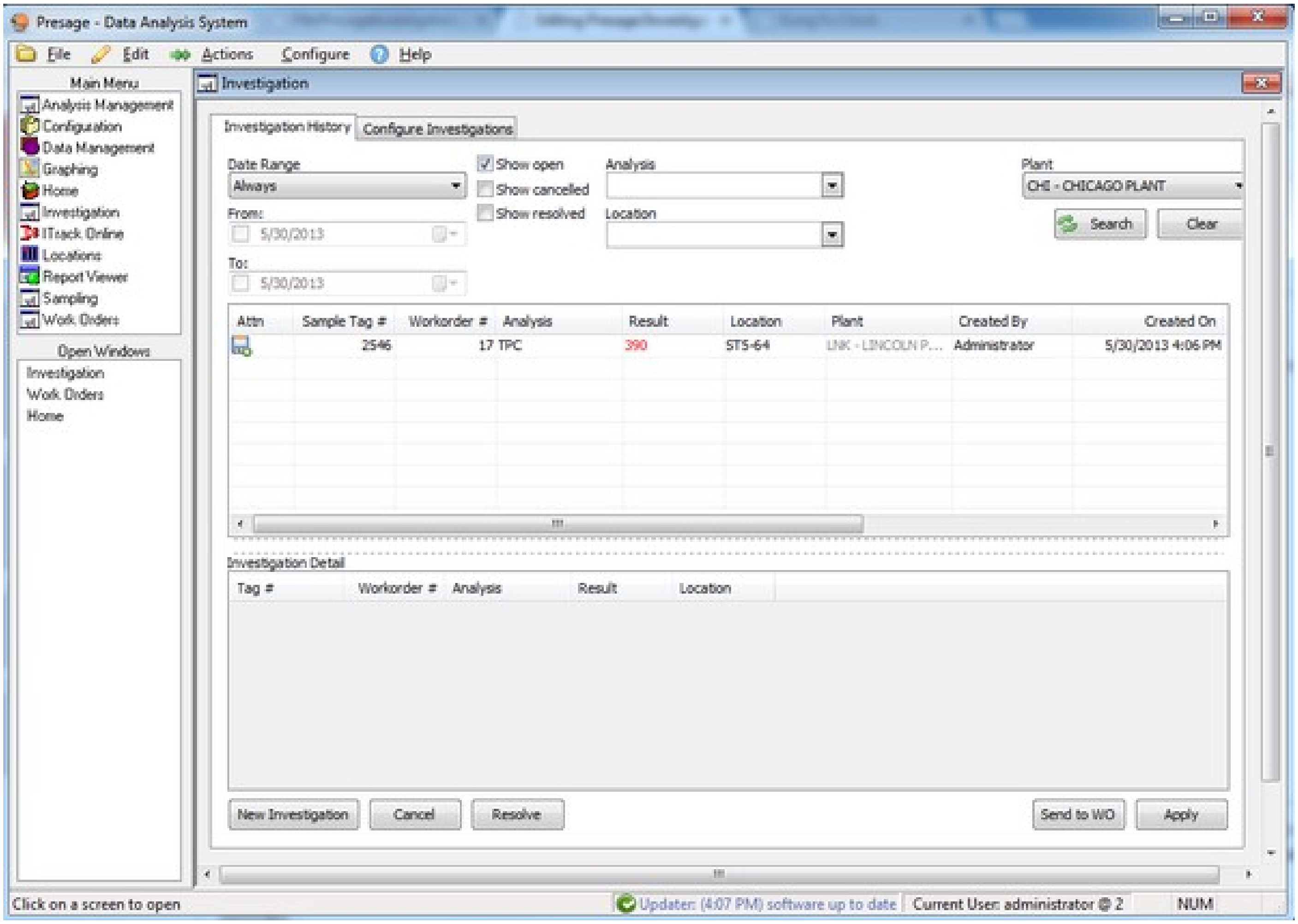


Investigation Screen

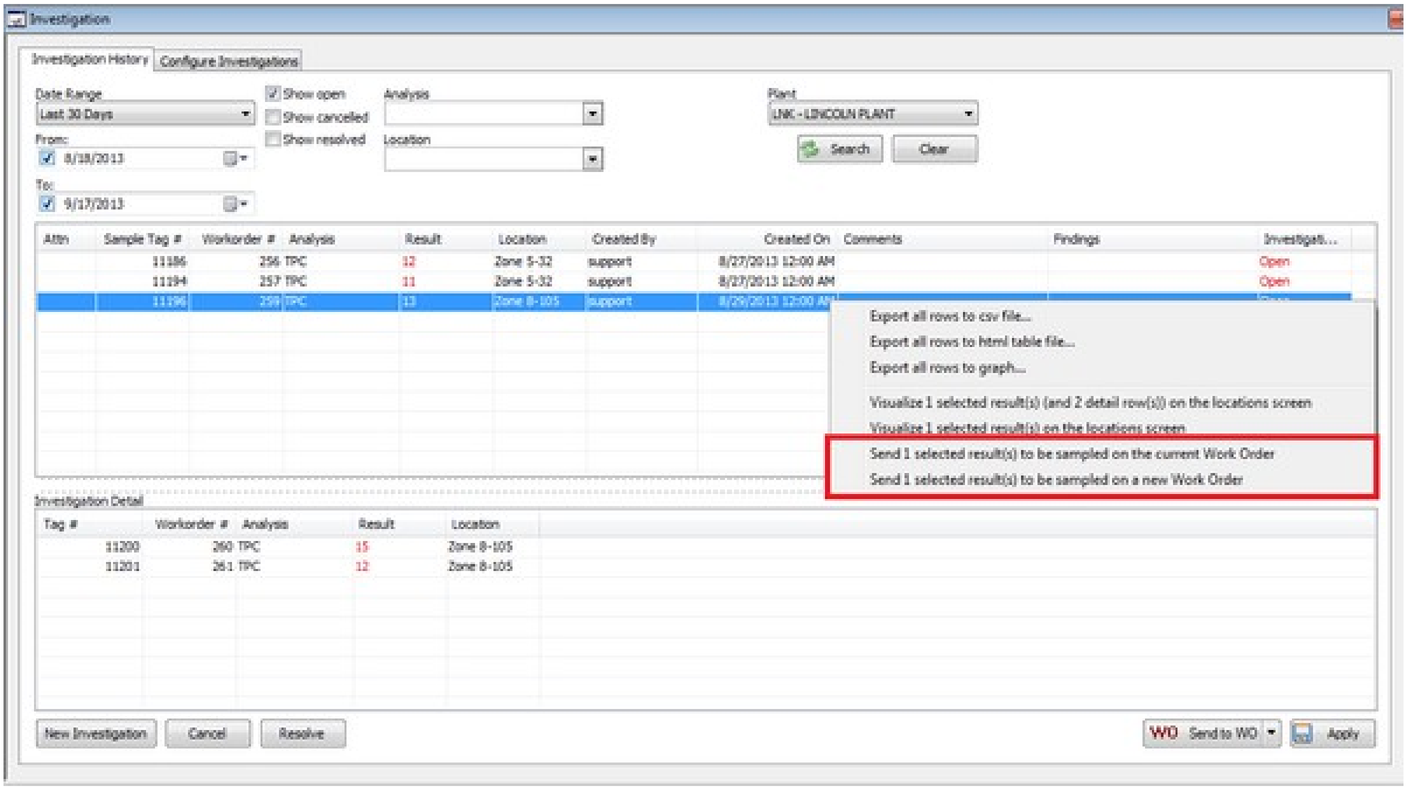
To send an analysis for investigation, first find the failed analysis from the *Work Order* screen or *Sampling History*; right-click on it, and select **Begin investigation from # selected samples**.



User is automatically taken to the *Investigation Screen*.



Right-click on the analysis on this page to see multiple action options: view the location visualization or send to the work order. There is also a **Send to WO** button on the lower, right-hand portion of the screen. Click on the arrow next to the **Send to WO** button, choose between *Send to Existing WO* and *Send to New WO*.



Drag the bar to the right to see the column titled *Comments* and *Findings*. Type in the findings and corrective actions taken in the space provided.

If there are more fails related to this investigation, send the next failed analysis to the investigation screen. This fail will be listed in the *Investigation Detail* (lower) box. Make sure to send the initial analysis listed in the upper box in the *Investigation Screen* to the WO for a retest.

Configure Investigations

Constraints Sub-Tab

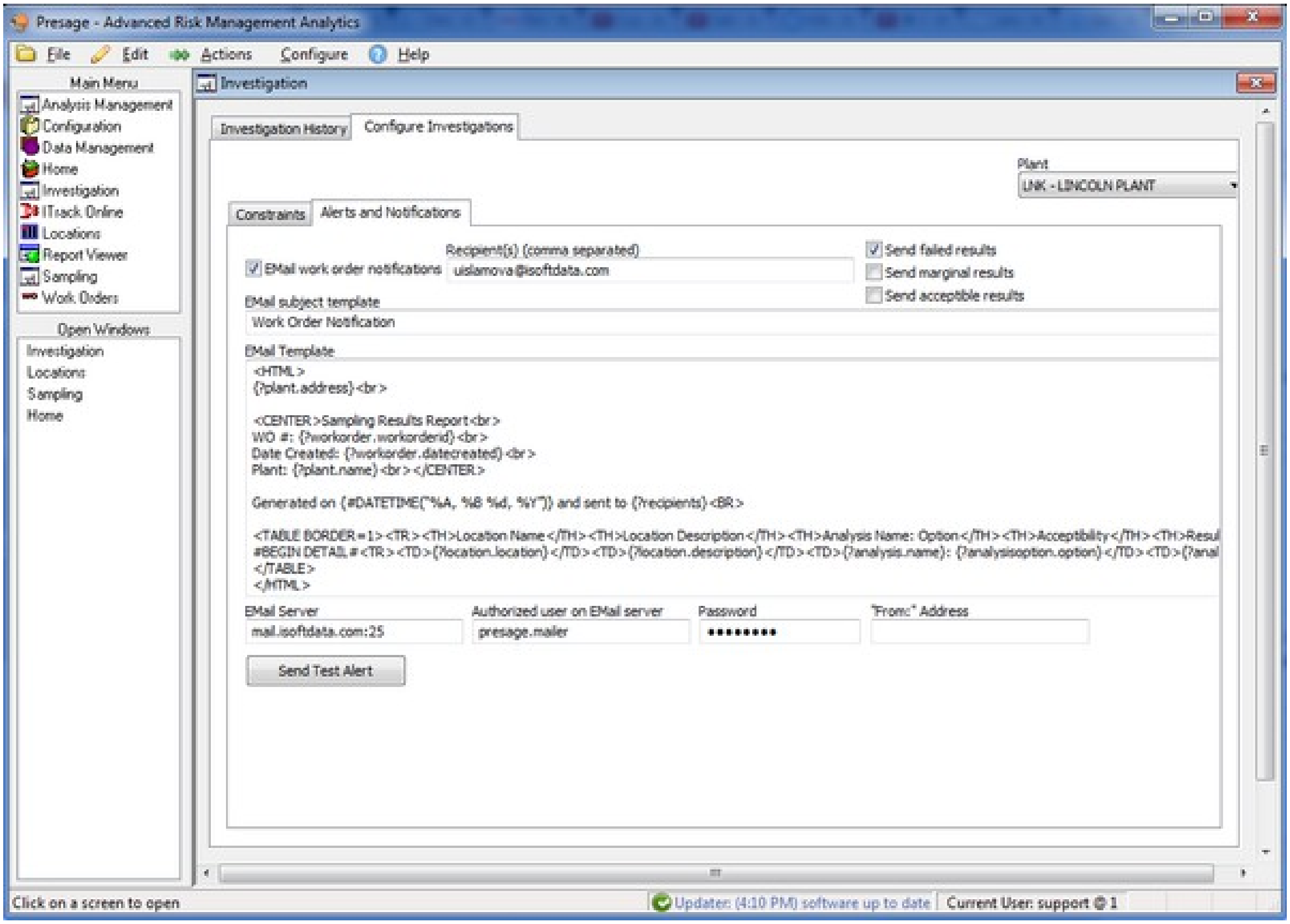
This screen is currently under construction.

Alerts and Notifications Sub-Tab

Use this screen to set up e-mail alerts which can be sent from the 'Work Order' screen by checking the box next to *Send Notification* in the 'Work Order' screen and clicking on *Save*.

First, select a plant. Second, enter the e-mail addresses for the recipients. Third, select which results should be included in the e-mail: fails, marginals, and/or acceptable results.

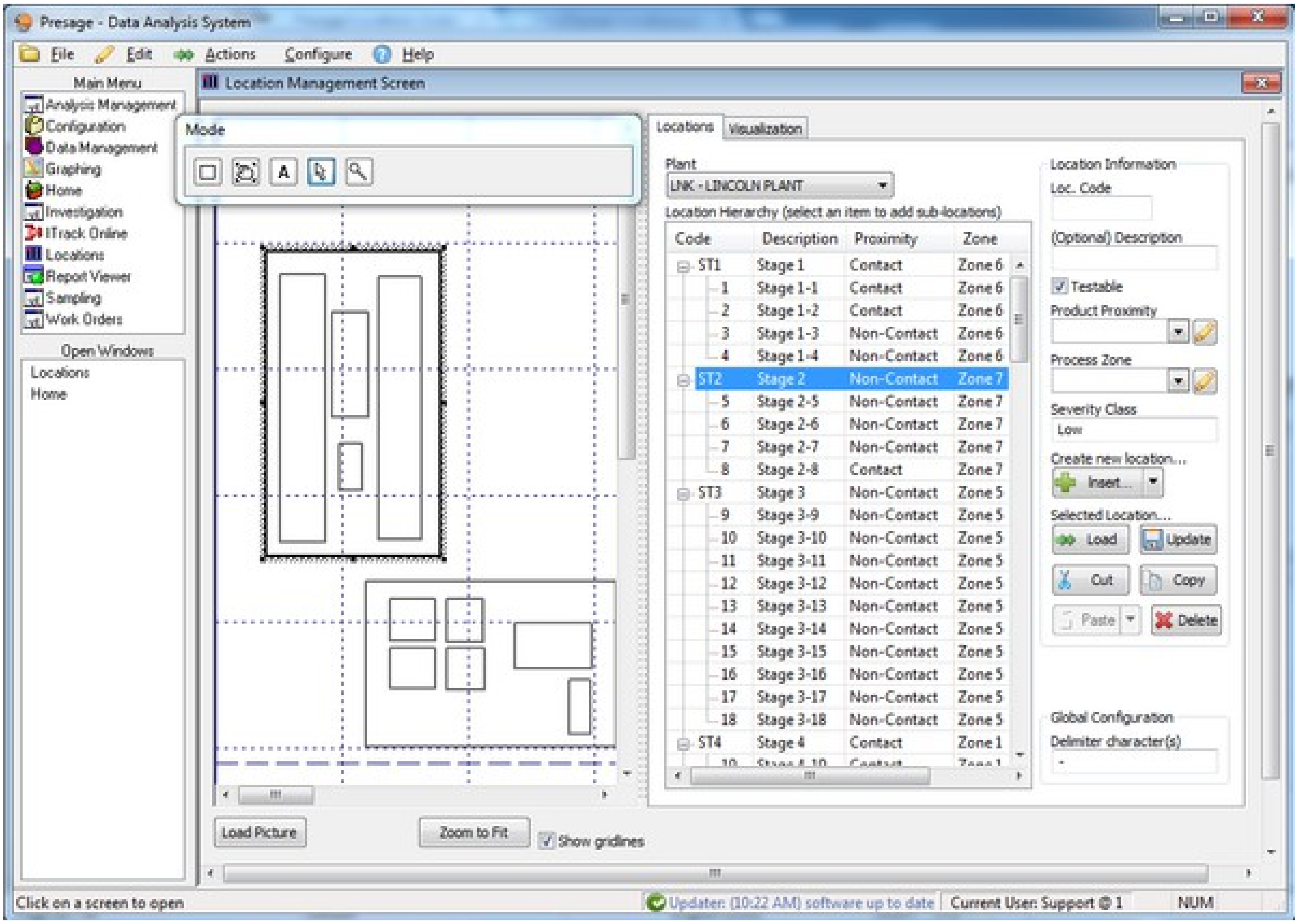
The template can be customized to each company's needs, simply contact a Presage representative with requests.



Locations Screen

This page is for creating a visual map of the plant or location.

First, select the plant from the *Plant* drop-down list in the upper right-hand portion of the screen.

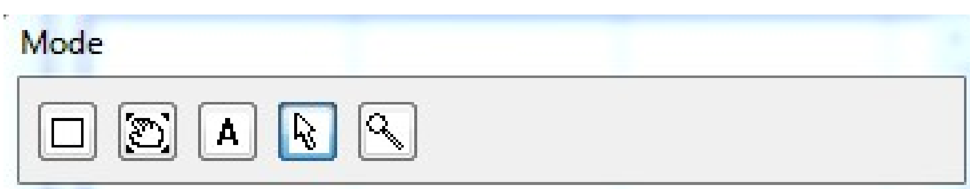


To upload a blueprint of the plant, click on **Load Picture**, locate and select the file, then click **Open**.

Click **Zoom to Fit** to make the entire map visible.

Uncheck the *Show gridlines* box to get rid of the background gridlines.

To map a location, select a location from the *Location Hierarchy*, then click on the button with a box on the *Mode* pop-up window. Click and drag cursor to place the box where appropriate. Now, the box and the location identification name are connected, and once the location identification name or code is selected, the coinciding box on the map will be highlighted automatically.



To add text, click on the box with the letter *A* then click and drag on the canvas to see the cursor appear.

Use the magnifying glass button on the *Mode* pop-up window to zoom in the picture or diagram.

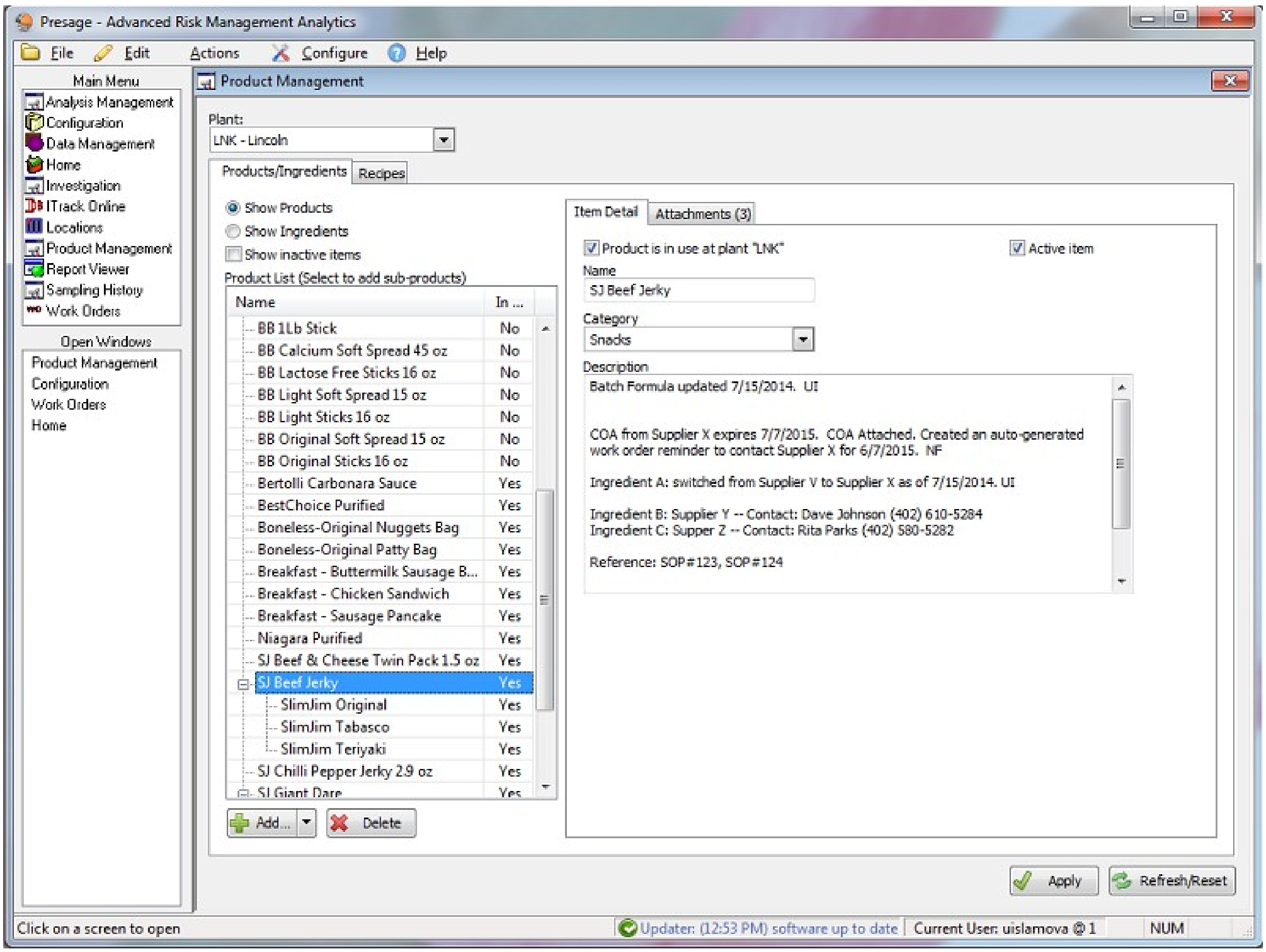
Use the arrow button to rearrange and re-size boxes. Use the hand button to drag the canvas.

Tip: When several small squares are laid out very close to each other, it is easier to select one particular box by clicking on any box within the group and pressing Tab on keyboard to select other boxes within that area.

For guidelines on *Location Information*, refer to *Configuration: Locations*.

Product Management Screen

Product Management is where ingredients and products can be managed. In this screen, users can keep an up-to-date global list of products and assign and remove products and ingredients to and from each plant.



Report Viewer

The report viewer previews reports before they are printed, making it easy to find the correct report and to make necessary adjustments.

Report Selector

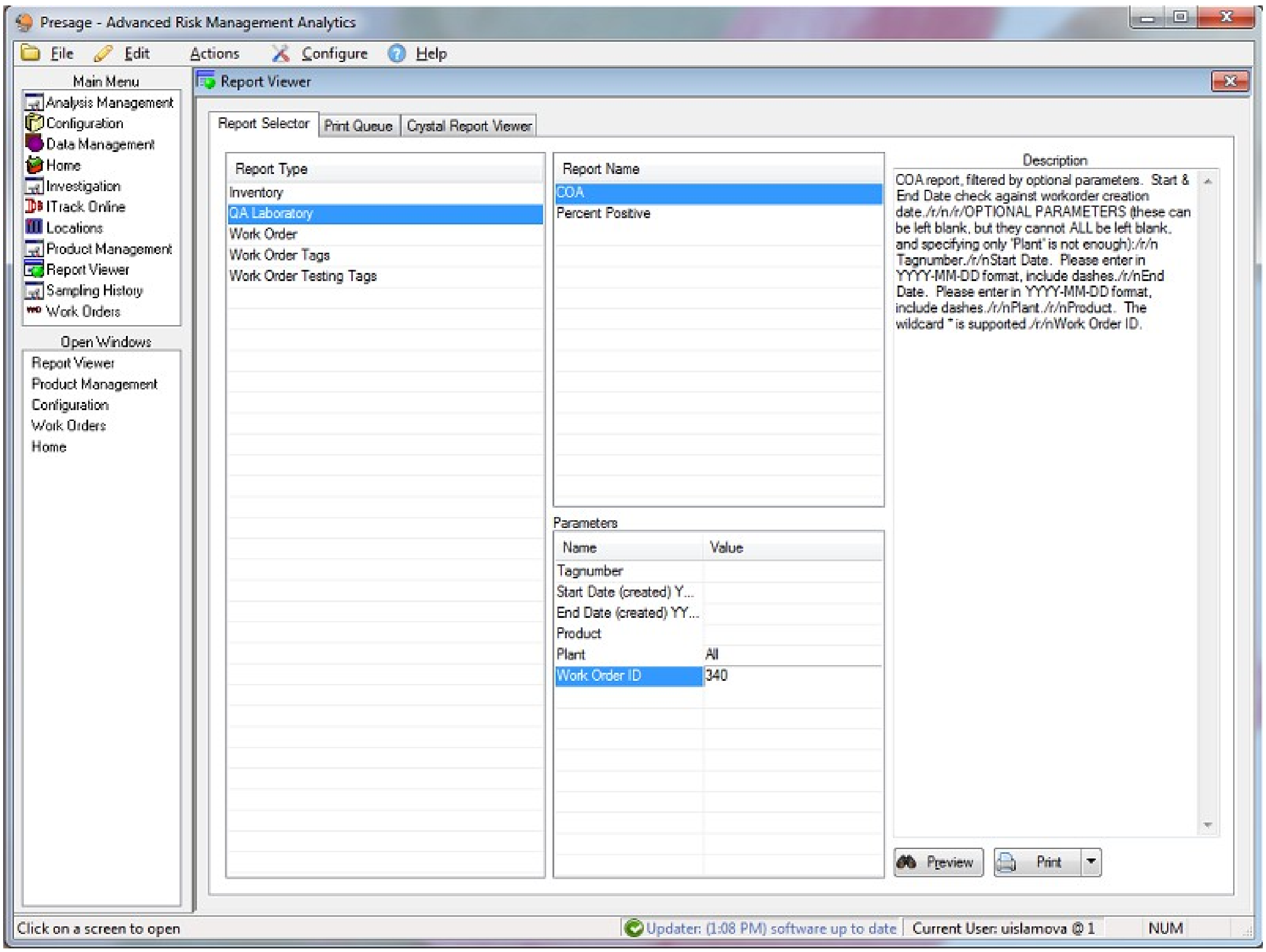
The *Report Name* lists the names of available reports for each report type. Click on a report. *Description* area gets automatically filled out with information about the report.

The *Parameters* section shows what extra information Presage needs to generate a report. Some reports require date ranges, while others require a user to type in information.

Click on the appropriate parameter, then input values into the box provided. For date parameter, click on the drop-down menu then the calendar icon to select the necessary date.

Click on the *Preview* button to open the *Crystal Reports Viewer* tab which displays the preview of the report before printing.

Click *Print* to print the selected report.



Crystal Reports Viewer

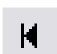
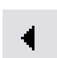
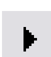
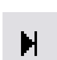
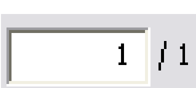
The *Export Report* button exports the report into Excel format. On the pop-up window, select *Microsoft Excel 97-2000(XLS)* from the *Format* drop-down menu. Make sure the *Destination* drop-down menu has *Disk file* selected. Click *OK*. Another window will pop-up. In the new window, define the formatting options for the Excel file. Click *OK* and specify where to save the report.

Reports can also be exported in other formats such as CSV (comma separated value) and XML (eXtensible Markup Language).

The *Print Report* button prints the report from the *Crystal Reports Viewer*.

Toggle Group Tree button will slide over the report preview and open up a new section of the screen. The new section will show how many items are in each group and subgroup of data (if it is applicable for the report).

The *Crystal Reports Viewer* has a group of navigation buttons near the top of the page.

Image	Button
	Go to the first page
	Go to the previous page
	Go to the next page
	Go to the last page
	Go to page x of y

Sampling History Tab

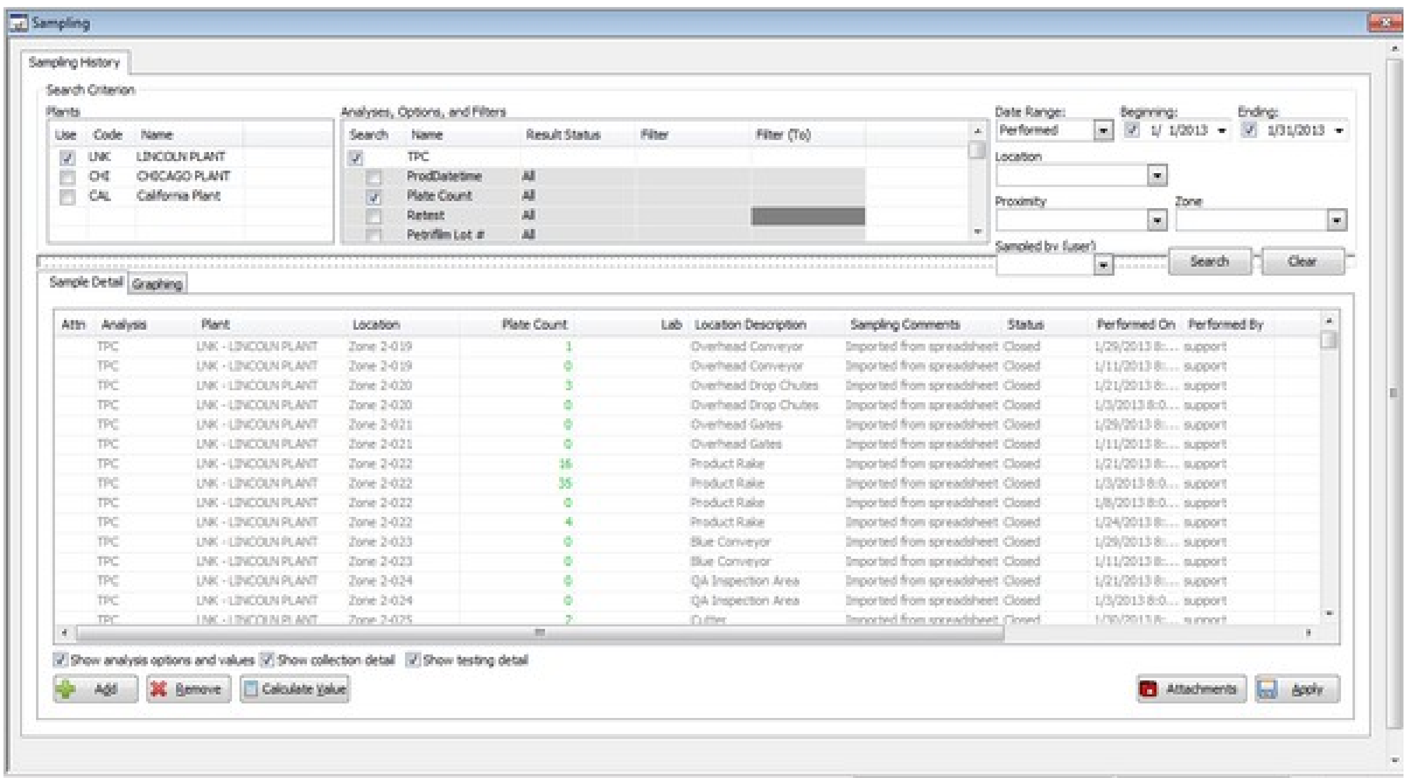
This screen is used as a search engine which makes it easy to find past sampling information.

Check the box(es) next to the analysis name(s) which pops-up the list of options: check the appropriate boxes for options; plug in the variables in the boxes provided: date range, plants, location, user, proximity and/or zone.

Click **Search**.

Once historical data is pulled up on the sampling screen, four things can be done:

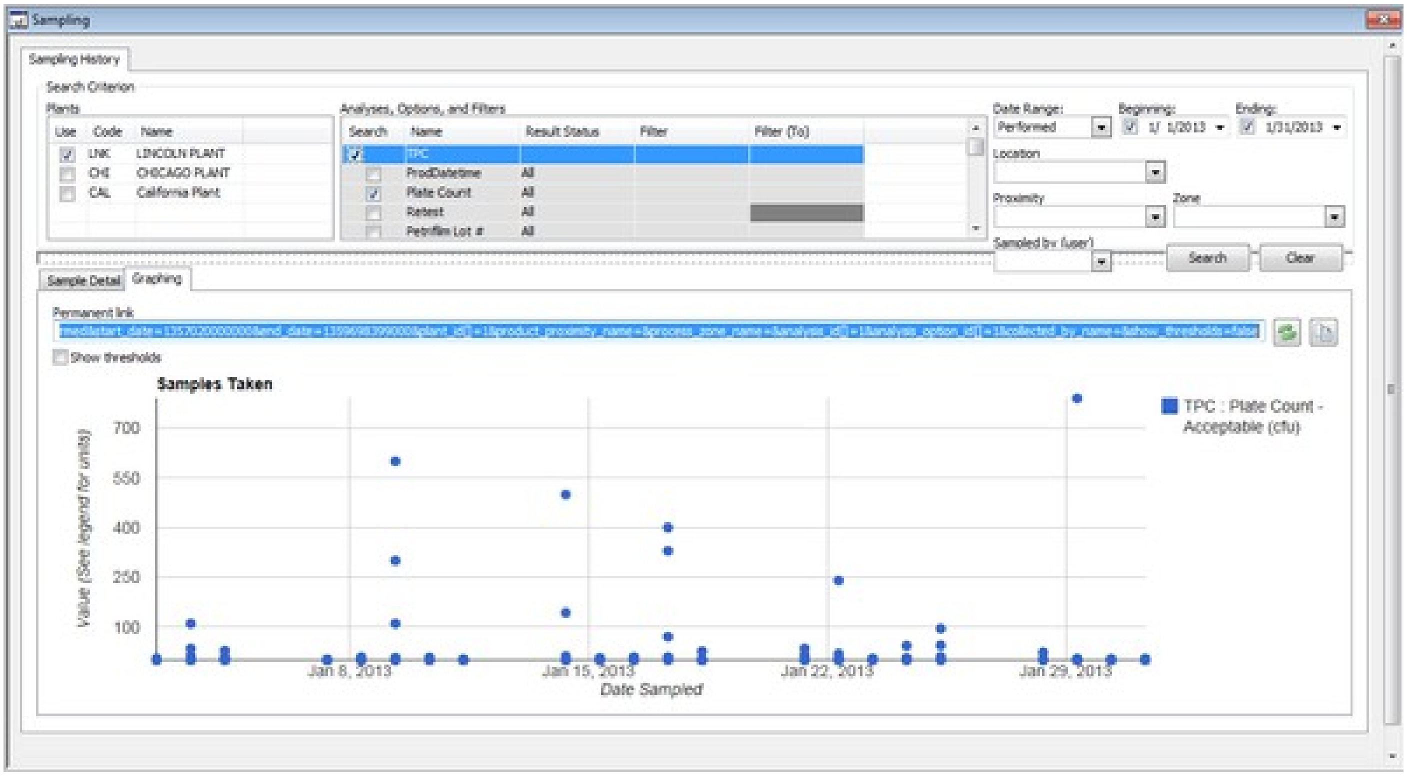
1) Sent to the scatter graph for trending as explained above. 2) Sent to the Location Visualization which shows the sample acceptability over the plant layout. 3) Sent to the Graphing to do more complex charting. 4) Exported to a .csv file.



Click on the column title on the table, to order the data by a certain criteria. For example, click on the *Performed On* to see results in chronological order of date performed.

To Trend Data:

Click the *Graphing* tab for a scatter graph for the criteria selected.



Select and change criteria for the graph using the same panel as used with sampling history on the upper portion of the screen.

Hover over points to view details such as date, time, analysis, analysis option, location, etc.

Hover over a label in the map key to highlight the group of points which belong to that label.

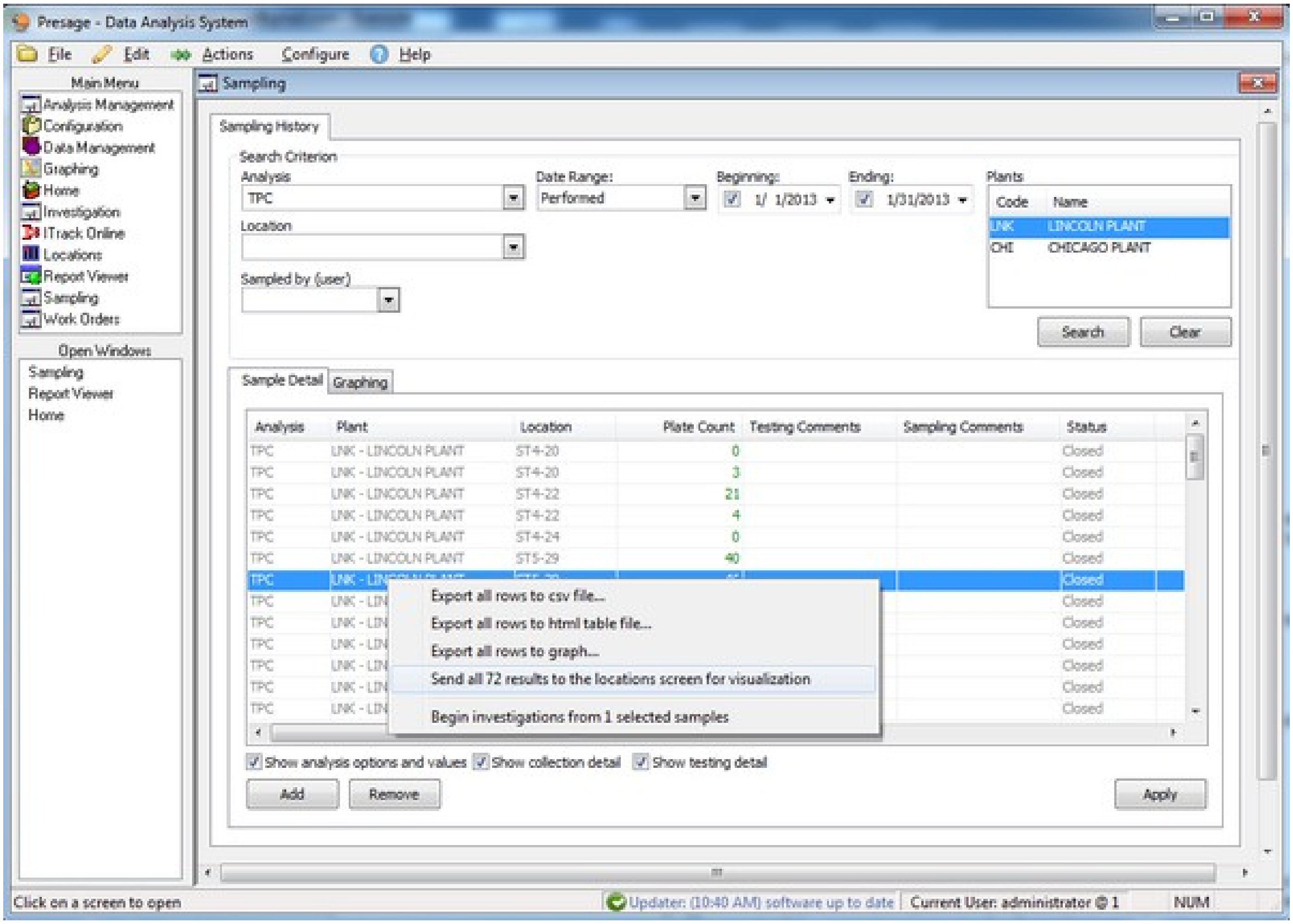
To share the graph with a third party, click on the copy icon next to the browser link box on the mid-right side of the screen (it is the second button next to the browser link box). Then paste the link.

Check the **Show Thresholds** button to see the threshold lines on the graph.

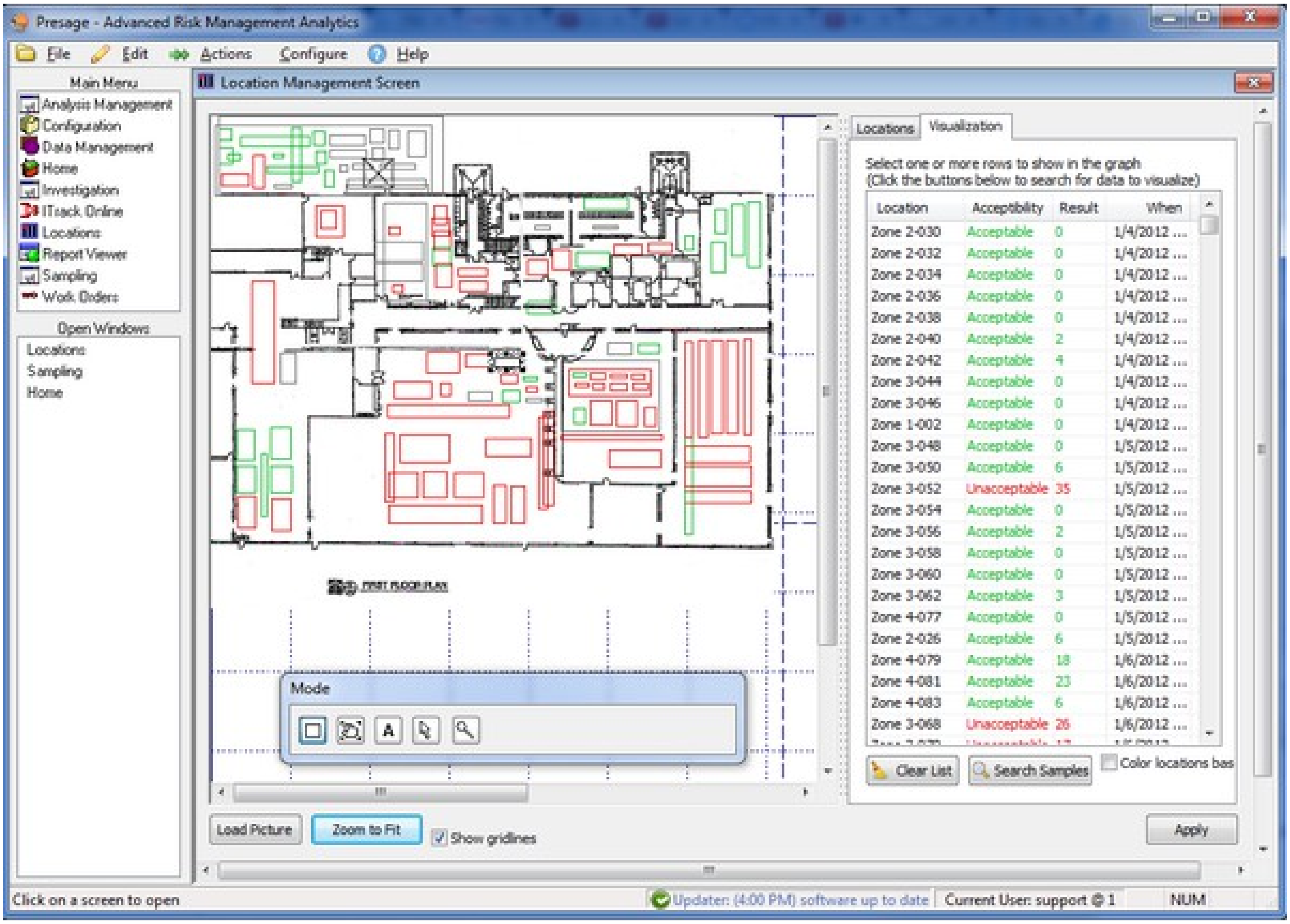
Visualization

To see Location Visualization:

First, enter criteria and pull up historical data. Right-click on one of the rows. A dialog with multiple options pops-up. Click **send all # results to the locations screen for visualization**.



A new screen opens up. This is the *Visualization* tab under *Locations* Screen.



On the left display, locations are highlighted in green and red. Green locations are ones with acceptable counts, and red locations are ones with fail counts.

Highlight a row on the right and the corresponding location will be highlighted automatically on the left display and vice versa.

Order the *Date Sampled* column by clicking on the column title **When**. Click the first row and press the down-arrow on the keyboard to see a play-through of where the samples were taken in a chronological order.

Note: Every row can be ordered by clicking the column title.

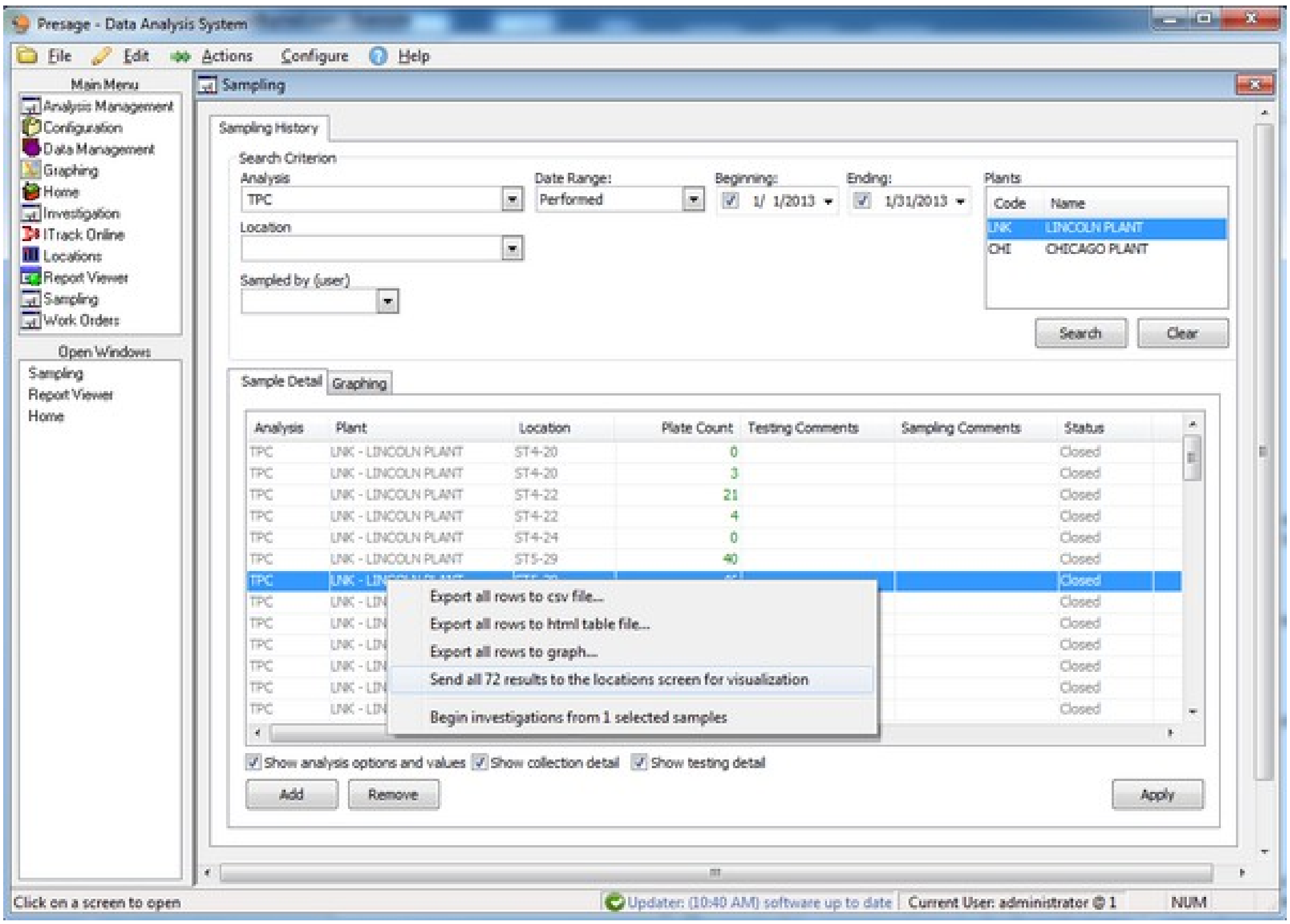
Check the box **Color locations based on selected samples only**, and select a group of rows to see the colors only for those selected.

Graphing

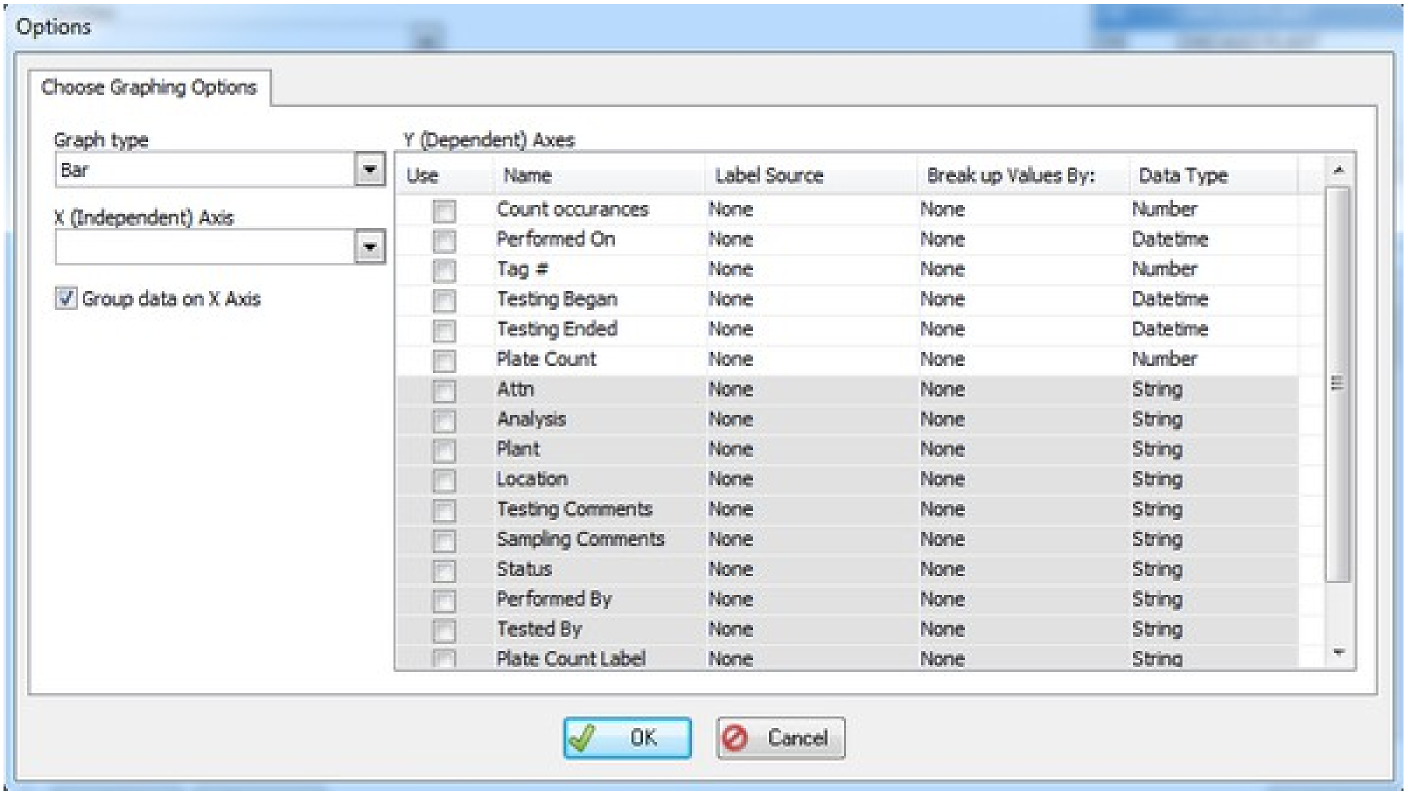
To do more complex graphing:

Once historical data is pulled up on the sampling screen, this data can be graphed in multiple ways, in addition to the scatter graph.

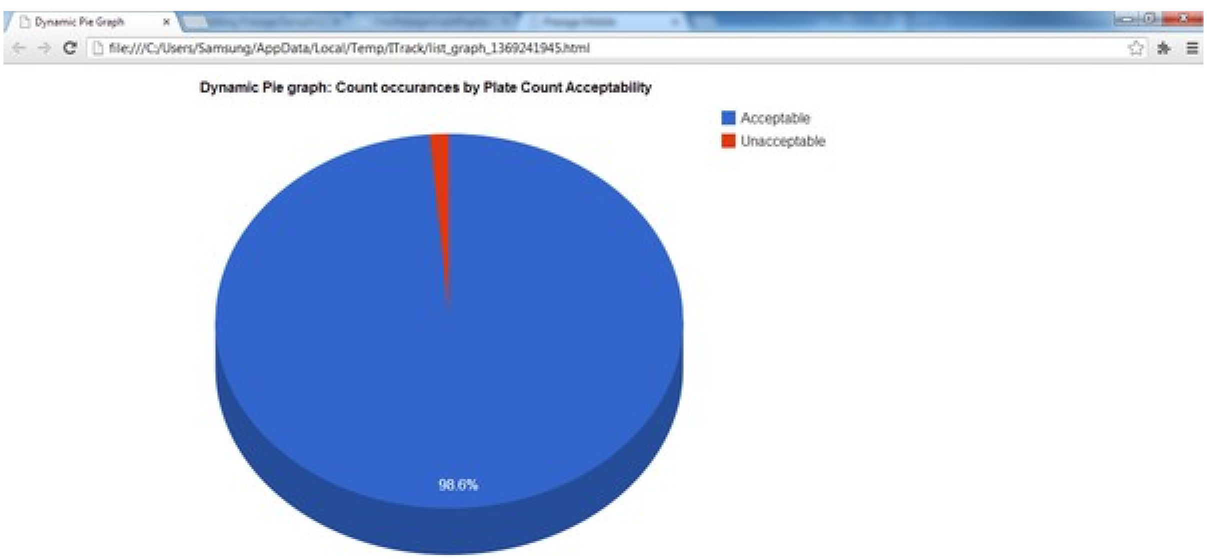
First, enter criteria and pull up historical data. Right-click on one of the rows. A dialogue with multiple options pops-up. Click **export all rows to graph**.



The following dialog pops-up. Make selections, then click **OK**. The graph will open in a new browser page.



These graphs are saved in the local computer and cannot be shared with third-parties.



Export Data

Please, contact a Presage representative for exports.

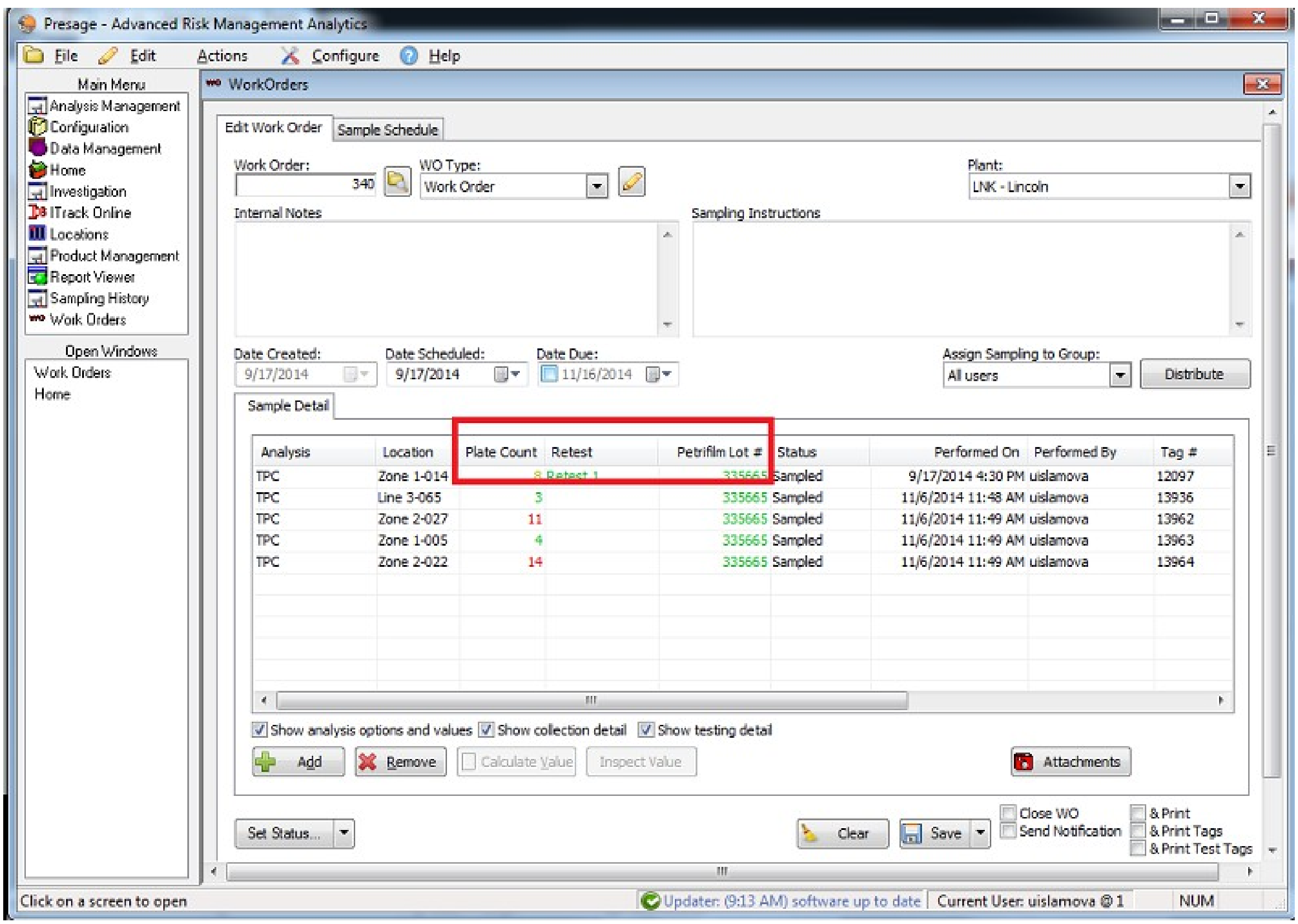
Work Orders

Edit Work Order Tab

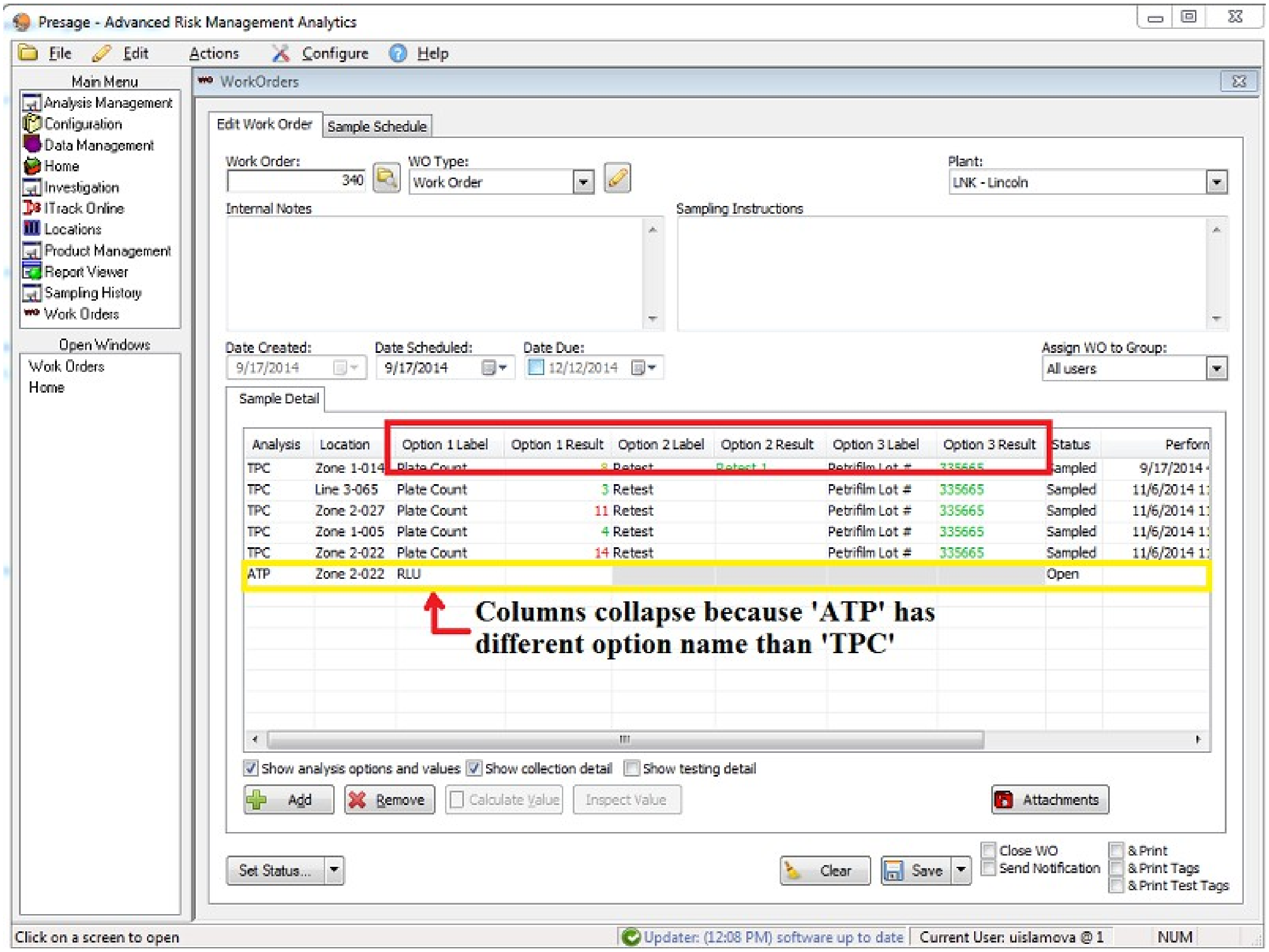
Use this page to create and edit work orders and enter data collected for analyses.

Double-click on the work order in the *Home* screen work order queue to pull it up in this page. There is also a way to search for work orders in this page by clicking on the blank box under *Work Order:*. On the pop-up window, enter the work order number. Click **OK**. The work order will show up under the *Sample Detail* tab.

The format of a work order will depend on the analyses selected. If all analyses have the same group of options, the option names will appear on the column headers. And results need to be entered below the option header. Like the following:



If the analyses have different set of options. Then the option names will appear in a field under the column header **Option # Label**. And results need to entered next to the option name under the column **Option # Result**. See below:



Add Sample

To add an analysis, click **Add**, a new highlighted row will pop up. Start entering data for the new analysis. To remove an analysis, select the row (use CTRL key to select multiple rows), then click **Remove** on the lower portion of the screen.

Set Date Scheduled and Date Due

Date Scheduled is by default the date work order is created. To change *Date Scheduled* click on the calendar icon next to the date. To add *Date Due*, check the box on left of the date then click on the calendar icon on the right of the date.

Note: If there is no *Date Due* field, it could be because the work order type is configured not to have a due date.

Assign to Group

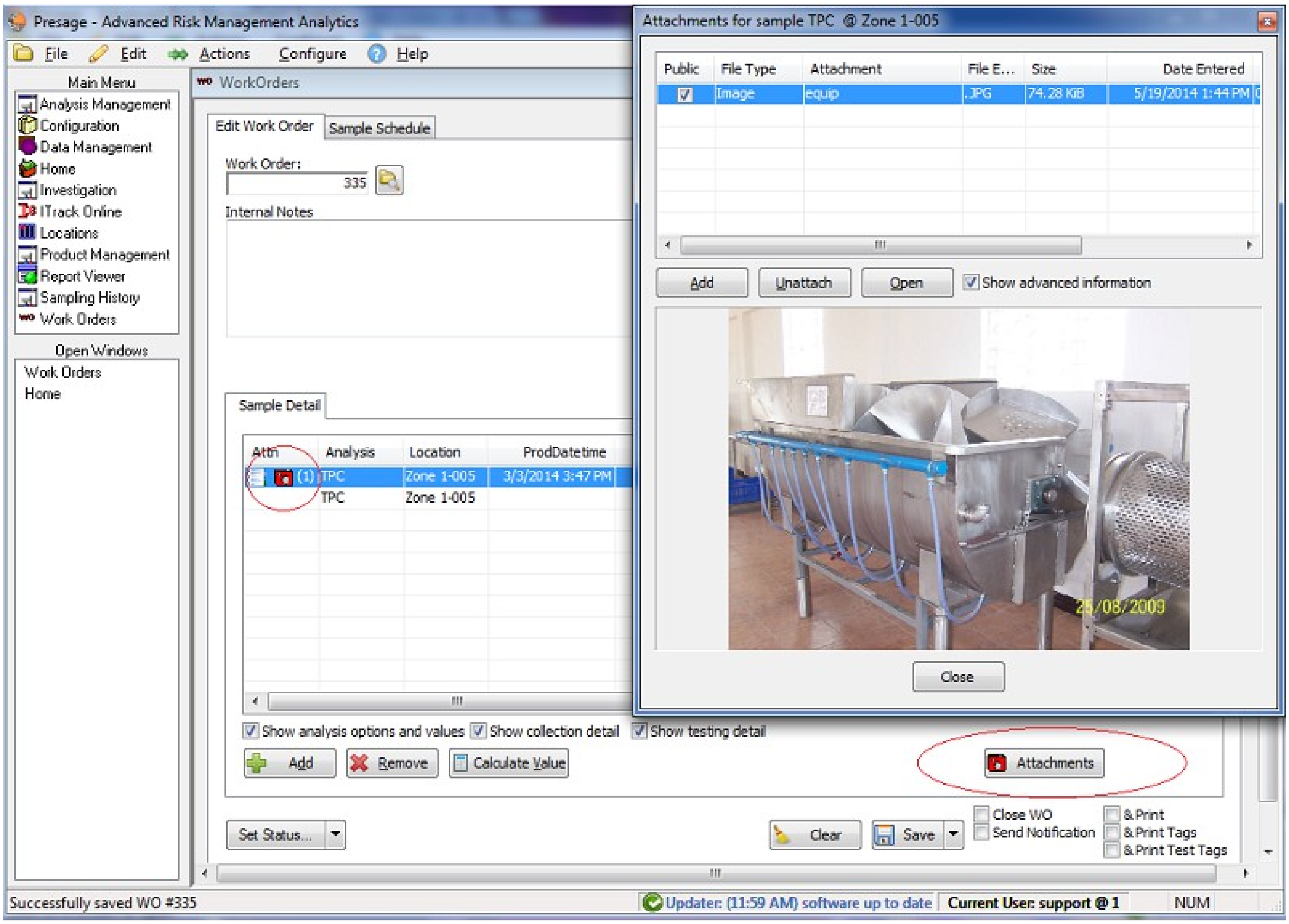
To assign the work order to a group, select the group name from the drop-down list under *Assign WO to Group:*. Click **Save**.

Make Attachment

To make an attachment to a sample, select a sample and click the **Attachments** >> **Add** >> select an image >> **Open** >> **Close**.

A camera icon will appear for the sample under the **Attn** column.

To view the attachment, select the sample and click **Attachments**. Select the attachment you would like to view.



Save

To save all changes, click **Save**. There are options to print and close work order once changes are saved; if applicable, click on the check-box.

Print Tags

When printing reports or tags, check the **&Print** box when saving the work order. A window will pop-up with list of reports, choose the appropriate report. On the next window, choose the appropriate printer and click **OK**.

Status, Performed On, Performed By

These columns are filled out automatically, but can be edited (only on the software, not the scanner).

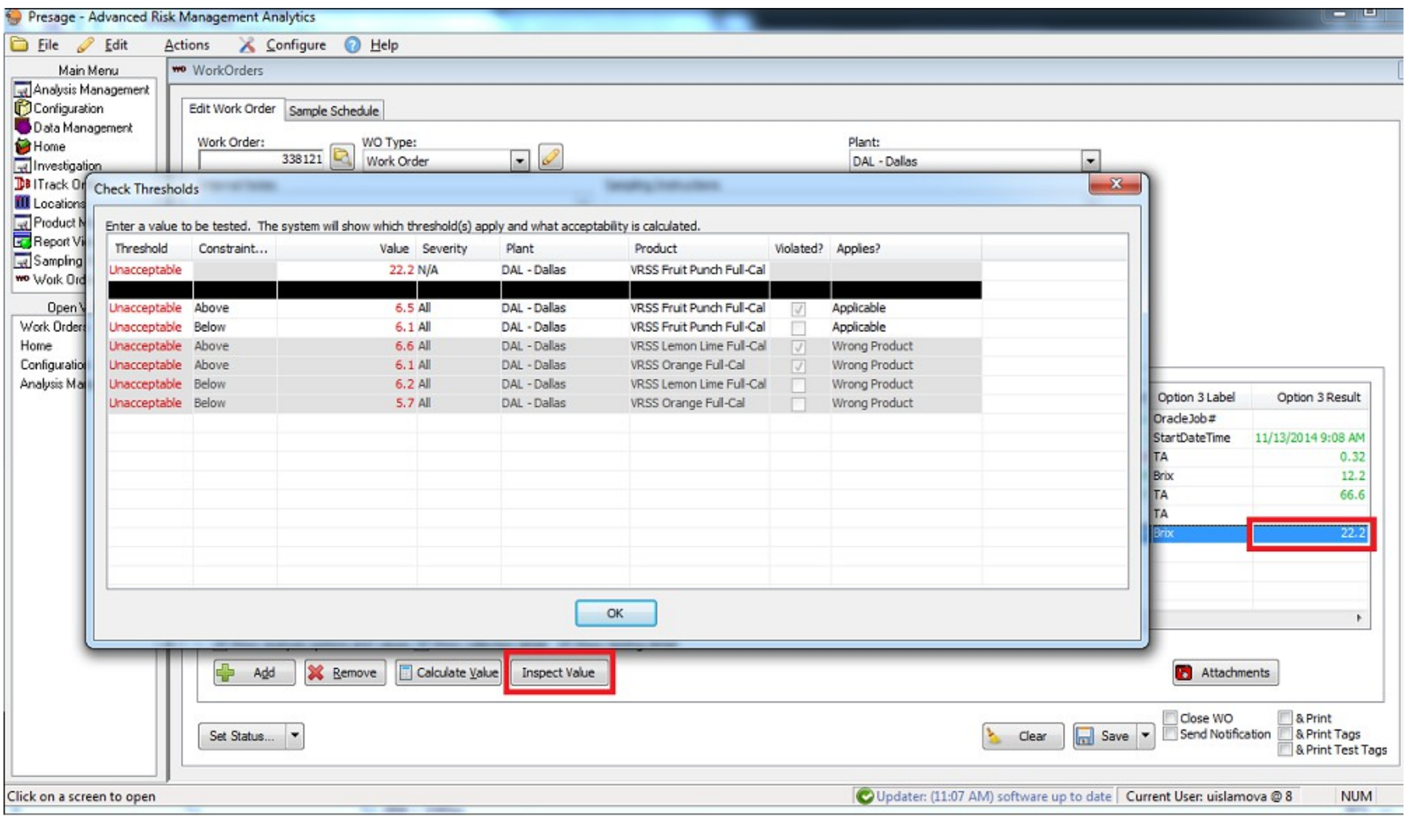
Under *Sample Detail*, click on the box under *Status* to change the status of the work order. Select the appropriate status title from the drop-down menu: *Opened, Sampled, Closed,* and *Cancelled*. Set the appropriate date and time the analysis was performed in the next column. Then select the appropriate user who performed the test in the column *Performed By*.

Edit Tag#

By default *Tag#* is generated by Presage. Turn on the ability to edit Tag# in the **Configuration >> Users or Groups >> Permissions**.

Inspect Value

To inspect how the result acceptability was calculated, select the result then click **Inspect Value**. A pop-up window will show all the specifications considered in calculating the value acceptability.



Set Status

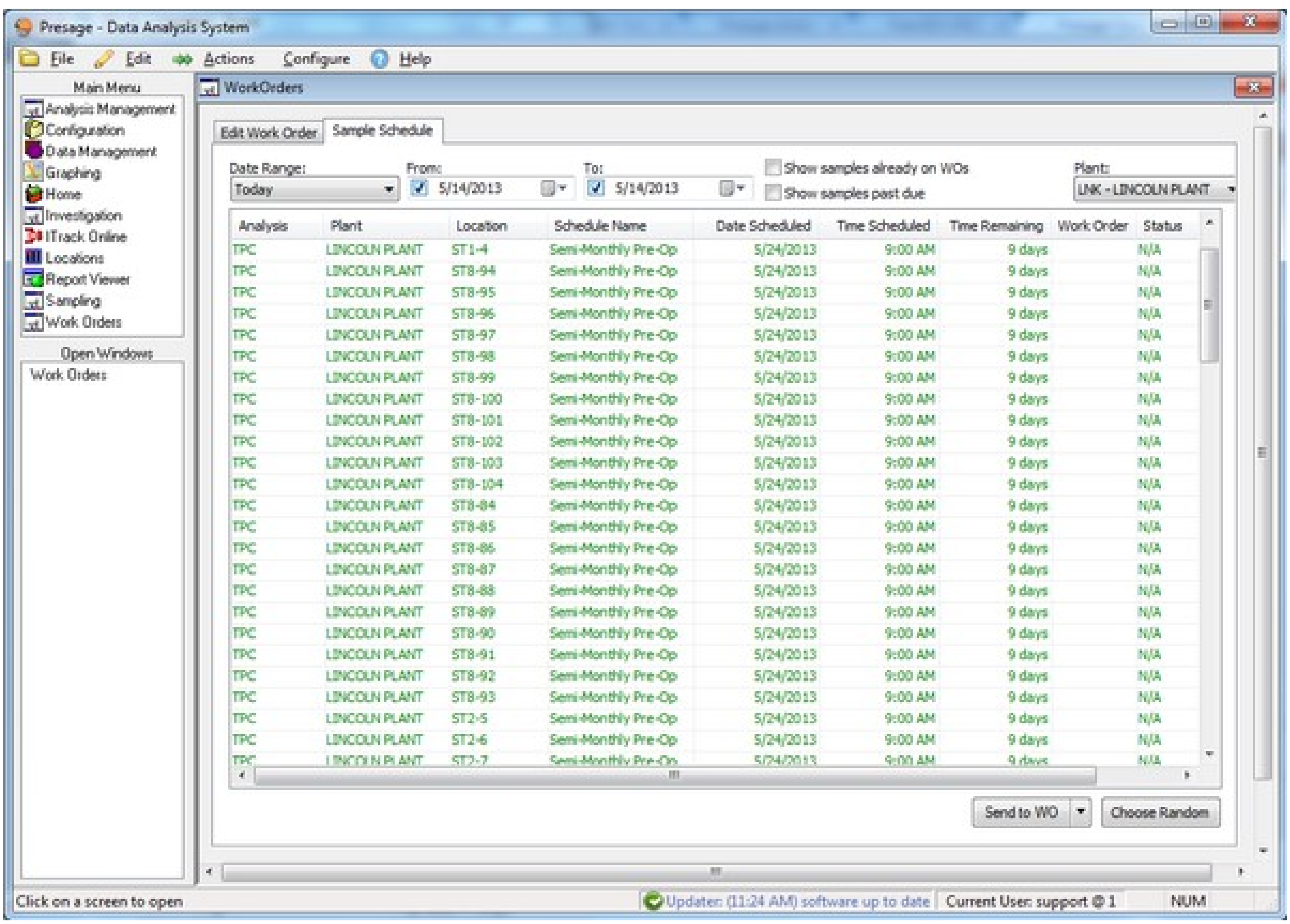
The **Set Status...** button on the lower left corner of the screen allows users to change the status of groups of analyses simultaneously. Select **All Items...** or **Only Selected Items...**, then **To Cancelled** to invalidate the analyses in the Work Order. Invalidated samples do not show under *Sample History*.

Hide Select Columns

To hide some columns, uncheck the appropriate boxes on the lower portion of *Sample Detail* tab. The *Show analysis and option values* box hides columns titled *Option Name* and *Option Value*. The *Show collection detail* box hides columns titled *Performed On* and*Performed By*. The *Show testing detail* box hides columns titled *Testing Began, Testing Ended* and *Tested By*.

Sample Schedule

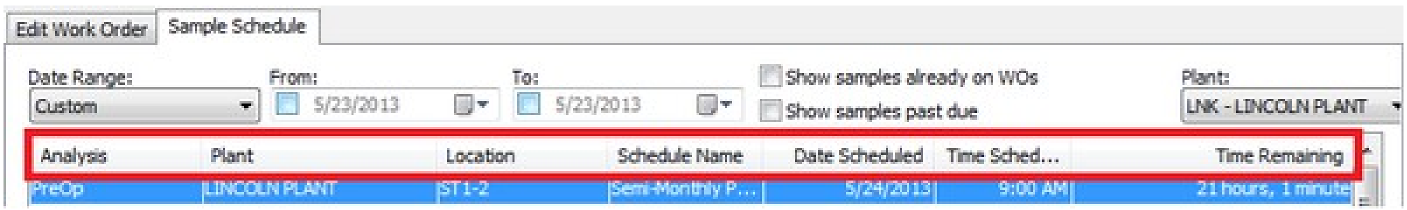
This screen lists all the analysis tests which need to be performed. First column lists the analyses which are created in the *Analysis Management* screen. Second column lists the plant that particular analysis is scheduled to be tested in. Next column is the site the analysis is assigned to be sampled from. *Schedule Name* is the interval period the test is performed in; it is created in the *Analysis Management*. *Date scheduled* is the date the analysis was created and *Time Remaining* is the number of days left to perform the test.



Choose the plant from the drop-down menu on the right.

Choose the date range to view analyses scheduled for specific dates. For custom date range, select *Custom* from the drop-down menu. Then set the *From:* and *To:* dates from the calendar icon or type it in. Otherwise, leave the date range as is, as this schedule is ordered by the least number of days remaining to be performed.

Check the **Show samples already on Wos** box to see all analyses, ones already assigned to a work order in addition to ones not assigned yet. The work order numbers will show up in the second to last column, and work order status will show up in the last column.



Click on the column titles to put the list in order based on the selected category. For example, click on the *Analysis* column title to order the list alphabetically by analysis names; click on *Location* to order the list alphabetically based on the name of locations, etc.

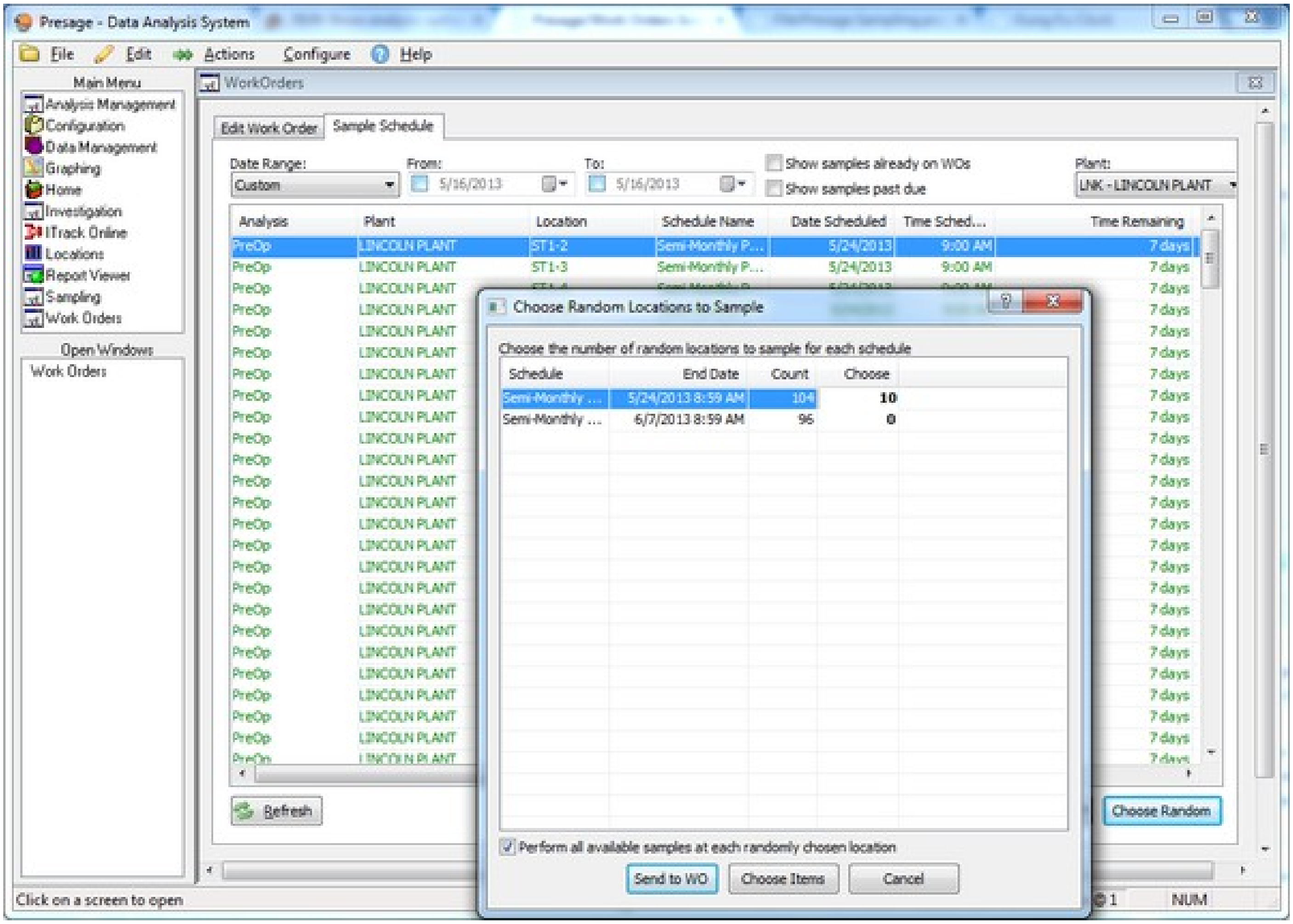
This list is color-coded. These colors are the default and can be customized.

Blue samples need to be completed within 24 hours.

Green samples have more than a day left to be completed.

Grey samples are past due.

Note: Past due samples will not automatically move to current schedule. They will have to be manually selected and sent to work orders. Set the date range to the past to see past due orders.



To assign random sights to a work order, click on **Choose Random**. A new window will pop-up. Specify the number of locations the system should pick out randomly under the *Choose* column.

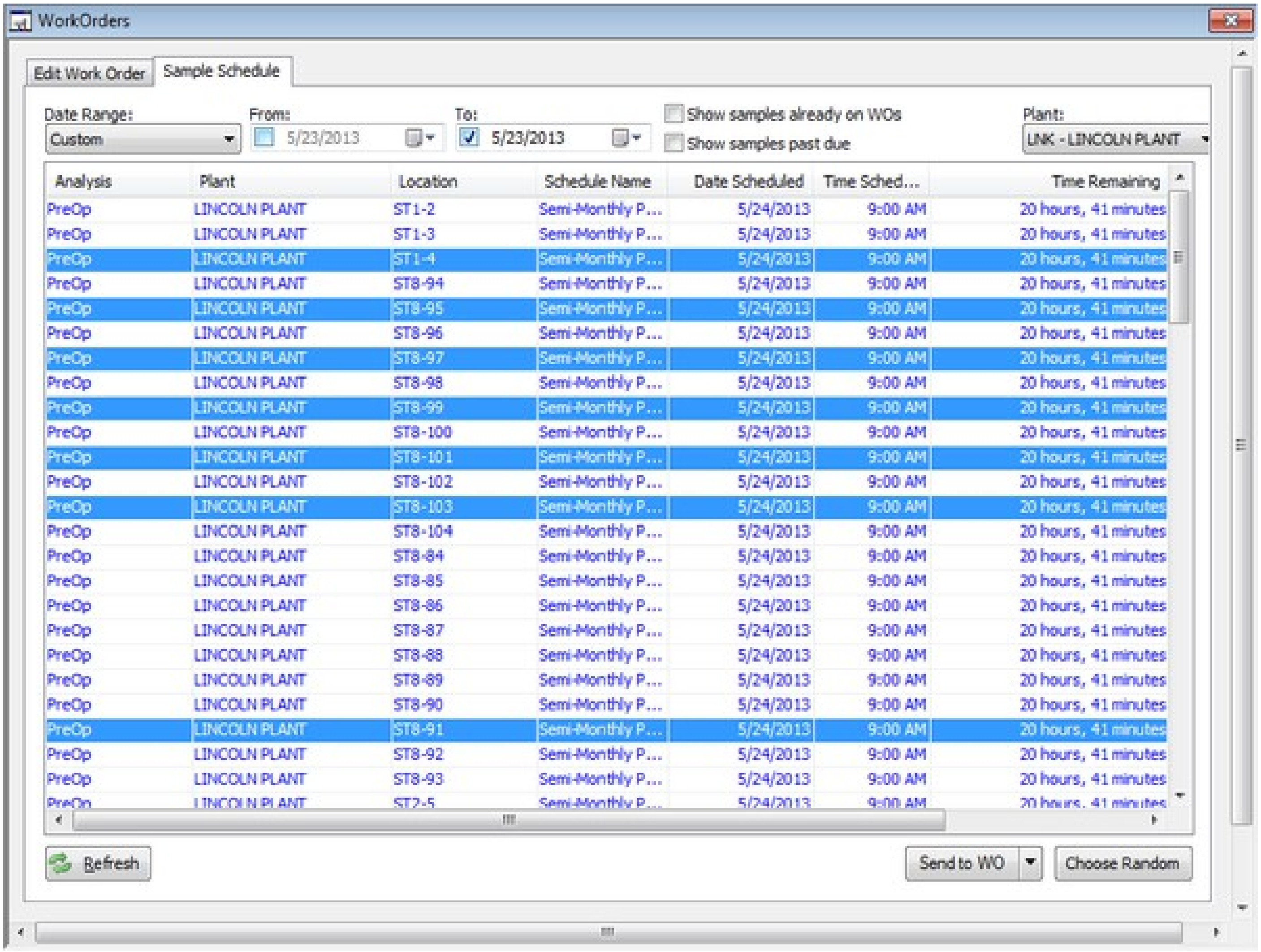
Click **Choose Items**. Random analyses will be selected in the Sample Schedule.

Now, click **Send to WO**. The highlighted analyses from *Sample Schedule* will now show up under the *Edit Work Order* tab.

Make sure the correct *Date Scheduled* and *Plant* are selected.

Click **Save**.

Now, the work order is created and is listed under *Work Orders* in the *Home Screen*.



There is also a way to make manual selections. Press on the *Ctrl* key while making multiple selections on the schedule. Click **Send to WO**, then **Save**. Click **Clear** (before **Save**) to put samples back to the schedule.

If multiple analyses are assigned to one location, the system can grab all analyses tied to that location to the work order. So, if a user specifies ten locations to be randomly picked out from the *Sample Schedule*, there may be more than ten analyses in that work order depending on how many analyses are assigned to each location chosen.

To achieve that, simply make selections from the schedule, click on the arrow-down button next to **Send to WO**, check the box next to the pop-up message **Perform all scheduled analyses for each chosen location**, then click **Send to WO**.

How to insert the iPod in the Linea Pro scanner.

Install the ITrack Mobile App on the iPod.

Click on the **App Store** icon on the iPod.

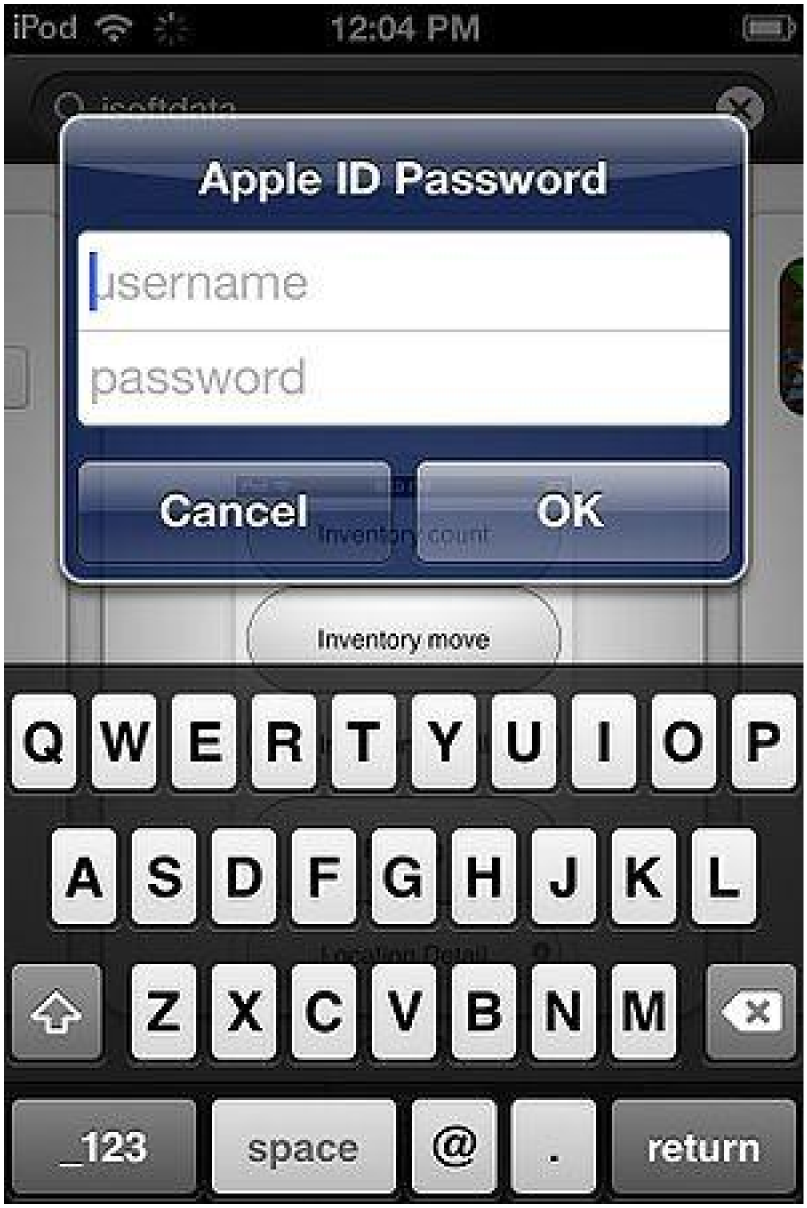
Type in *isoftdata*, and swipe from right to left until the following app shows up:

The button may look different.

Click on the cloud button to install the app.

The following icon pops-up. Click **Use Existing Apple ID**.

Enter the Apple ID and password. (A Presage representative will provide you with this information. It will be used across all plants in your companies, unless requested otherwise.)



In the Home Screen, find the ITrackLX Mobile app icon and click on it.

Enter the Server URL provided by the Presage representative.

Login with Presage username and password and click **Log in!**.

Perform Sample

To time-stamp when a sample was taken, click on the first button **Perform Sample**.

Press the button on the right side of the Linea Pro.

Place the red ray on the barcode. Hold until a sound comes out validating that the barcode was read.

A notification will appear confirming that the sample was marked.

Click **Add Image** to attach an image to the sample.

Click **Done** to go back to the main menu.

The above steps will automatically update the work order in Presage.

1) The *Status* will change from *Open* to *Sampled*. 2) *Performed On* will be filled with the scan date and time. 3) *Performed By* will be filled with the user who logged in to the mobile app.

Incubate Sample

To time-stamp when a test is going in and out of the incubator, click on the second button **Incubate Sample**.

Press the button on the right side of the Linea Pro.

Place the red ray on the barcode. Hold until a sound comes out validating that the barcode was read.

A notification will appear confirming that the sample was marked.

Click **Add Image** to attach an image to the sample.

Click **Done** to go back to the main menu.

The above steps will automatically update the work order in Presage.

1) *Testing Began* will be filled with the date and time the barcode was scanned the first time. 2) *Testing Ended* will be filled with the date and time the barcode was scanned the second time. 3) *Tested By* will be filled with the user who logged in to the mobile app.

Record Analysis

To enter the result for a sample, click on the third button **Record Analysis**.

Press the button on the right side of the Linea Pro.

Place the red ray on the barcode. Hold until it beeps and sample appears on the screen.

All the options for the sample will show up. Select or type in the results.

Click **Done** to save and go back to the main menu. Click **Cancel** to delete changes and go back to the main menu.

The above steps will automatically update the work order in Presage.

Work Order

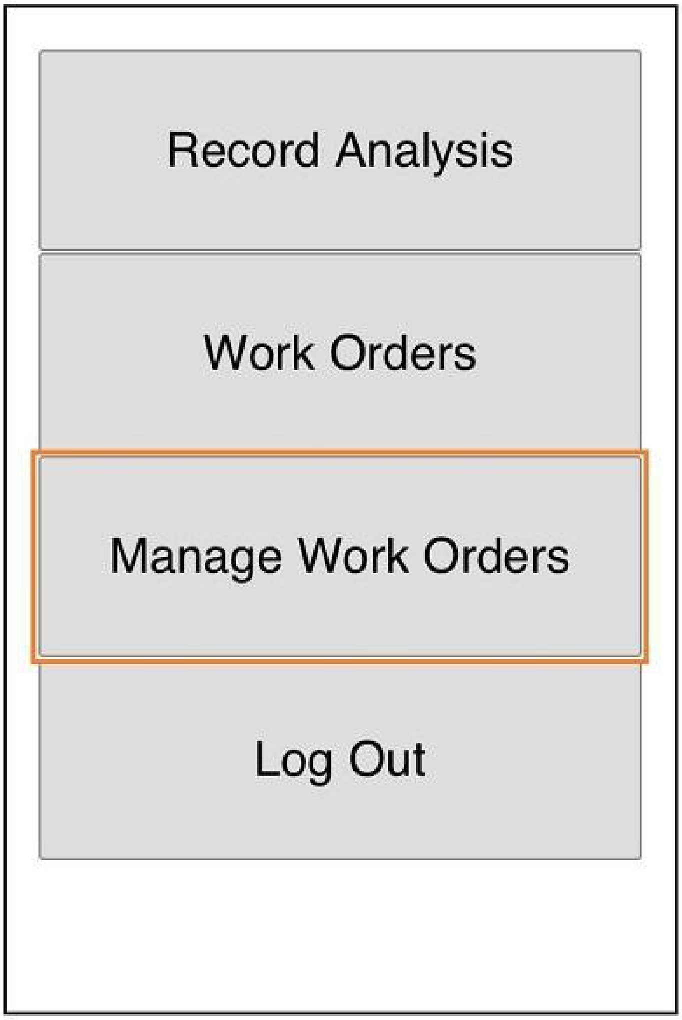
Open and edit a work order.

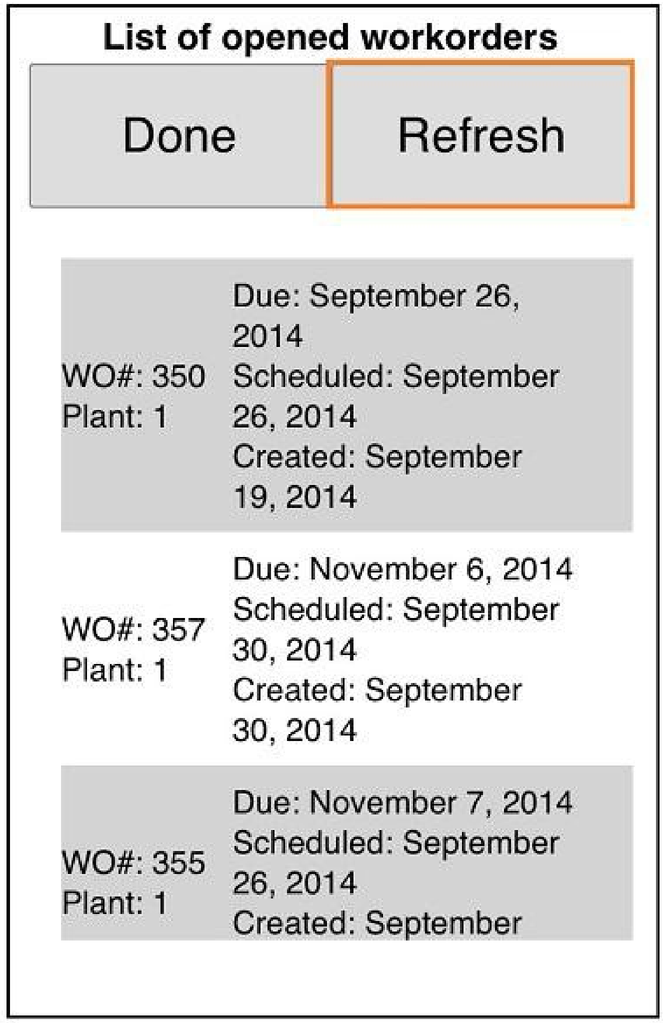
Click **Work Order** in the *Main Menu*. If you have a work order barcode, go ahead and scan that now to open the work order. If not, click **Menu** >> **Manual input: Work Order**.

To create a new work order, click **Menu** >> **New Work Order**.

Manage Work Order

Open a list of all open work orders.





Click **MANAGE WORK ORDERS**

Click **REFRESH** to see an updated list of all open work orders.
Tap on the work order to see the list of samples.

How to create a work order.

1. Click on the **Work Order** screen under *Main Menu*.

2. Click the **Add** button on the lower-left of the *Edit Work Order* tab.

3. Once the row is selected, click on the cell below *Analysis* to choose an analysis name.

4. Click on the cell below *Location* to choose a location name.

Note: If there are multiple analyses to be entered, click the **Add** button multiple times to make the rows editable.

Note: Make sure to remove any blank editable rows before saving the work order. Select the row to be deleted, and click **Remove**.

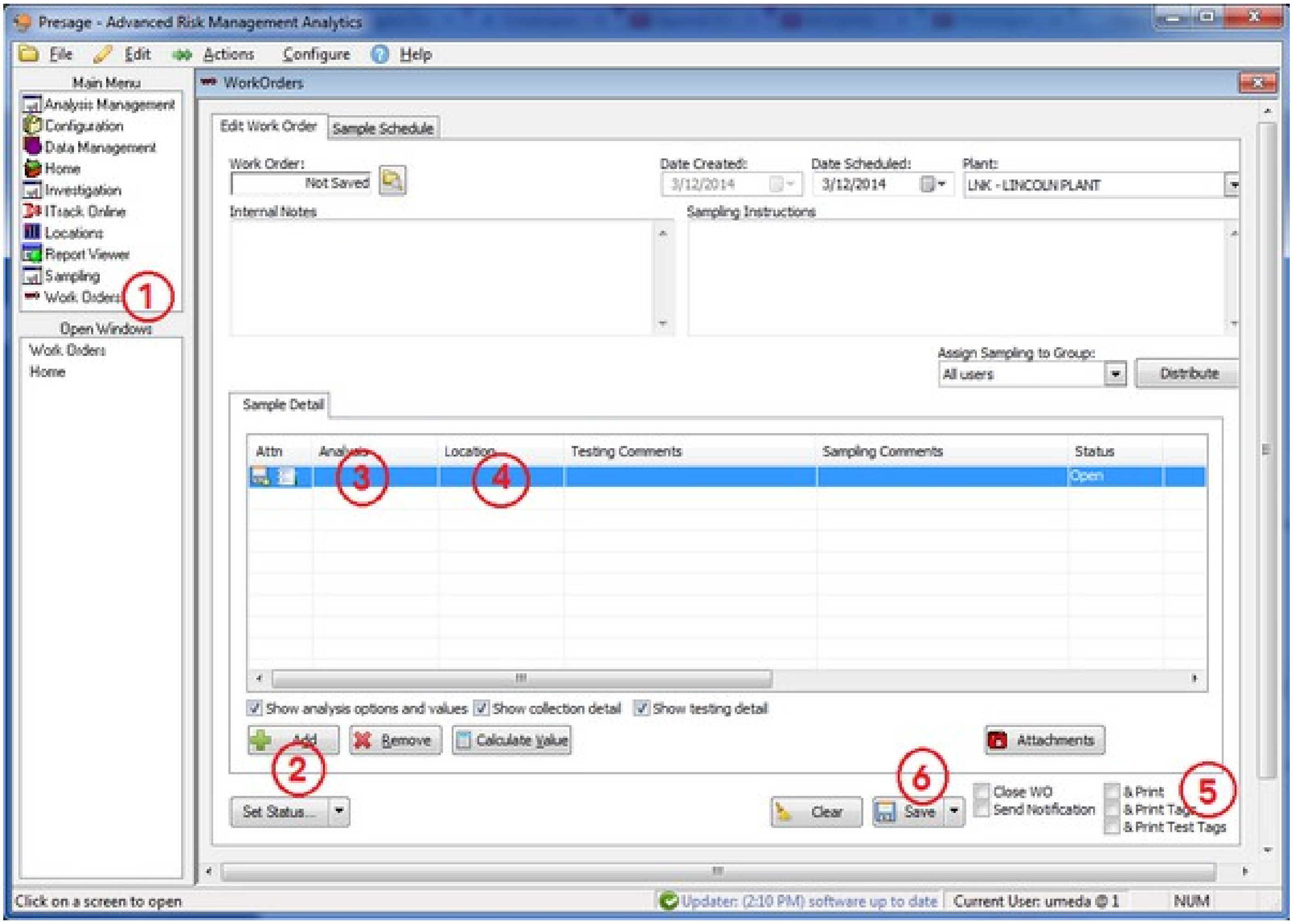
5. Check the appropriate boxes next to printing options.

Remember: **&Print** prints letter size reports.

&Print Tags has two options: with Barcode and without Barcode. Usually it's a 1.1 x 2.4 label, but it can be customized to your needs.

&Print Test Tags prints very thin labels with barcodes. These are meant for sticking on petri dishes and petri films.

6. Click **Save**.



How to hide unnecessary columns from the work order screen.

Uncheck the appropriate boxes on the lower portion of work order screen.

The **Show analysis and option values** box collapses option name columns.

The **Show collection detail** box collapses columns titled Performed On and Performed By.

The **Show testing detail** box collapses columns titled Testing Began, Testing Ended and Tested By.

How to open a closed work order.

1) Pull up the work order in the work order screen.

2) Click on the **Set Status** button on the lower left, select **Re-open closed work order**.

How to edit a closed sample.

1) Pull up the work order in the work order screen.

2) Click on the **Set Status** button on the lower left, select **Only Selected Items...** or **All items...**, then select **To Sampled**.

3) Click **Re-Open WO** on the pop-up dialog.

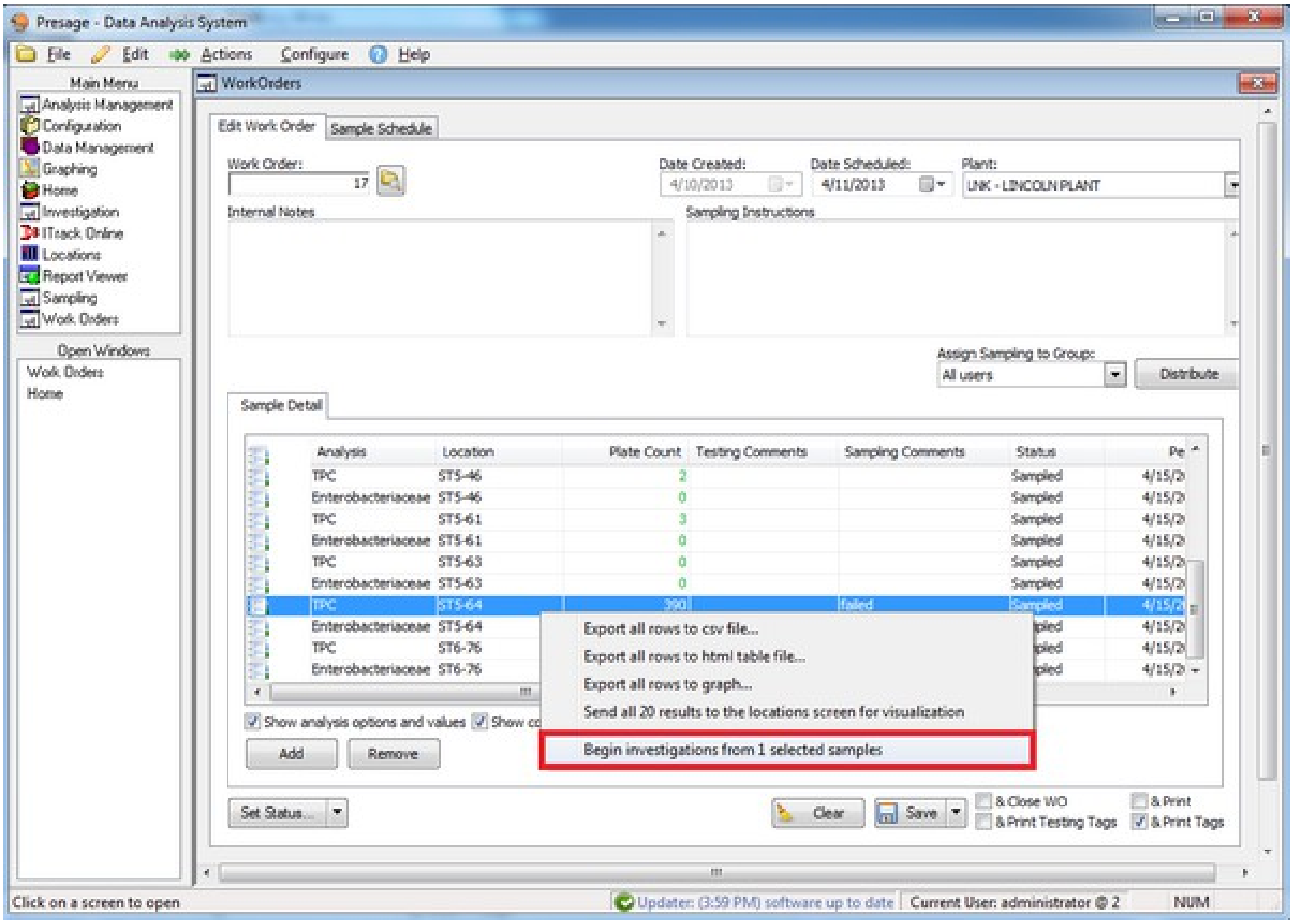
How to invalidate a sample.

In the work order screen, select the row, click on the arrow next to **Set Status...** on the bottom-left corner. Hover over **Only Selected Items** or **All Items** then click on **To Cancelled...**.

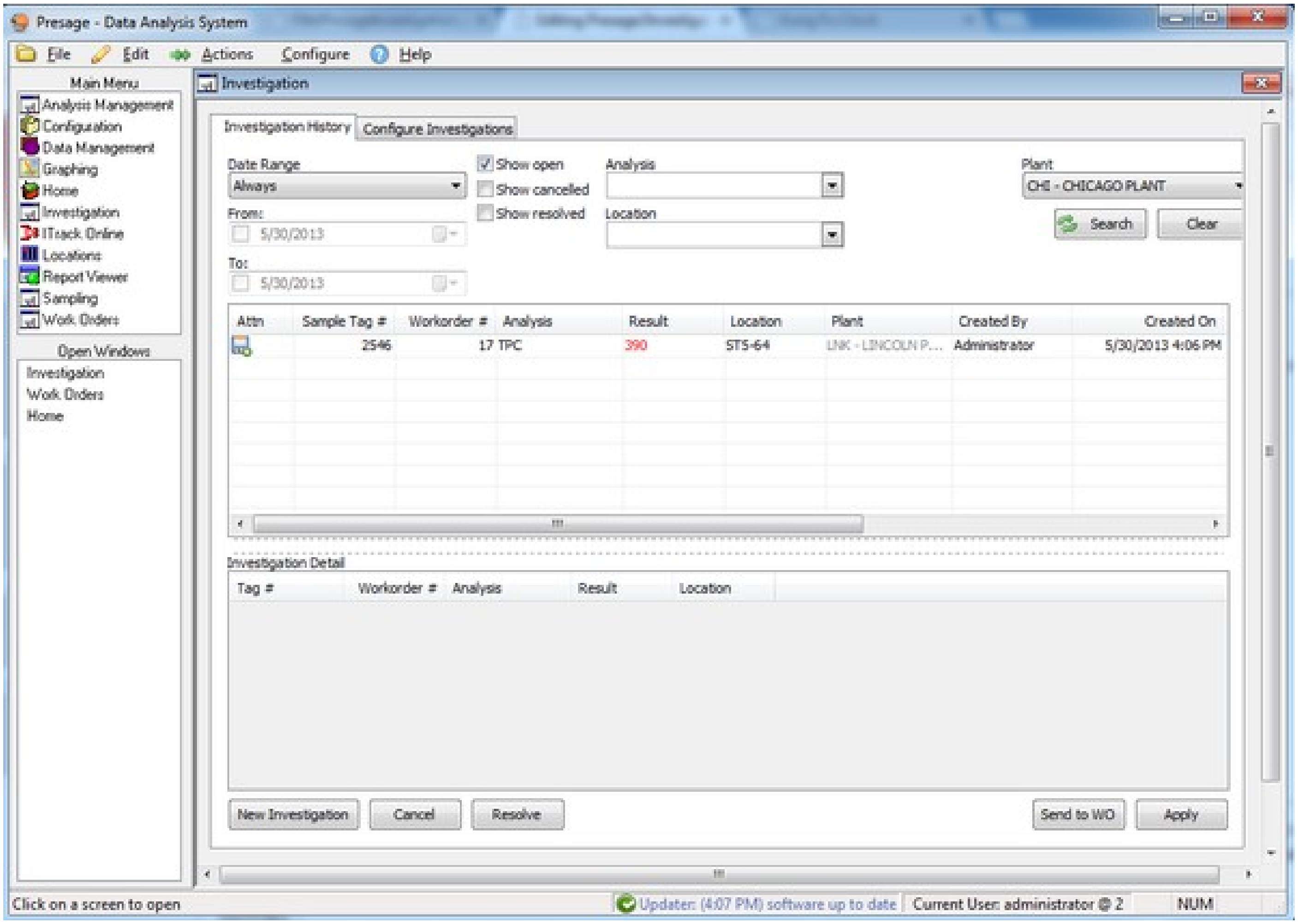
Invalidated samples do not show in the Sample History.

How to send an analysis for investigation.

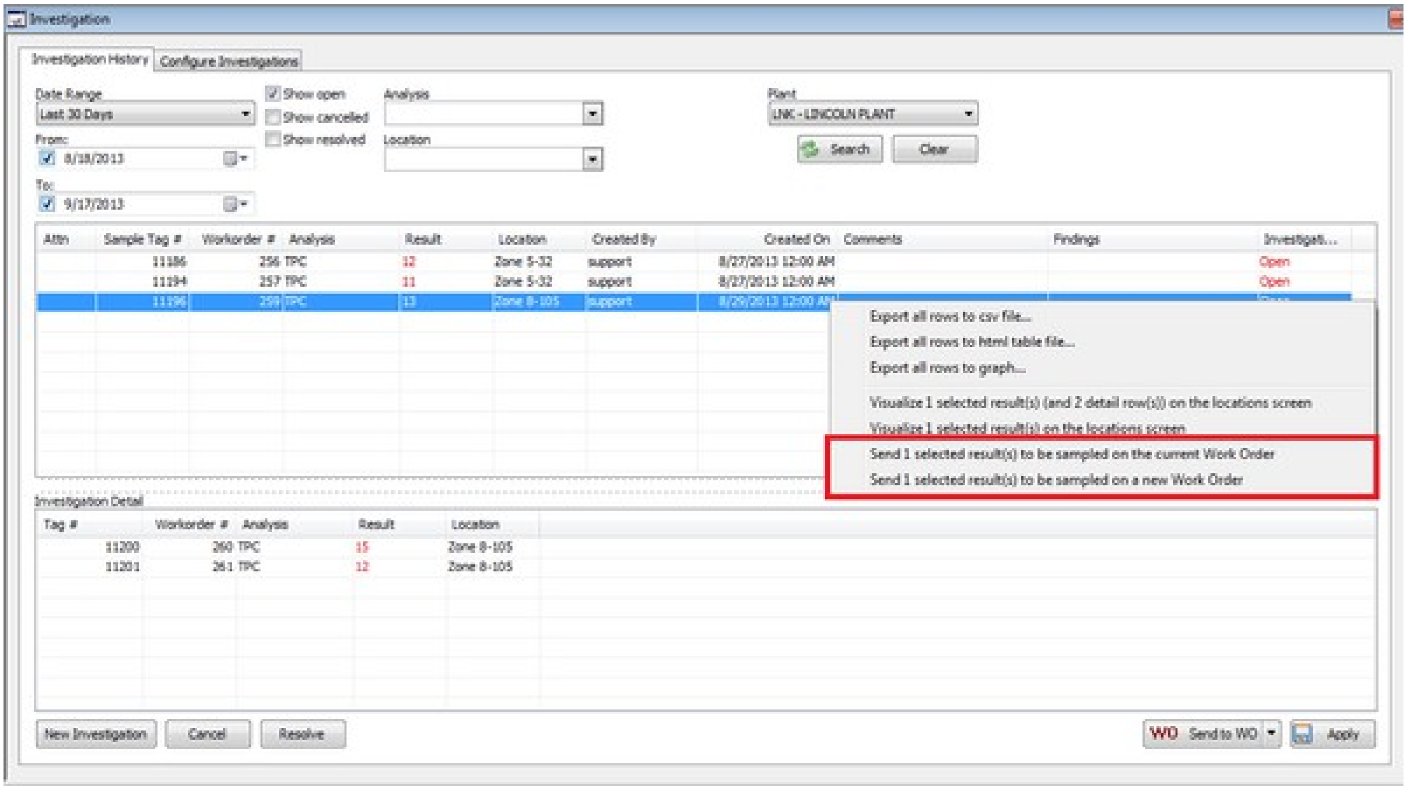
To send an analysis for investigation, select the sample analysis from the *Work Order* screen or *Sampling History*; right-click on it, and select **Begin investigation from # selected samples**.



User is automatically taken to the *Investigation Screen*.



Right-click on the analysis on this page to see multiple action options: view the location visualization or send to the work order. There is also a **Send to WO** button on the lower, right-hand portion of the screen. Click on the arrow next to the **Send to WO** button, choose between *Send to Existing WO* and *Send to New WO*.



Drag the bar to the right to see the column titled *Comments* and *Findings*. Type in the findings and corrective actions taken in the space provided.

If there are more fails related to this investigation, send the next failed analysis to the investigation screen. This fail will be listed in the *Investigation Detail* (lower) box. Make sure to send the initial analysis listed in the upper box in the *Investigation Screen* to the WO for each retest.

How to set up a sample.

1) Click on **Analysis Management** under the *Main menu*. Stay on the first tab *Analyses*.

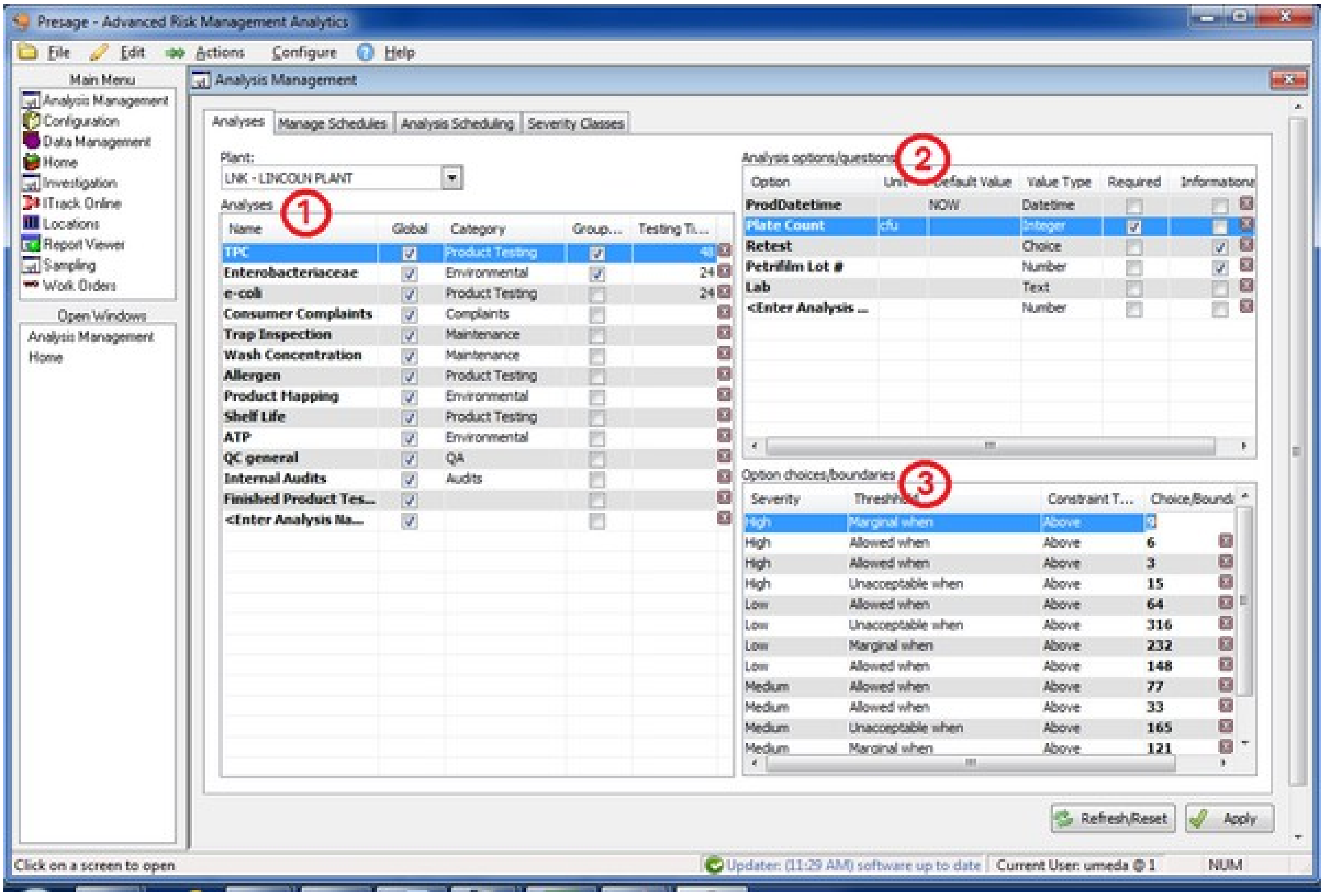
2) To create a new analysis, click on the box *<Enter Analysis Name>*. Enter the analysis name on the box provided. In the next column, type in the category name. This column is used to group analyses. *Testing Time* is for lab use. Type in the number hours the sample must rest in the incubator.

3) Under *Option*, enter what is being tested for; for example: plate count, sample area, detect-ability, etc.

When appropriate, enter the unit in the next column; for example: cfu, ppm, etc.

The **Informational** box is for options which don't need to be graphed. Check if appropriate.

Then choose the *Value Type* from the drop-down list provided. For example, plate count is measured in number, and qualitative analyses are measured in boolean: positive or negative. To create a drop-down list of various options to select from, choose *Choice*.



An analysis disappeared from Analyses screen, and I did not delete it.

If an analysis has a global setting, then it is used by all plants within a company. If one plant unchecks the global setting, then the analysis is hidden for all other plants except for the plant that set the analysis to be plant-specific.

Note: This does not mean the analysis and all data attached to it is deleted for all other plants. It is simply hidden.

Make sure another plant has not unchecked the global setting in the *Analysis Management-Analyses* screen.

If not, call the customer support rep at 1.800.309.1704.

How to set up a threshold.

1) Click on **Analysis Management** under the *Main Menu*. Stay on the first tab *Analyses*.

2) Select an analysis and a specific option you want to set thresholds or specifications to. Set thresholds for each option separately.

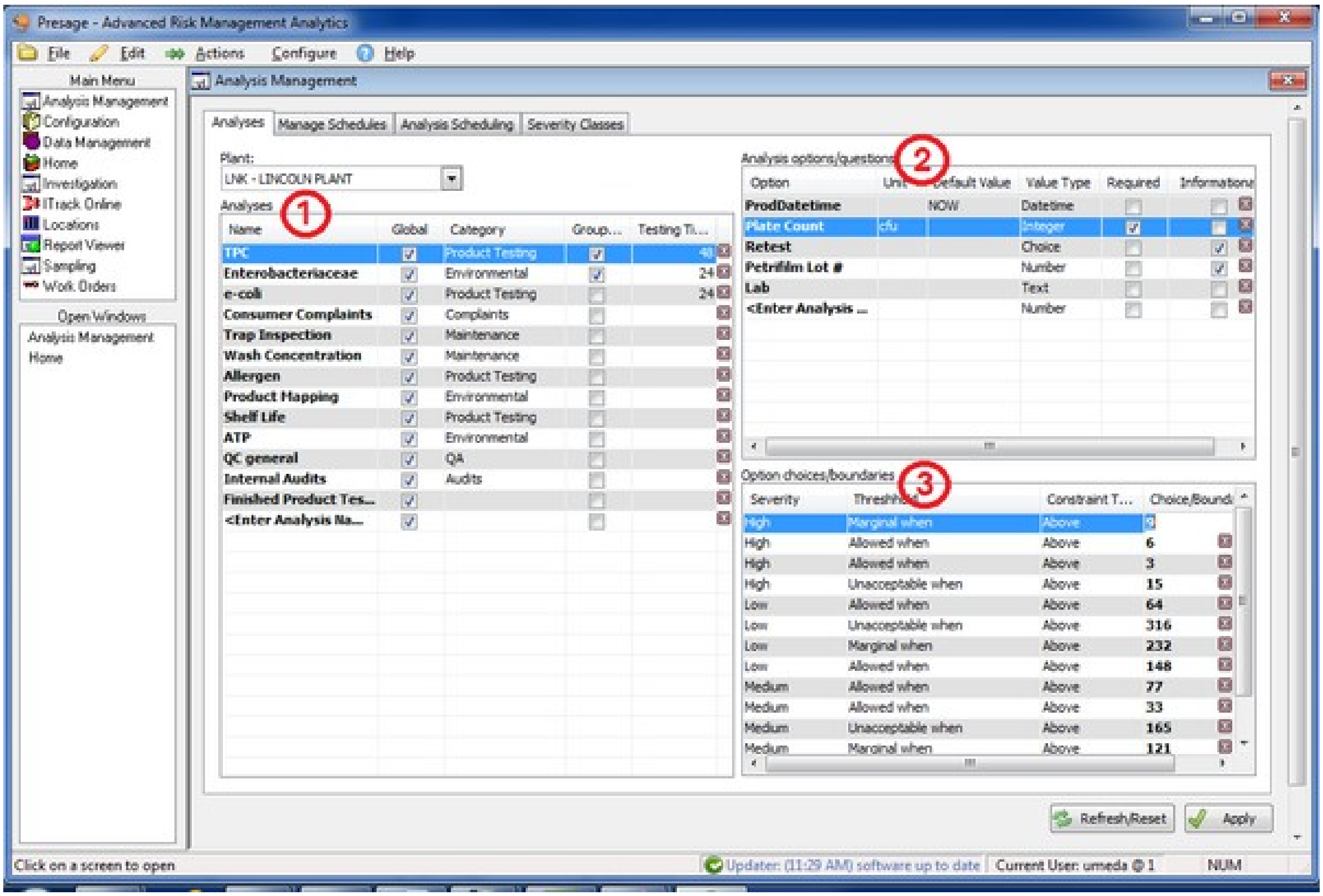
3) Select the severity from the drop-down list. To set up severity classes, refer to *Analysis Management: Severity Classes*.

4) For *Threshold*, select an option from the drop-down list. Marginal values are those which are acceptable, but are nearing the unacceptable boundary. Unacceptable values represent failure. Invalid values signify an error.

5) *Constraint Type* sets the minimum and maximum boundary value.

6) Choose the appropriate *Choice/Boundary Value*, or fill it in manually. For example, to set 10 cfu as a maximum acceptable limit for an analysis, choose *Unacceptable when* under *Thresholds*; *Above* under *Constraint Type*; and type in *10* under *Choice/Boundary*.

7) Click **Apply** to save changes.



How to make thresholds plant-specific for global analyses.

Select a plant-specific *Severity* or *Product* with each threshold and make sure the **Global** box is not checked.

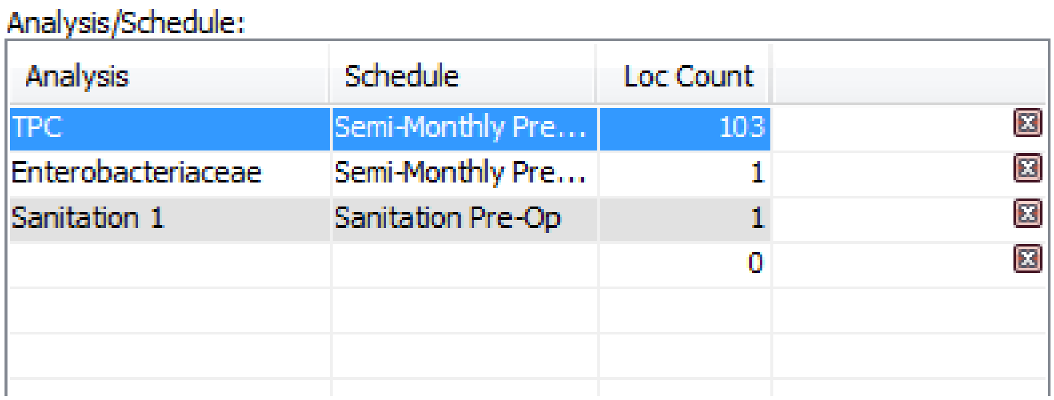
How to set up a schedule.

1) First select the plant from the drop-down list.

2) Under the column title *Analysis*, select the analysis type from the drop-down list. This list includes all analyses created under *Analysis Management: Analyses*.

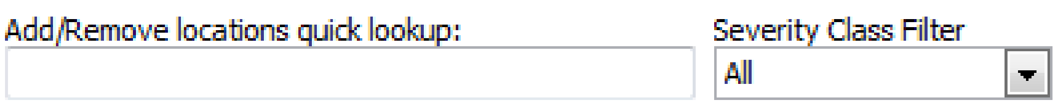
3) Select the schedule period interval from the drop-down list under Schedule. This list is created under *Analysis Management: Manage Schedules*.

4) The *Location Count* is automatically filled out when the locations are selected in the next step.

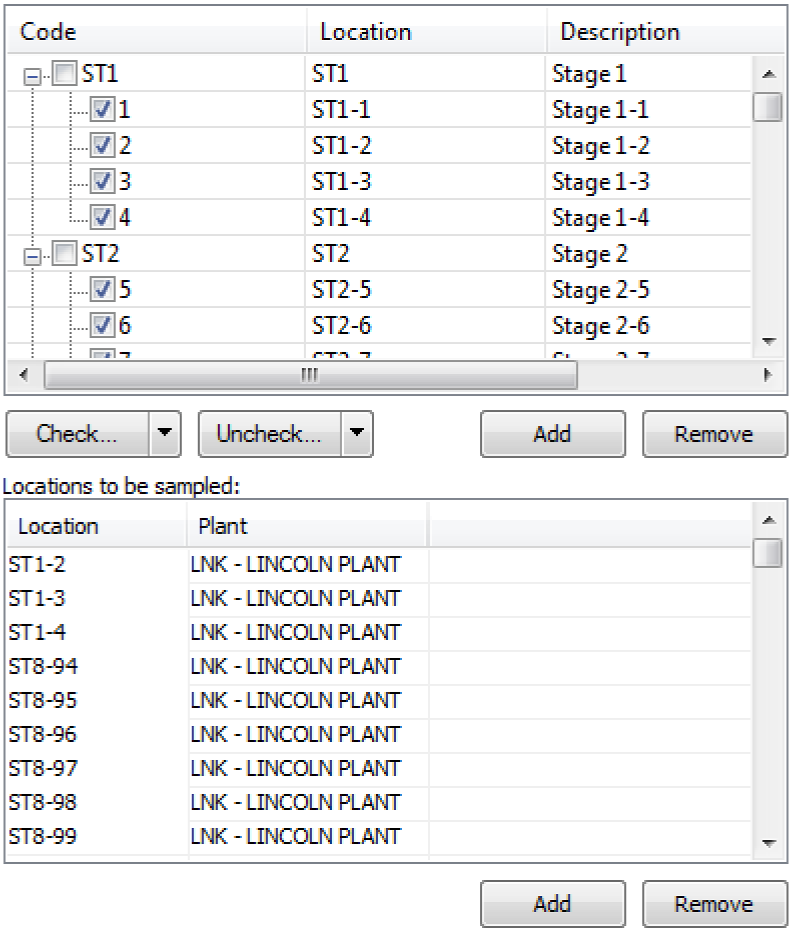


5) The right side of the screen is for choosing locations to perform the analyses on. Select one analysis under *Analysis/Schedule:*. Then check the box next to the locations which need to be sampled.

6) For a quick look-up, type in the first letter or number of the display code for the location. The corresponding locations will show up in the display below. The list can also be filtered by severity class. Choose the severity class from the drop-down; the list in the display will narrow down to locations with the severity class specified.



7) Use the **Check...** button to select groups of locations. Use the **Uncheck...** button to deselect groups of locations. Click **Add** (on the middle of the screen, not the one on the lower portion of the screen) to add the selected locations to the list of locations to be sampled. Click **Remove** to take out locations from the list.



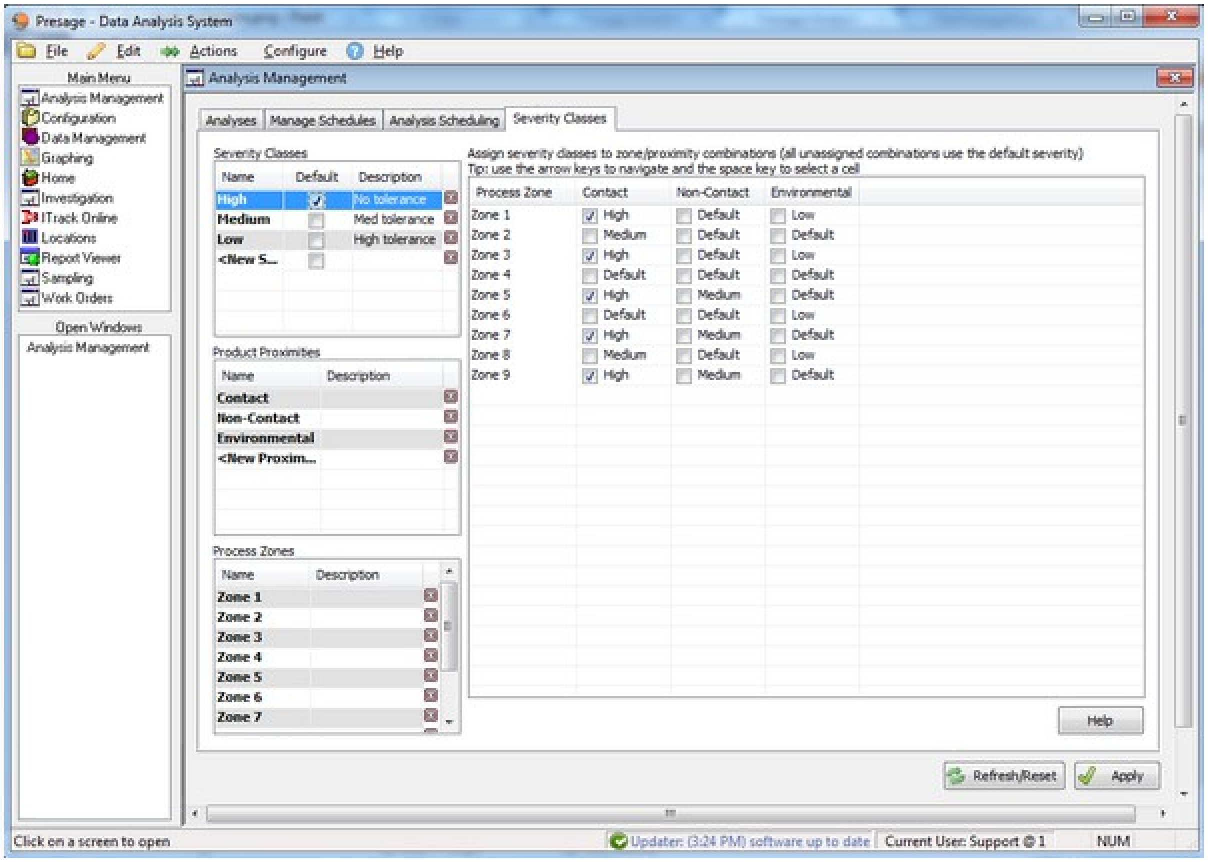
8) To add locations one at a time, use the **Add** button on the lower portion of the screen under *Locations to be sampled:*. To remove a location, select a location from the list and click **Remove**. To remove multiple locations, hold down the *Ctrl* button on keyboard while selecting multiple locations, then click **Remove**.

9)Click **Apply** to save any changes and use the **Refresh/Reset** button to remove any unsaved changes.

How to set up a severity/proximity/zone matrix.

1) Click **Analysis Management** under the *Main Menu*.

2) Click **Severity Classes**.



3) Give a unique names to all severity classes. Severity classes are based on limits which are set based on product proximities and process zones. Examples are High, Medium, and Low; or 10, 100, 500. Use terminology or numbers which best fit your system.

4) Click on the box under *Default* (may have to expand the column to see the full column title), to set a default. Default severity class is usually the one used the most often.

5) *Product proximity* refers to contact or non-contact surfaces. To add new product proximities, click on *<New Proximity>*. Type in the new proximity title. To remove product proximity, click on the red box with a white X to the right of each line. To rename, select the appropriate row and type in the new proximity name.

6) *Process zones* are different operational zones such as Chilling or Packaging. To add new zones, click on *<New Zone>* (It may be necessary to scroll down to see this row.) To delete a zone, click on the red box with white X to the right of each line. To rename, select the appropriate row and type in the new zone name.

7) To assign severity classes to zone/proximity combinations, click on one severity class under *Severity Classes* on the left display to highlight it, then check the boxes where the selected severity class is appropriate on the right display.

Process Zone	Contact	Non-Contact	Environmental
Zone 1	☑ High	☐ Default	☐ Low
Zone 2	☐ Medium	☐ Default	☐ Default
Zone 3	☑ High	☐ Default	☐ Low
Zone 4	☐ Default	☐ Default	☐ Default
Zone 5	☑ High	☐ Medium	☐ Default
Zone 6	☐ Default	☐ Default	☐ Low
Zone 7	☑ High	☐ Medium	☐ Default
Zone 8	☐ Medium	☐ Default	☐ Low
Zone 9	☑ High	☐ Medium	☐ Default

8) Now, click the next severity class to highlight it, then check the boxes where this severity class is appropriate on the right display. Not all boxes have to be checked.

9) Click **Apply** to save the changes.

Not able to mark a threshold as 'Global'.

To be able to mark a threshold or thresholds 'Global', no specific *Severity* or *Product* must be selected. *All Severities* must be selected under *Severity* and *All Products* must be selected under *Product* for each specific threshold (each row) for it to be able to be 'Global'.

How to add a plant.

1) Click **Configuration** under the *Main Menu*.

2) Click **Plants**.

3) Click **New Plant**.

4) For *Plan #*, select *New* from the drop-down list.

5) If the new plant is private, check the box provided. With this option, users must log in to have access to the new plant.

6) The *Display Code* will appear in the *Plant #* drop-down list after the new plant has been created. Choose a unique code which will best identify the plant.

7) *Company Name* refers to the business name a particular plant is associated with.

6) Fill in all information in the spaces provided.

How to delete a plant.

1) Click **Configuration** under the *Main Menu*.

2) Click **Plants**.

3) To delete a plant, choose the plant from the *Plant #* drop-down list.

4) Click **Delete Plant**.

How to add a user.

1) Click **Configuration** under the *Main Menu*.

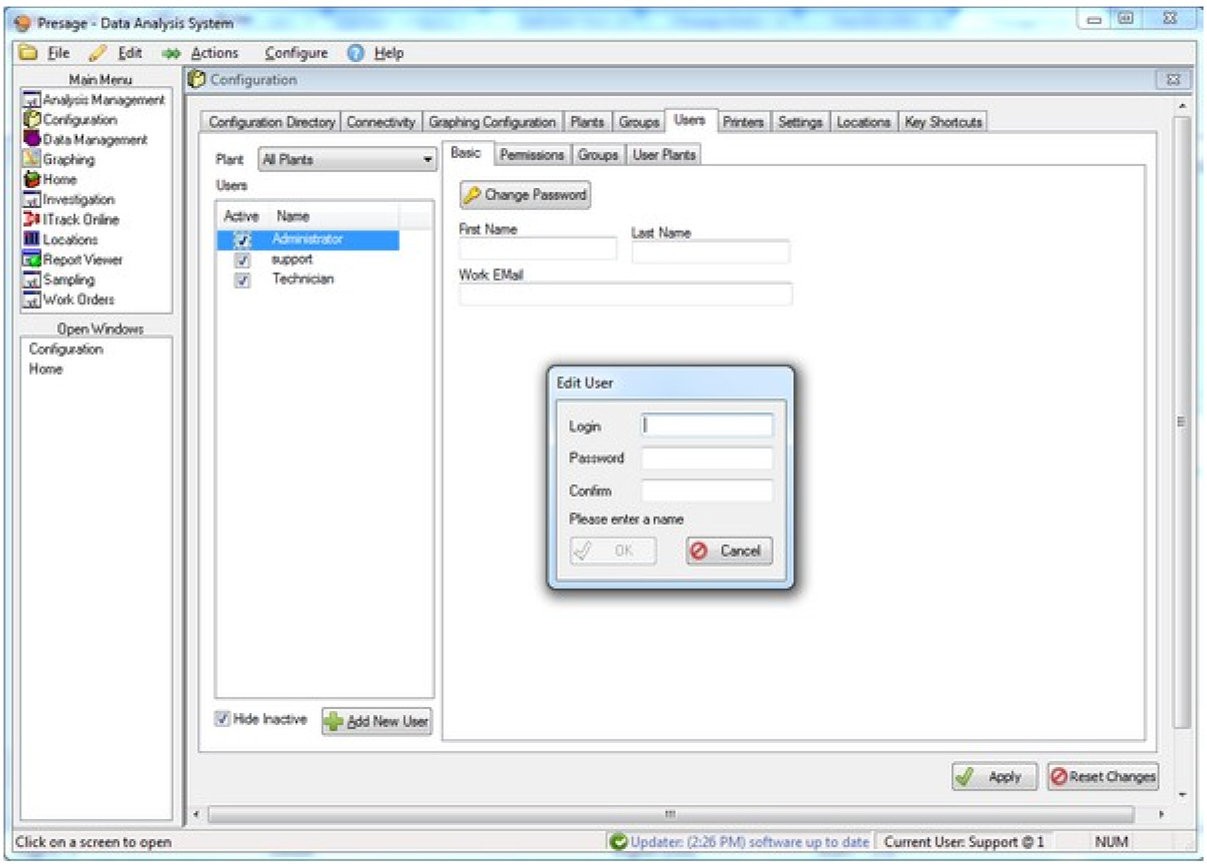
2) Click **Users**.

3) Click **Add User**.

4) Enter login name and password, and re-enter password to confirm in the pop-up box.

5) Click **OK**.

Note: Password is case sensitive, but the username is NOT case sensitive. Choose a password that is easy to remember but difficult for others to guess. A good password has a combination of both letters and numbers.



How to delete a user.

In Presage, there is no way to completely delete a user, however users can be inactivated and hidden from the users list.

1) Click **Configuration** under the *Main Menu*.

2) Click **Users**.

3) Uncheck the box next to the users' names.

4) Check *Hide Inactive* box to take inactive users out of the users list.

How to change a user password.

1) Click **Configuration** under the *Main Menu*.

2) Click **Users**.

3) Select the username.

4) Click **Change Password** under the Basic Tab.

5) Enter and re-enter password.

Note: Password is case sensitive. Choose a password that is easy to remember but difficult for others to guess. A good password has a combination of both letters and numbers.

How to create groups.

1) Click on **Configuration** under the *Main Menu*.

2) click on the **Groups** tab.

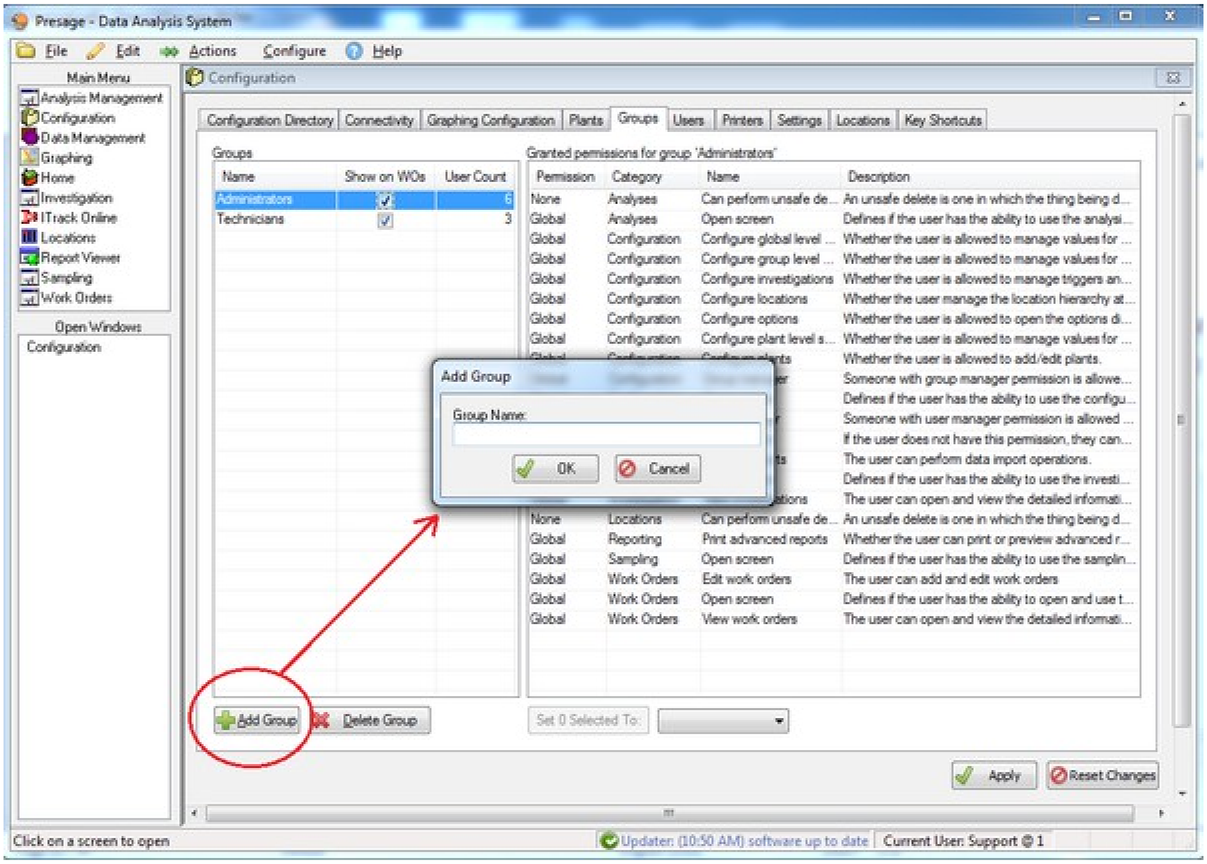
3) Click **Add Group**.

4) Type in the new group name.

5) Click **OK**, then **Apply**.

6) Check the box under *Show on WOs*, if this group will perform tasks in the work orders.

User Count shows the number of users in the group.



How to set permissions to groups.

1) Click on **Configuration** under the *Main Menu*.

2) Click on the **Groups** tab.

Permission settings are *none* (not permissible), *plant* (permissible only for this plant), and *global* (permissible for every plant).

3) Scroll to the right to see complete descriptions for each permission.

4) Click on the group name to highlight it.

5a) To change permission levels individually for each category, click on each row in the *Grant* column, a drop-down list of permission levels will appear. Make a selection.

5b) To change permissions for multiple categories at once, hold down the **Ctrl** key on the keyboard and select desired rows. The first button directly below the permissions list will update the number of rows selected. Click on the blank button next to it, a drop-down list with permission levels will appear. Make a selection, then click the updated **Set # Selected To:** button to left of it.

6) Click **Apply** to save changes.

Note: Users get the best of user and group permission levels. For example, if the user is assigned global permission on the user level, but plant permission for the group level, then that user gets the global level for that specific action.

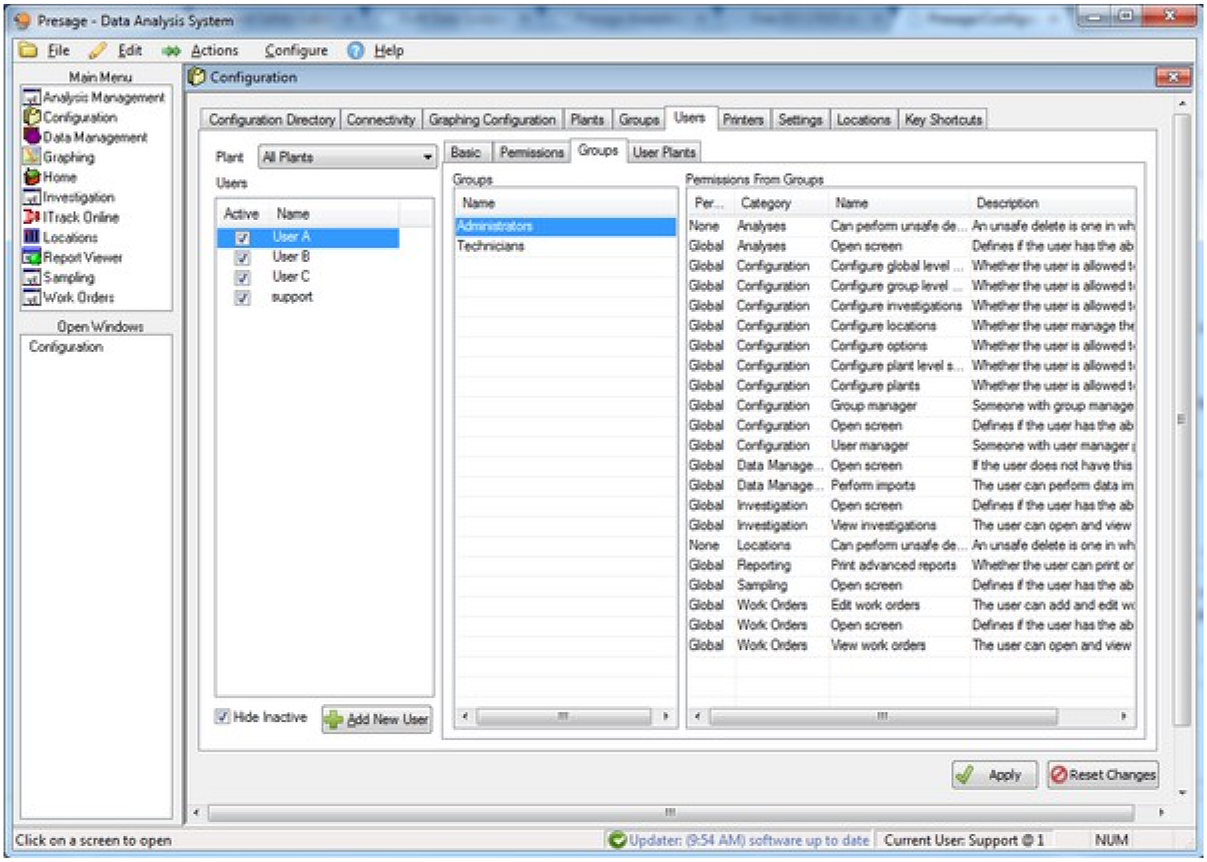
How to assign users to groups.

1) Click on **Configuration** under the *Main Menu* Then click on the **Users** tab.

2) Click on **Groups** which is the third sub-tab under the **Users** tab.

3) Click on the user's name to highlight it, then click on the group name.

4) Click **Apply**.



How to set permission for each user.

Go to **Configuration**, then **Users**.

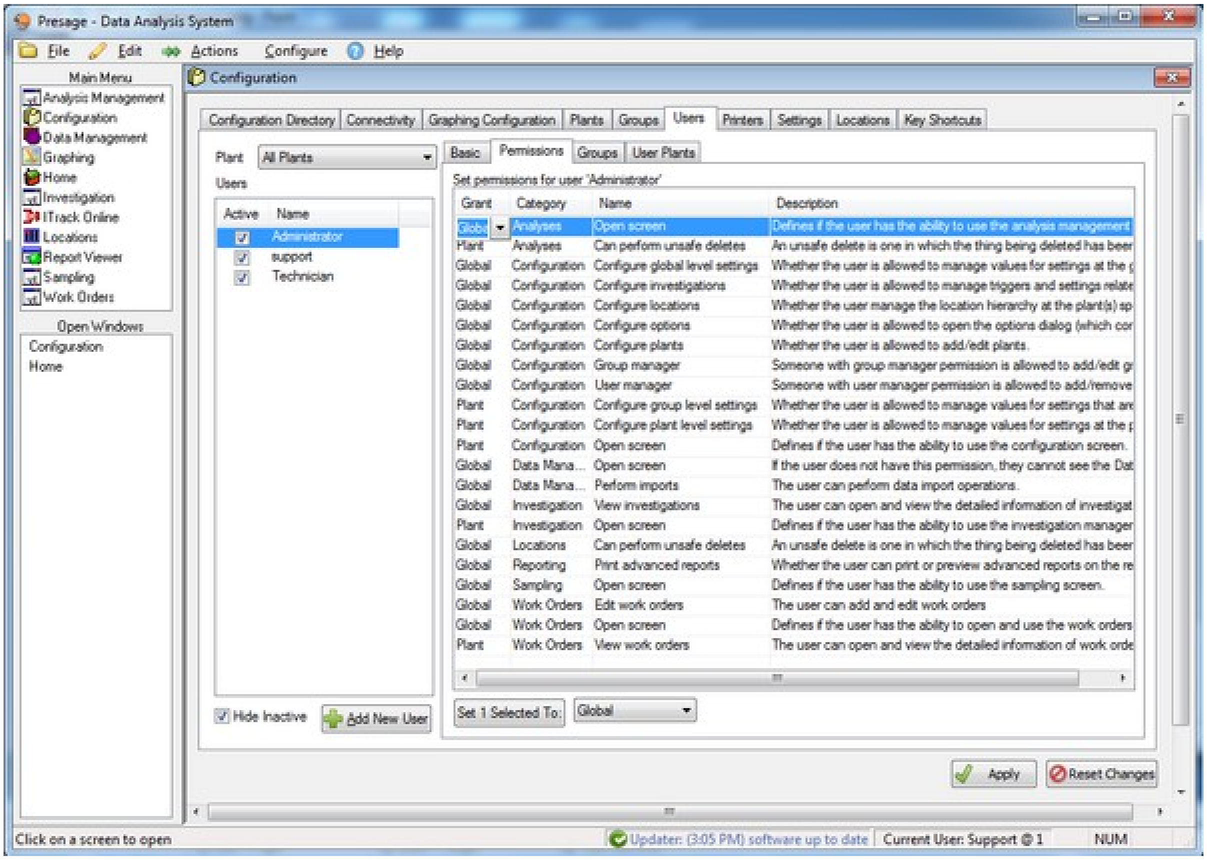
Click on the user's name to highlight it, then click on *Permissions* sub-tab.

To change permission settings individually for each category, click on each row in the *Grant* column, a drop-down list of permission settings will appear. Make the desired selection.

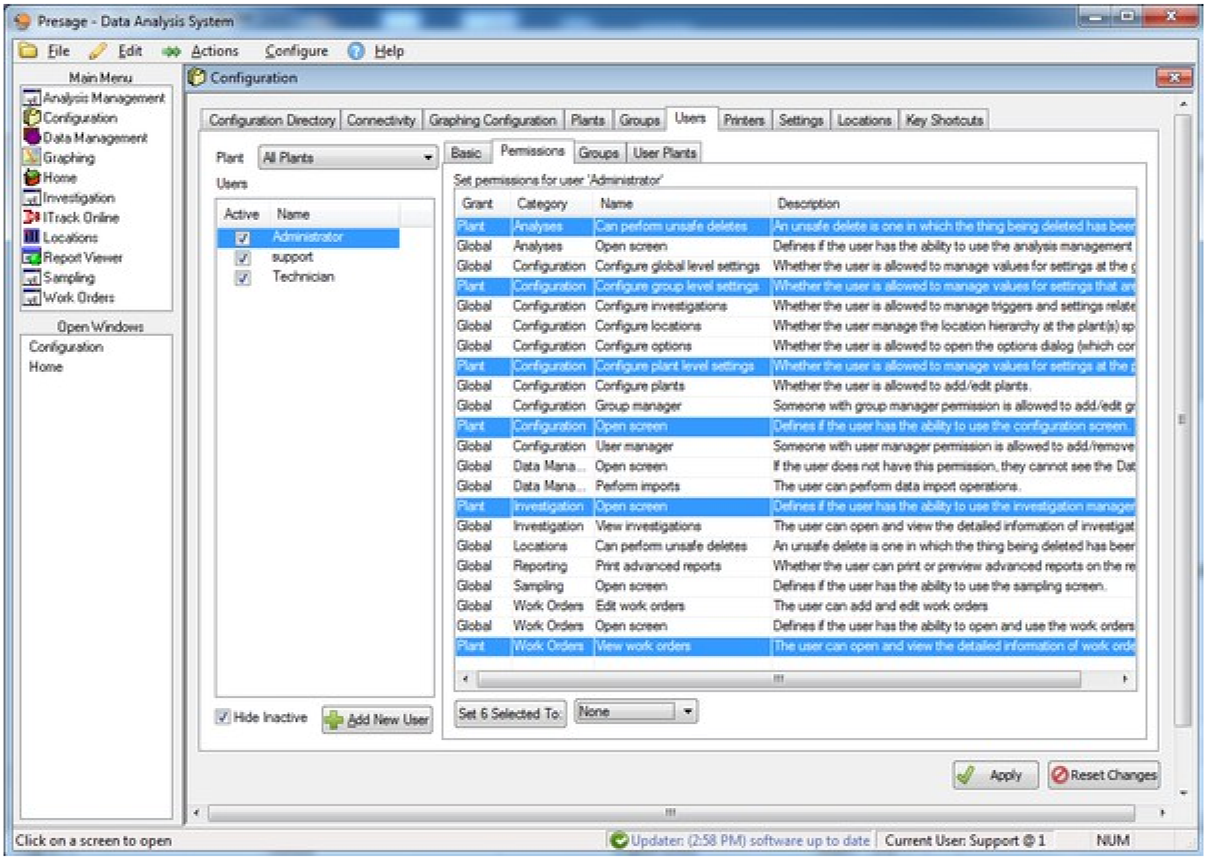
Permission settings are *none* (not permissible), *plant* (permissible only for this plant), and *global* (permissible for every plant).

Scroll to the right to see complete descriptions for each permission.

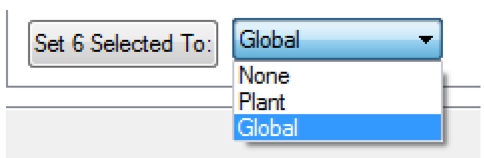
Click **Apply** to save changes.



To change permissions for multiple categories at once, hold down the **Ctrl** key and select desired rows or **Ctrl-A** to select all.



The first button directly below the permissions list will update the number of rows selected. Click on the second button, a drop-down list with permission settings will appear. Make the desired selection, then click the updated **Set # Selected To:** button to left of it.



Click **Apply** to save changes.

User's user permission is different from his/her group permission.

In this case, the user is given the best of both permission levels. For example, if the user permission for deleting data is set to plant-level and the group (which the user is assigned to) permission for the same task is set to global-level, then the user gets the global permission.

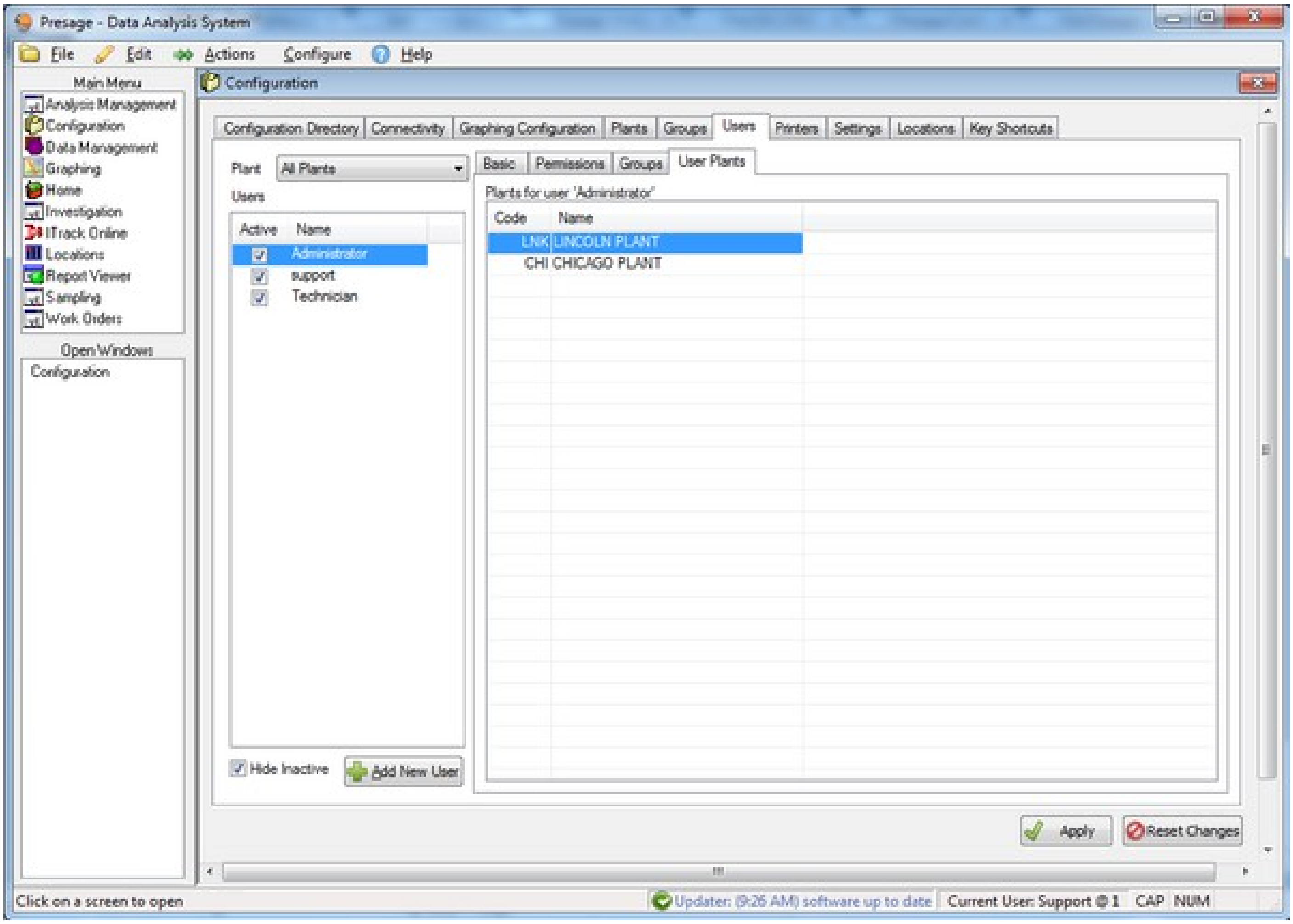
How to assign users to plants.

1) Click on **Configuration** under the *Main Menu*.

2) Click **Users** tab then **User Plants** sub-tab.

3) Click on the user name, then select the plant name(s).

4) Click **Apply**.



How to configure printer settings.

1) Click on **Configuration** under the *Main Menu*.

2) Click on the **Printers** tab.

3) *Report Type* drop-down list includes reports with various dimensions used by the plant. Select one.

4) *Printer Name* is the printer name (identification code) used to print the report. Select the appropriate printer used for the report dimensions selected.

The Driver and Port information will be filled automatically.

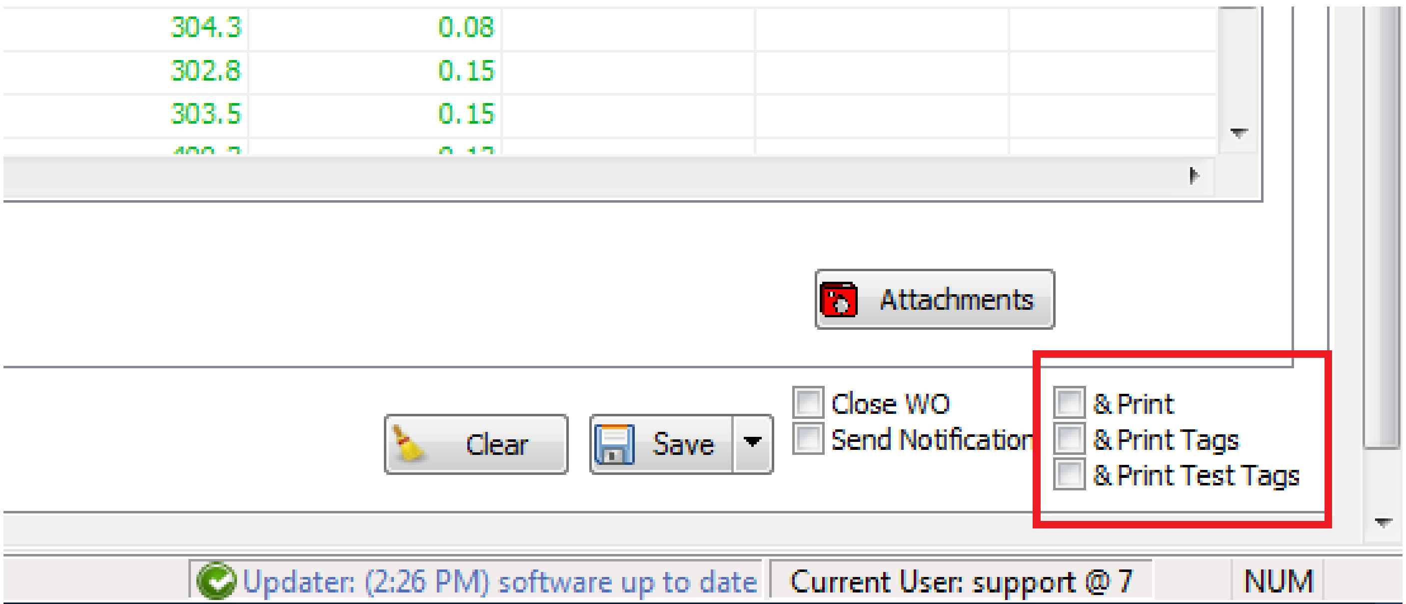
5) Click **Apply**.

If there are multiple report types, follow the steps above to attach a report type to a printer name. Click Apply after each setting.

How to print labels.

After adding all the samples in the work order, check the appropriate **&Print** box. Then click **Save**.

&Print prints letter size reports. **&Print Tags** has two options: with Barcode and without Barcode. Usually it's a 1.1 *x 2.4* label, but it can be customized to your needs. **&Print Test Tags** prints very thin labels with barcodes. These are meant for sticking on petri dishes and petri films.



How to print a report.

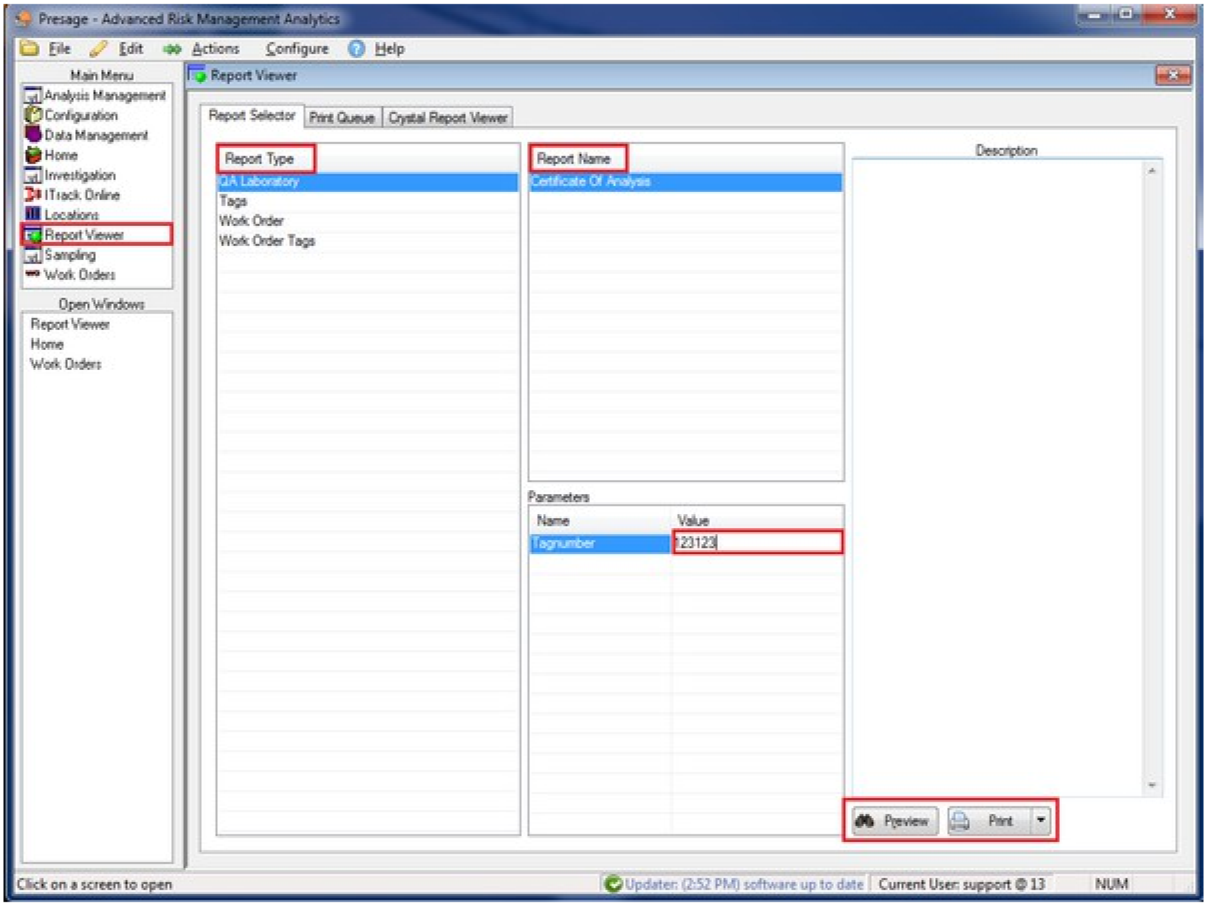
1. Click **Report Viewer** under *Main Menu*.

2. Select the report type.

3. Select the report name.

4. Enter a parameter value next to the parameter name listed.

5. Click **Preview** to view the report, or click **Print**.



How to narrow down history search using option filters.

1. Click **Sampling** under *Main Menu*.

2. Check the box next to the plant you want to pull up data on.

3. Check the box next to the analysis you want to search for.

4. Check the box next to the option you want to search for.

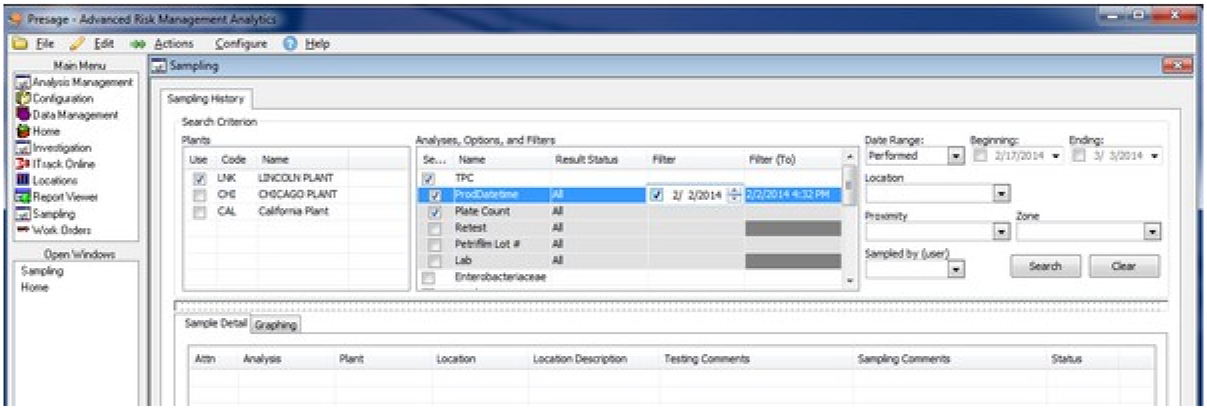
5. Click on the field under *Result Status* and select a status from the list to narrow down search to a specific result status.

6. Click on the field under *Filter* to type in or select a value to narrow down search. If the option is set up to be a choice, this field will be a choice. If the option is set up to be a date, then select an appropriate date. If the option is set up to be a number, type in a number to filter.

7. Make appropriate Date Range, Location, Proximity, Zone, and User selections.

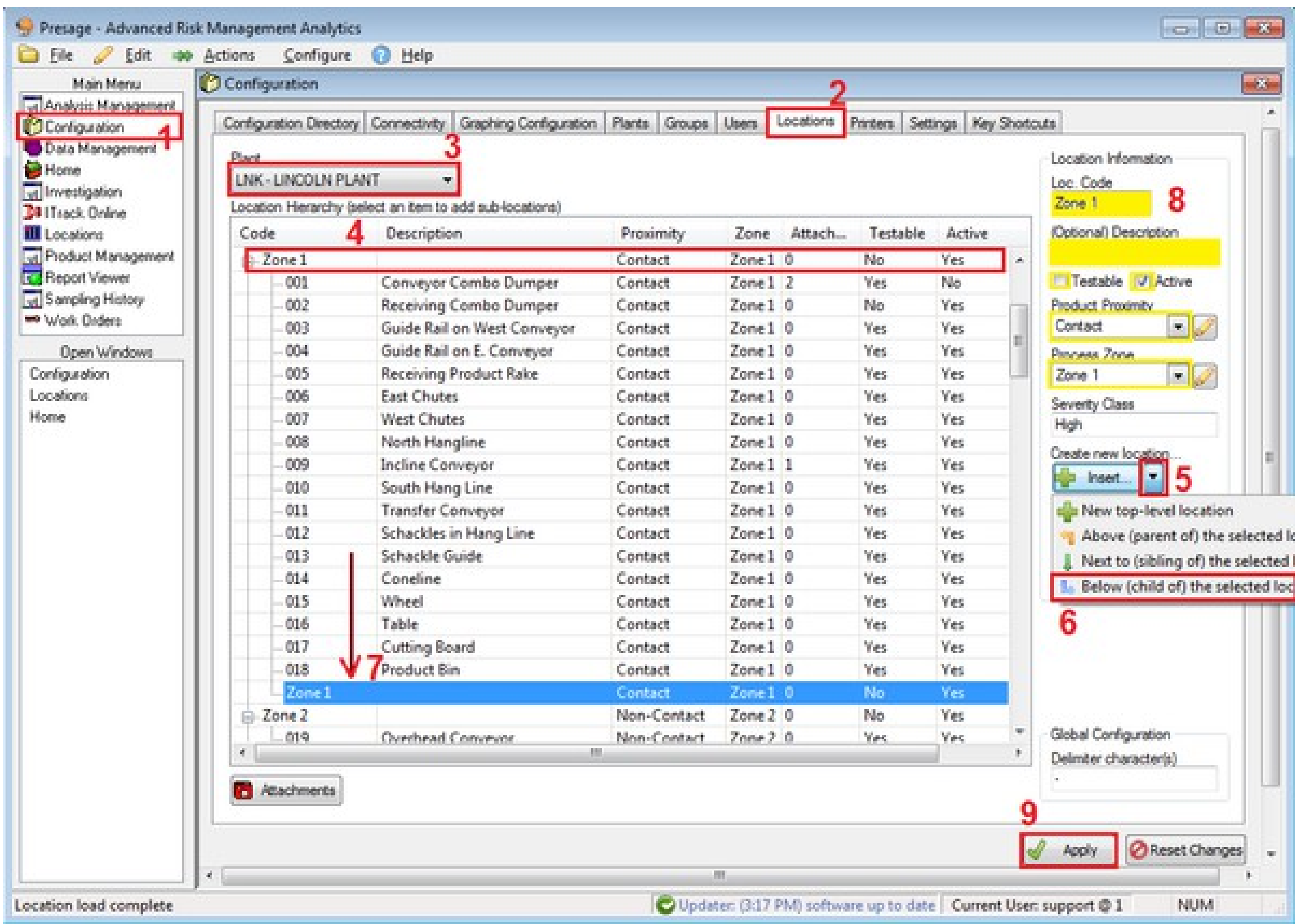
8. Click **Search**.

9. Data will appear under the *Sample Detail* tab.



How to add locations.

To add locations or make changes to the location hierarchy, first select the appropriate plant from the *Plant* drop-down list.



1. Click **Configuration** under the *Main Menu*.

2. Click **Locations** tab.

3. Make sure the correct plant is selected.

4. Select an existing top location.

5. Click the down-arrow next to the **Insert** button.

6. Select a position (which applies to the location selected in step 4).

7. New location appears on the bottom of the list.

8. The *Location Information* is pre-filled with the selected location's (in step 4) information. Enter the correct *Loc Code*, *Location Description*, *Proximity*, and *Zone*. Make sure to check the *Testable* and *Active* boxes when appropriate. The *Severity Class* is filled automatically based on the process zone selected.

9. Click **Apply**.

How to delete locations.

1. Click **Configuration** under the *Main Menu*.

2. Click **Locations** tab.

3. Make sure the correct plant is selected.

4. Select an existing location.

5. Click **Delete**.

6. Pop-up will appear with the number of samples attached to this location. **Please note that these samples will be permanently deleted along with the location and will not be replaced.**

7. Click **Apply**.

How to edit locations.

1. Click **Configuration** under the *Main Menu*.

2. Click **Locations** tab.

3. Make sure the correct plant is selected.

4. Select an existing location.

5. Make changes under *Location Information*.

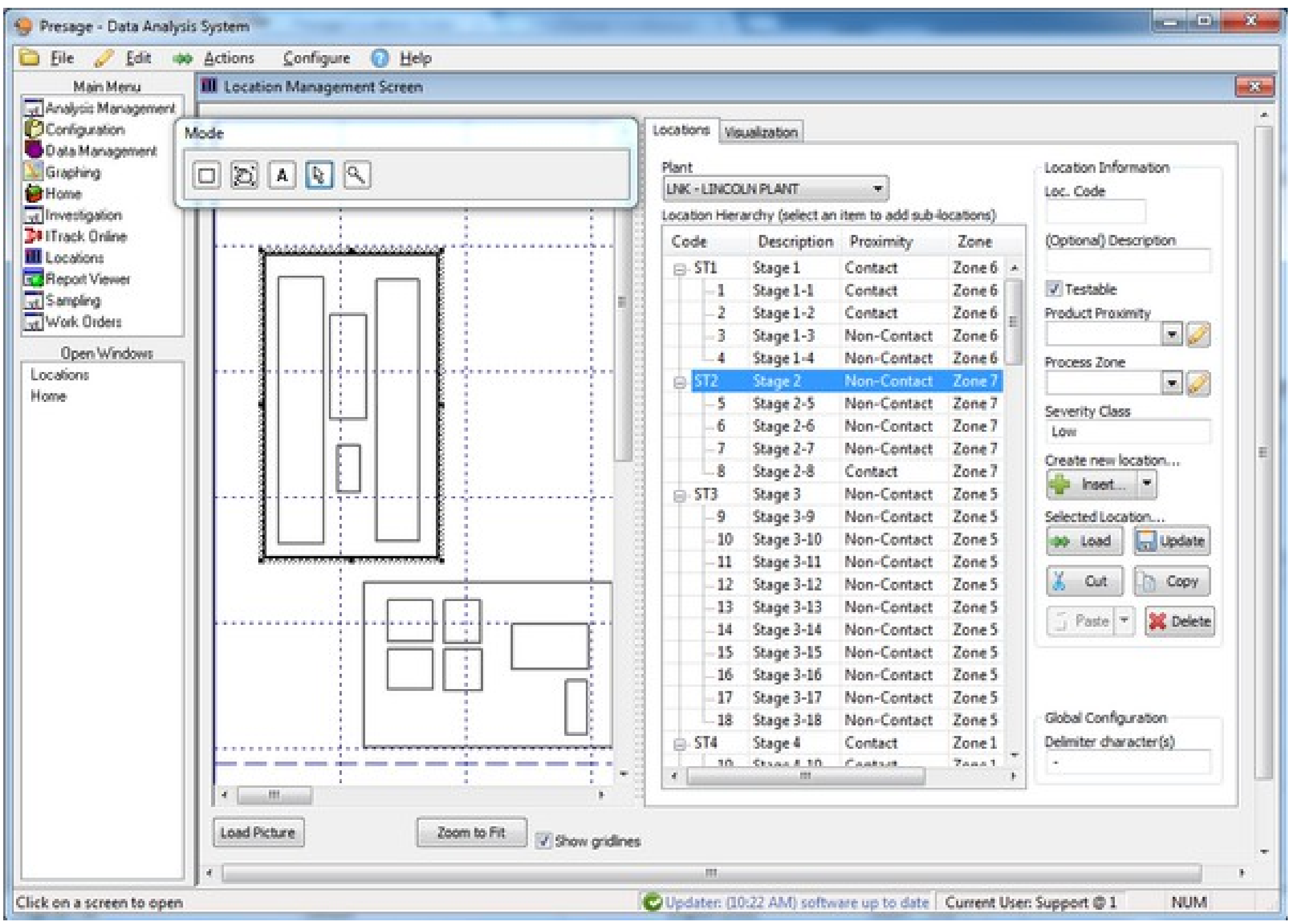
6. Click **Apply**.

How to rearrange locations list.

1. Click **Configuration** under the *Main Menu*.
2. Click **Locations** tab.
3. Make sure the correct plant is selected.
4. Select an existing location.
5. Click **Cut** or **Copy**.
6. Select the top level location you want to paste the cut or copied location.
7. Click the down-arrow next to the **Paste** button.
8. Select the position which applies to the selected location in step 6.
9. Location will appear on the bottom of the list.
10. Click **Apply**.

How to create a plant map.

First, select the plant from the *Plant* drop-down list in the upper right-hand portion of the screen.

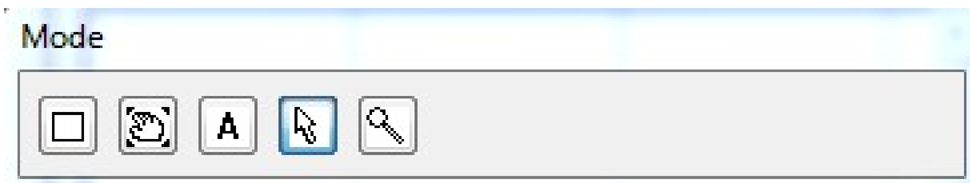


To upload a blueprint of the plant, click on **Load Picture**, locate and select the file, then click **Open**.

Click **Zoom to Fit** to make the entire map visible.

Uncheck the *Show gridlines* box to get rid of the background gridlines.

To map a location, select a location from the *Location Hierarchy*, then click on the button with a box on the *Mode* pop-up window. Click and drag cursor to place the box where appropriate. Now, the box and the location identification name are connected, and once the location identification name or code is selected, the coinciding box on the map will be highlighted automatically.



To add text, click on the box with the letter *A* then click and drag on the canvas to see the cursor appear.

Use the magnifying glass button on the *Mode* pop-up window to zoom in the picture or diagram.

Use the arrow button to rearrange and re-size boxes. Use the hand button to drag the canvas.

Tip: When several small squares are laid out very close to each other, it is easier to select one particular box by clicking on any box within the group and pressing Tab on keyboard to select other boxes within that area.

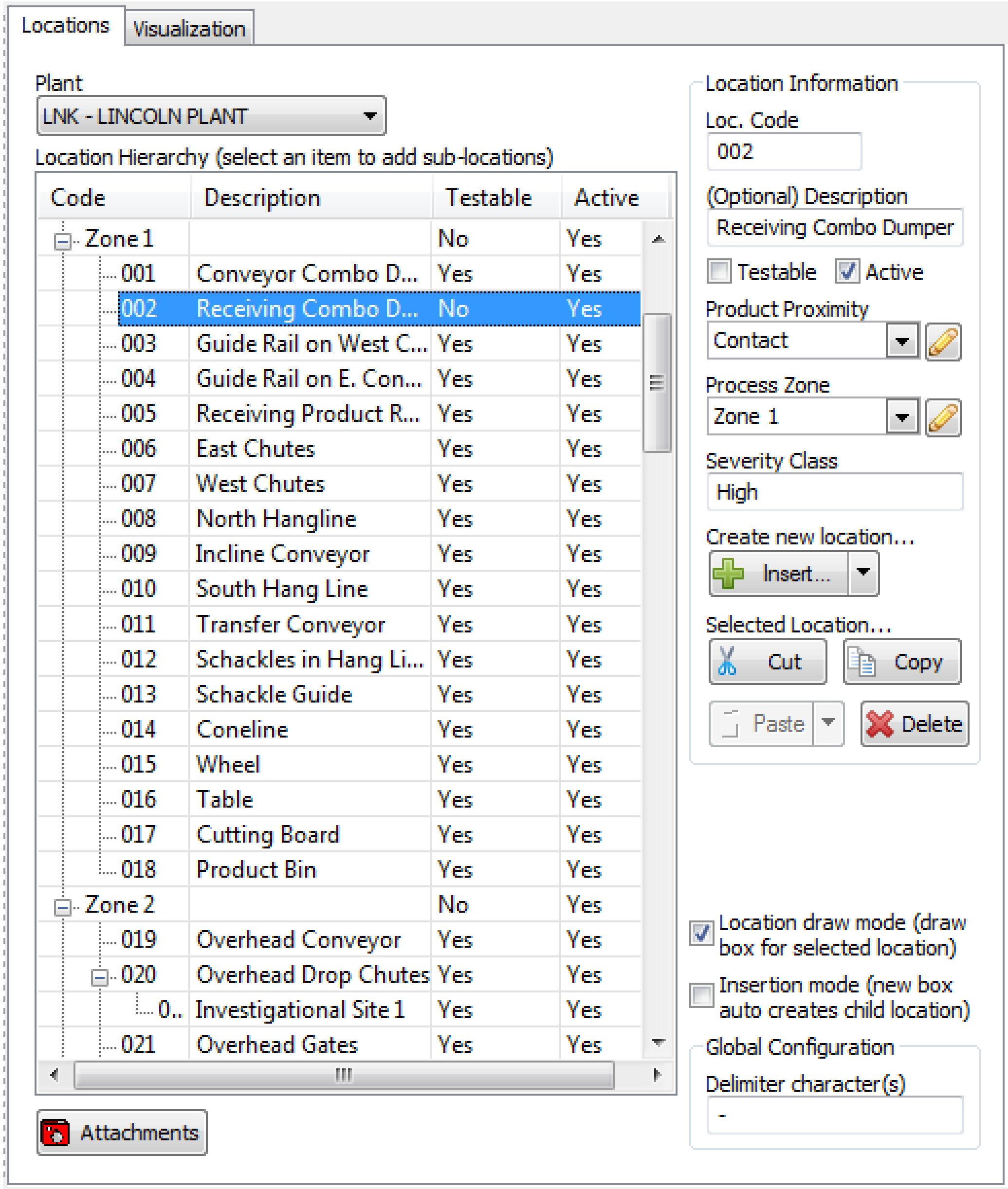
For guidelines on *Location Information*, refer to *Configuration: Locations*.

Why aren't my locations listed in the Location drop-down list in the Work Order screen?

When a location is marked as not testable (by unchecking the **Testable** button under *Location Information*), that location will not show up in the Location drop-down list in the work order.

Make sure to check the **Testable** box for all locations that get tested.

For example, in the location below (Loc. Code 002 : Receiving Combo Dumper), the **Testable** box is unchecked.



Why aren't all my testable locations not showing in the Location drop-down list in the Work Order screen?

The location drop-downs (for speed purposes) only load the first couple hundred items by default. Typing narrows down the list so you can find items that aren't shown.

How to add a product.

1. Click on **Product Management** under the *Main Menu*.

2. Select the appropriate plant from the *Plant:* drop-down menu.

3. Click an existing line in the *Product List*.

4. Click the down-arrow next to the **Add...** button.

5. Select an appropriate position for the new product: above, next to, or below.

6. A new selected line appears in the list. Fill in the appropriate fields: *Name*, *Category*, *Description*, and make sure to check the appropriate boxes: *Product is in use at the plant* and *Active Product*.

7. Click **Apply**.

How to delete a product.

1. Click on **Product Management** under the *Main Menu*.

2. Select the appropriate plant from the *Plant:* drop-down menu.

3. Click an existing line in the *Product List*.

4. Click **Delete**.

Note: Deleting a product means all data attached to it will also be deleted from the historical samples. An alternative is to *inactivate* the product.

5. A pop-up appears warning about the above note. Click **OK**.

6. Click **Apply**.

How to inactivate a product.

1. Click on **Product Management** under the *Main Menu*.

2. Select the appropriate plant from the *Plant:* drop-down menu.

3. Click an existing line in the *Product List*.

4. Uncheck the box next to *Active Product*.

5. Click **Apply.**

Note: To see this inactivated products, check the box next to *Show inactive products* directly below the *Products* tab.

How to assign a product to a plant.

1. Click on **Product Management** under the *Main Menu*.

2. Select the appropriate plant from the *Plant:* drop-down menu.

3. Click an existing line in the *Product List*.

4. Check the box next to *Product is in use at plant "X"*.

5. Click **Apply**.

How to attach an image to a product.

1. Click on **Product Management** under the *Main Menu*.

2. Select the appropriate plant from the *Plant:* drop-down menu.

3. Click an existing line in the *Product List*.

4. Click on the **Attachments** tab.

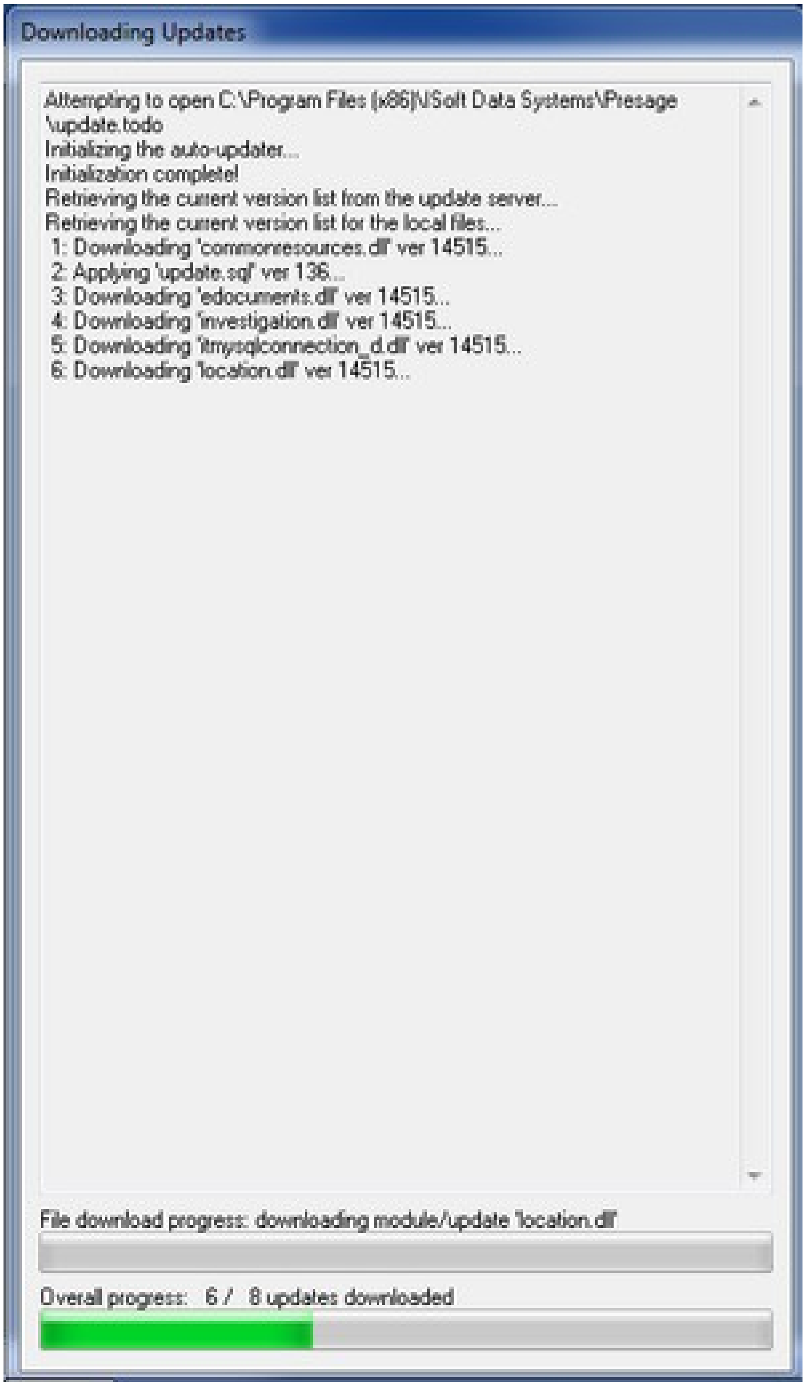
5. Click **Add**.

6. Select an image and click **Open** on the pop-up.

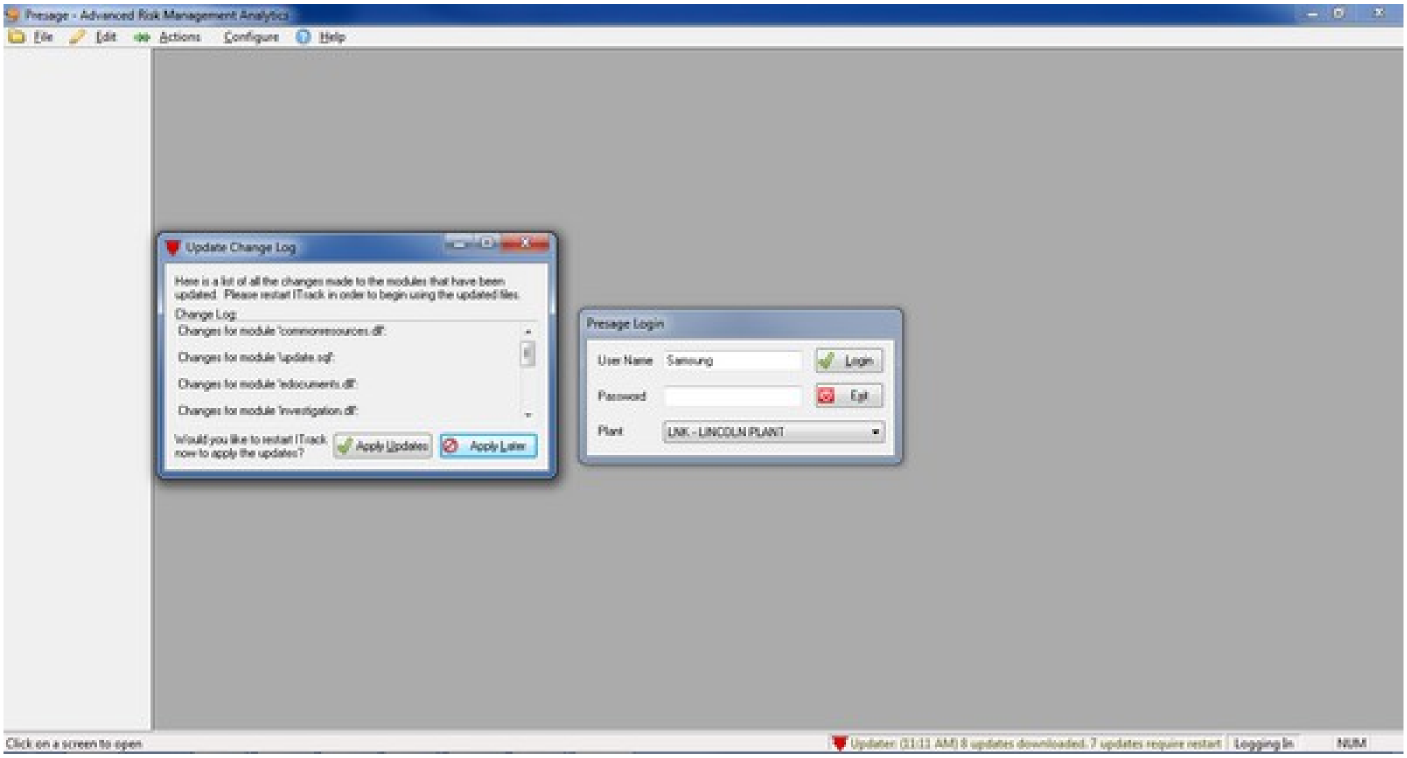
7. Click **Apply**.

How to apply updates.

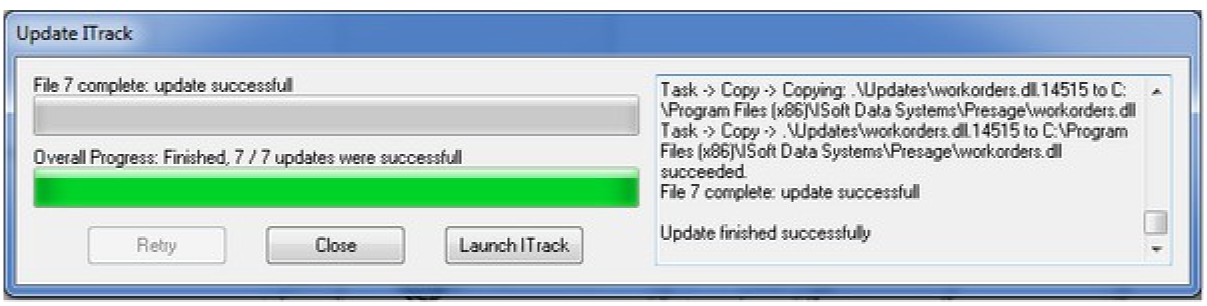
When updates have been released, the following dialog pops-up upon opening the Presage application.



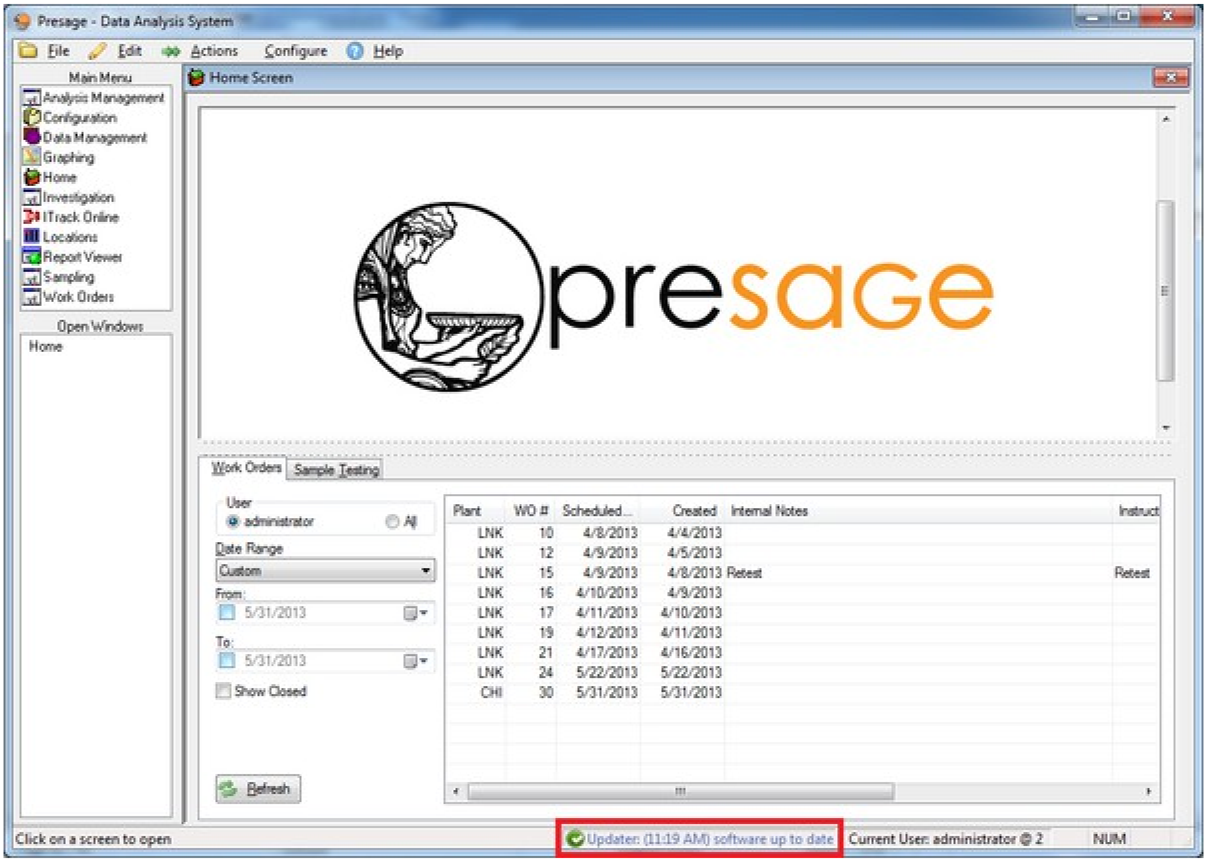
Once the following dialog pops-up, click **Apply Updates**.



Then click **Launch ITrack**.



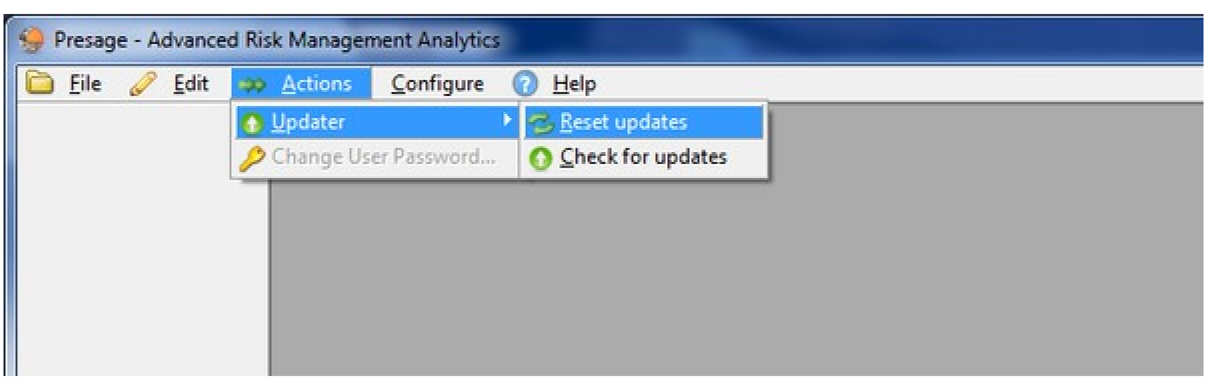
Note: If the updates have not been applied, the notification will appear on the very bottom-right panel of Presage.



Double-click on this notification to start the downloading updates process.

My update finished with errors. What do I do?

1. Reset the updates.



2. If the above does not work, call the Presage customer support at 1.800.309.1704.

Note: Presage can be used even if the updates have not gone through by clicking **Close** on the update pop-up dialog and logging in as usual.

How to send samples from the Sample Schedule to the work order.

1) Click on **Work Orders** under the *Main Menu*.

2) Click on the **Sample Schedule** tab.

3) Choose a plant from the drop-down menu on the right.

4) Choose the date range to view analyses scheduled for specific dates. For custom date range, select *Custom* from the drop-down menu. Then set the *From:* and *To:* dates from the calendar icon or type it in. Otherwise, leave it to *Today*.

5a) Select a sample. To select multiple samples, hold on to **Ctrl** key and select the samples.

5b) For random selection, click **Choose Random** on the bottom-right corner. Type in number of samples to pull under the *Choose* column in the appropriate *Schedule* row.

6) Click **Send to WO**.

If I add a sample to a work order manually, will it affect the status of the same sample that's in the schedule?

No. If a sample has been added to a work order through the **Add** button instead of the **Send to WO** in the *Sample Schedule* screen, these samples are independent and don't mark the same sample that is scheduled as completed. The scheduled sample will remain in the schedule.

How to view/graph historical data.

1) Click on **Sampling** under the *Main Menu*.

2) Stay on the *Sample Detail* tab to view data in spreadsheet form. Click on the *Graph* to view data in a scatter graph.

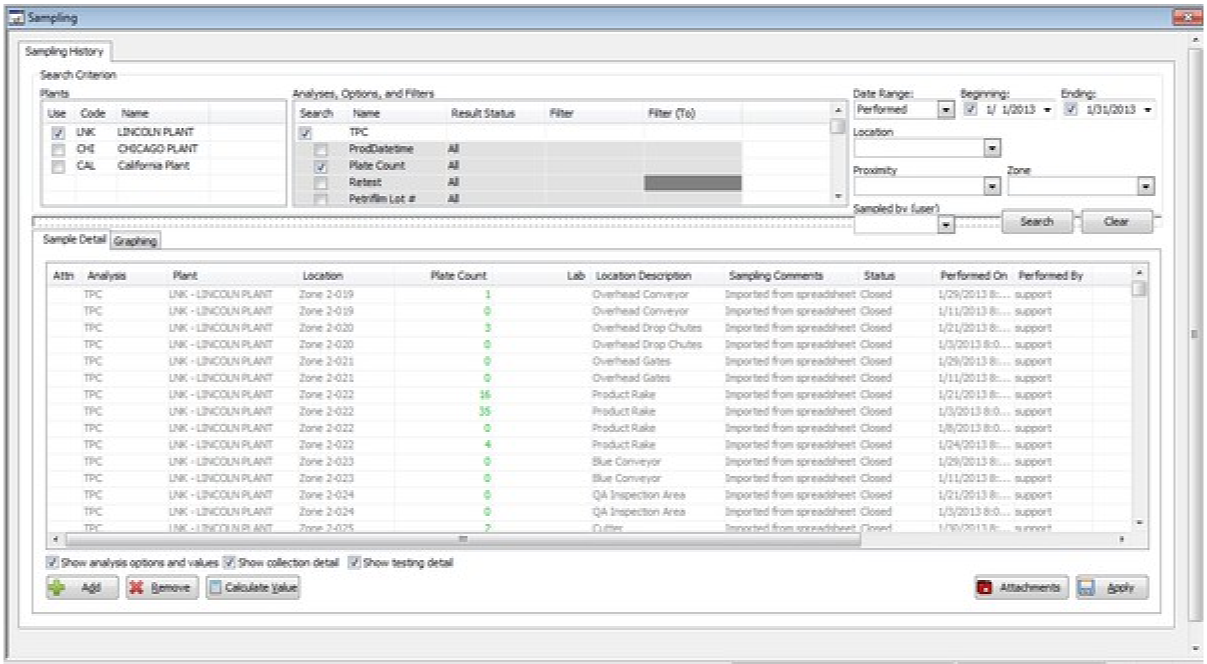
3) Select a plant.

4) Select an analysis name.

5) Check the boxes next to options you want pulled up. Enter filters when needed.

6) Plug in the variables in the boxes provided: date range, plants, location, user, proximity and/or zone.

7) Click **Search**.

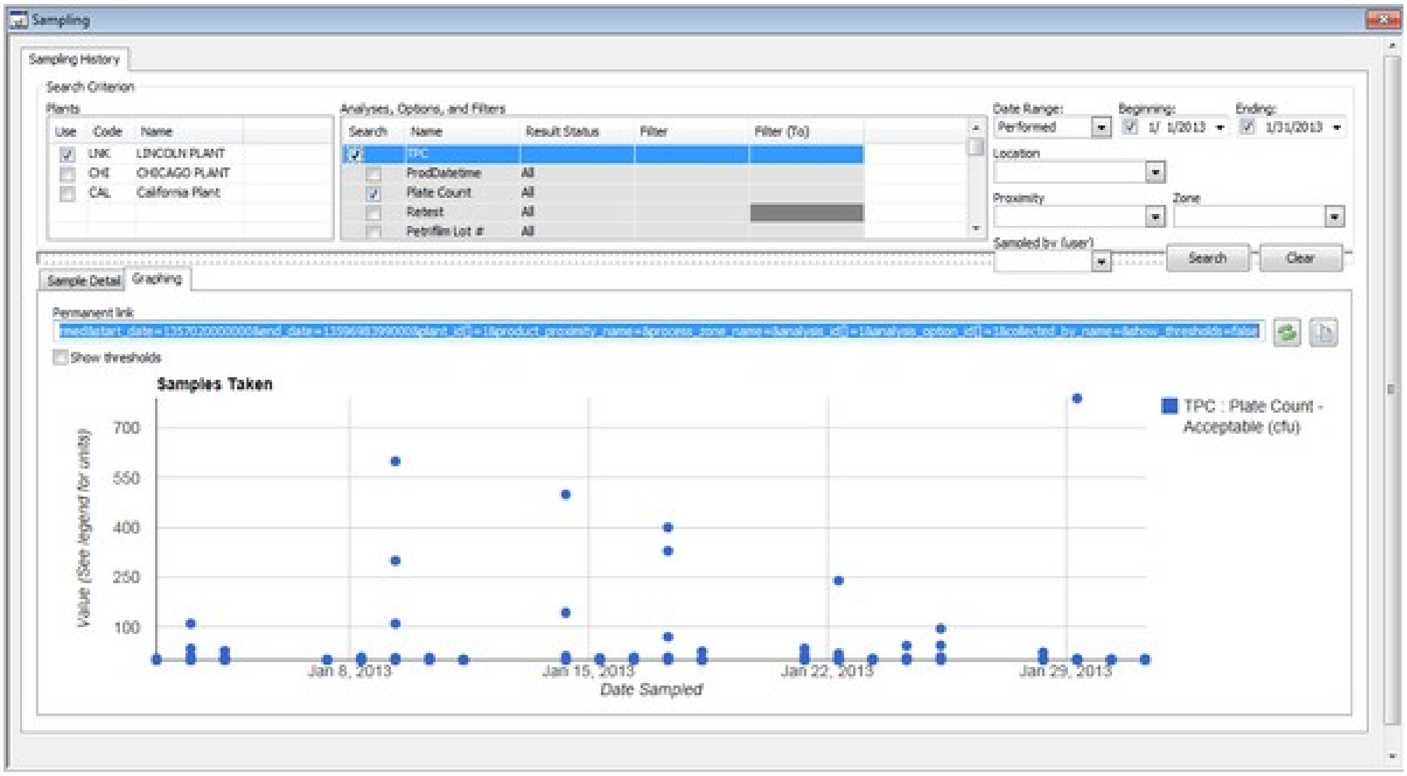


Click on the column title on the table to order the data by a certain criteria. For example, click on the Performed On to see results in chronological order of date performed.

Select and change criteria for the graph using the same panel used for sampling history on the upper portion of the screen.

Hover over points to view details such as date, time, analysis, analysis option, location, etc. This pop-up can be customized.

Hover over a label in the map key to highlight the group of points which belong to that label.



Why isn't an analysis option showing up on my Sampling Graph?

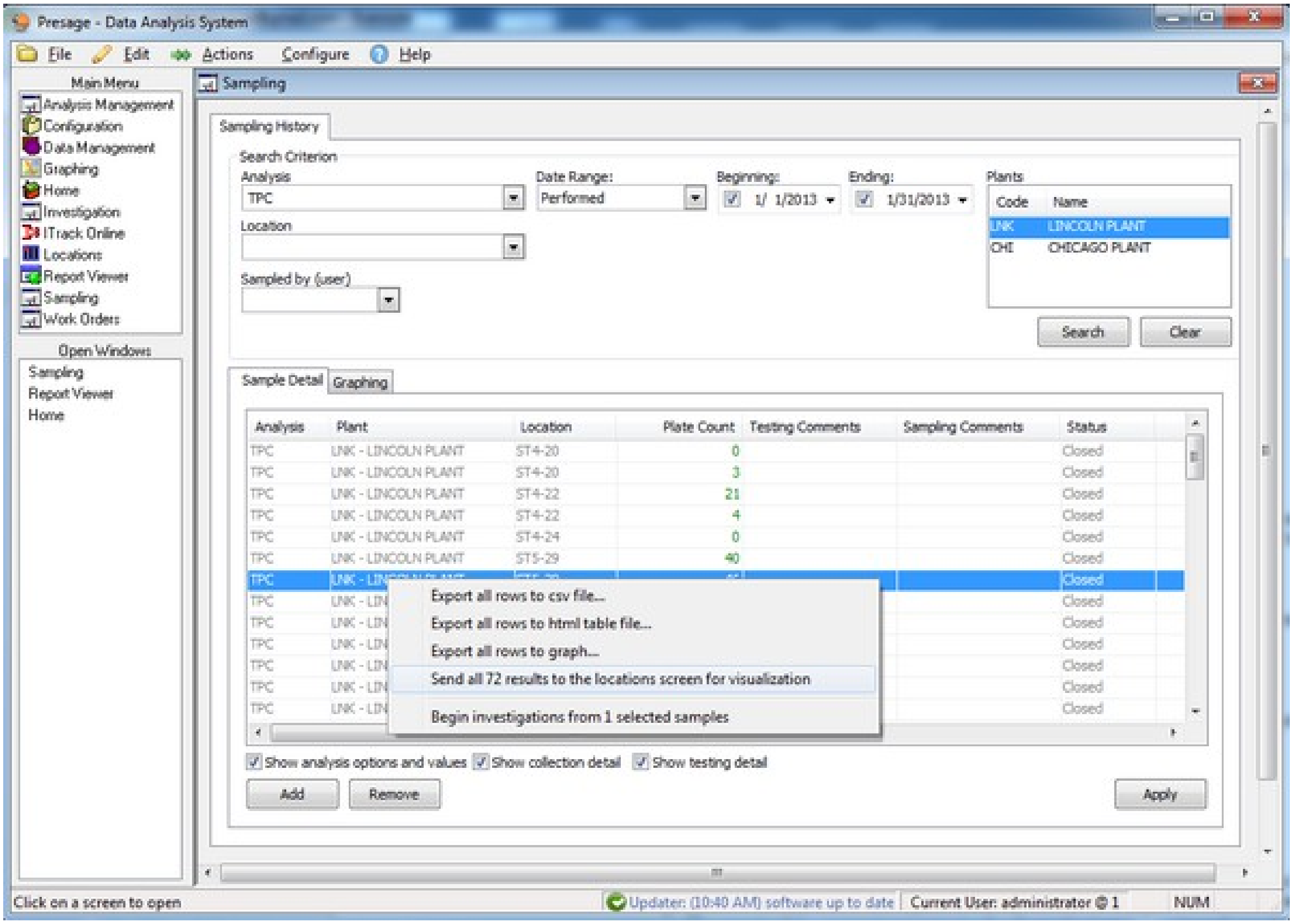
If an option has been checked as **Informational**, it will not appear in the graph.

How to view sample acceptability status over the plant map.

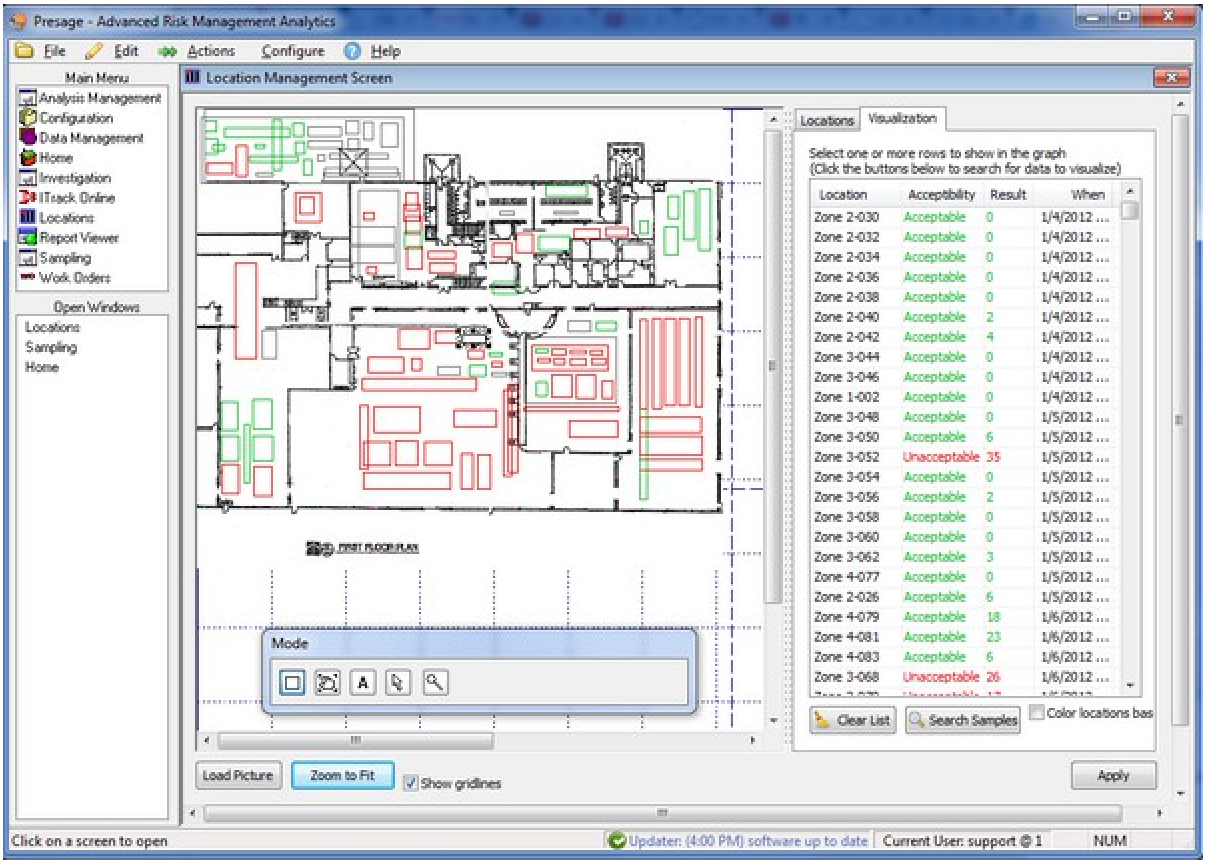
1) Click on **Sampling** under the *Main Menu*.

2) Enter search criteria and pull up historical data.

3) Right-click on one of the rows. A dialog with multiple options pops-up. Click **send all # results to the locations screen for visualization**.



A new screen opens up. This is the *Visualization* tab under *Locations* Screen.



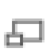
On the left display, locations are highlighted in green and red. Green locations are ones with acceptable counts, and red locations are ones with fail counts.

Highlight a row on the right and the corresponding location will be highlighted automatically on the left display and vice versa.

Order the *Date Sampled* column by clicking on the column title **When**. Click the first row and press the down-arrow on the keyboard to see a play-through of where the samples were taken in a chronological order.

Note: Every row can be ordered by clicking the column title.

Check the box **Color locations based on selected samples only**, and select a group of rows to see the colors only for those selected.



Part of data not being sent to plant map.

The number of samples transferred to the plant map will depend on the number of items listed in the first page of *Sampling History*. By default, this screen lists 1000 items. If there are more than 1000 samples, there will appear two arrows (one pointing to the left and one to the right) with the text **Next 1000 Samples** on the bottom of the screen.

To increase the limit on number of items to show in this list, click on **Configuration** under the *Main Menu*, then **Settings**.

Click **Sampling** under the *Category* list, and type in 10,000 (more or less) under the *Value* (second column).

Click **Apply**.

How to pull up a sample on the scanner without scanning a barcode.

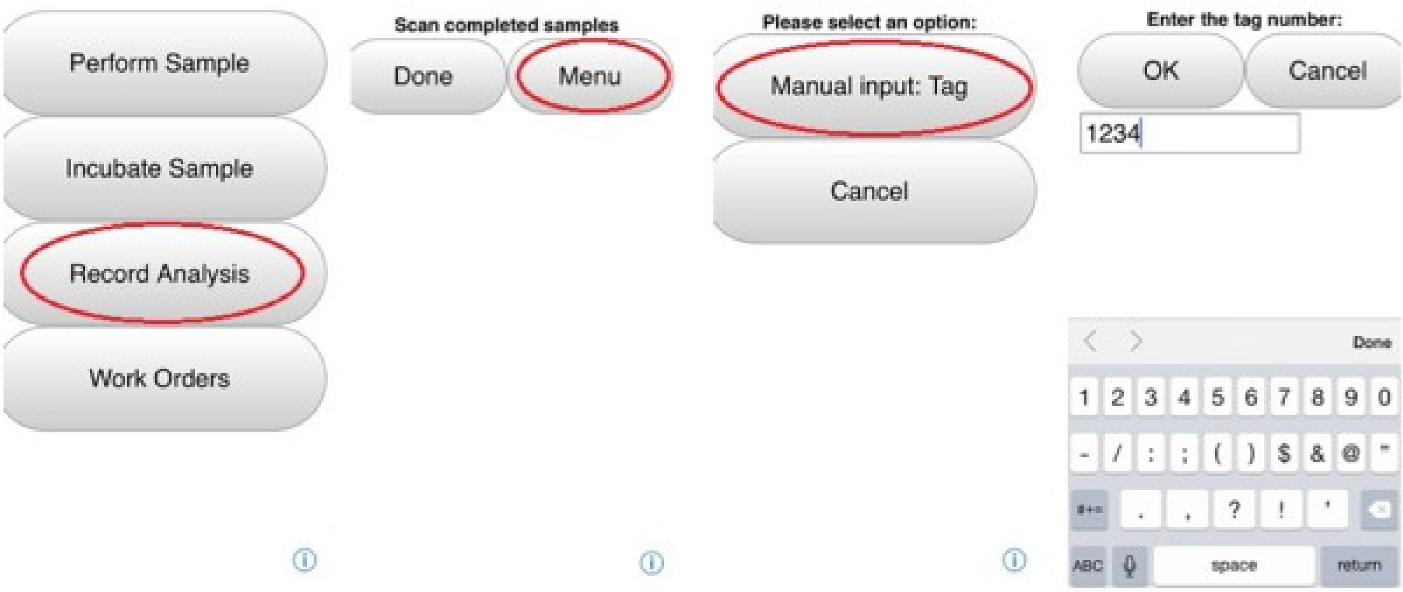
1) Press **Record Analysis**.

2) Press **Menu**.

3) Press **Manual Input: Tag**.

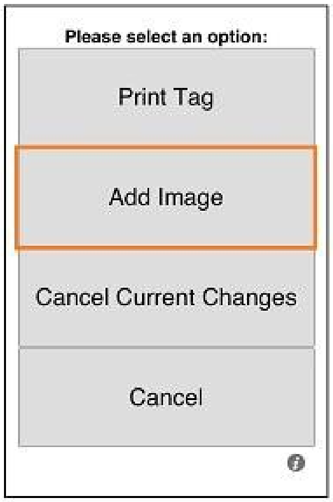
4) Type in the Tag# in the field provided.

5) Press **OK**.



How to add images to a sample on the scanner.

Open Sample
Click **MENU**



Click **ADD IMAGE**

Take a photo
Click **USE**
Or **RETAKE** to retake a photo

Scroll down to see the photo
Click **DONE**

How to create a work order on the scanner.

1) Press **Work Orders** in the main menu on the scanner.

2) Press **Menu**.

3) Press **New Work Order**.

4) Tap on the first empty box and select an analysis name.

5) Tap on the second empty box and select a location name.

6) Press **Done** on the scroll pop-up.

7) Press **+**. Wait for the newly added sample to appear on the top of the selection area.

8) Repeat steps 4, 5, 6, and 7 for each sample.

9) Press **Done**.

10) Repeat all the steps for each new work order.

How to add a sample to an existing work order.

1) Press **Work Orders** in the main menu on the scanner.

2) a. Scan work order barcode.

2) b. Press **Menu**, **Manual input: Work Order**, type in work order number.

3) Tap on the first empty box and select an analysis name.

4) Tap on the second empty box and select a location name.

5) Press **Done** on the scroll pop-up.

6) Press **+**. Wait for the newly added sample to appear on the top of the selection area.

7) Press **Done**.

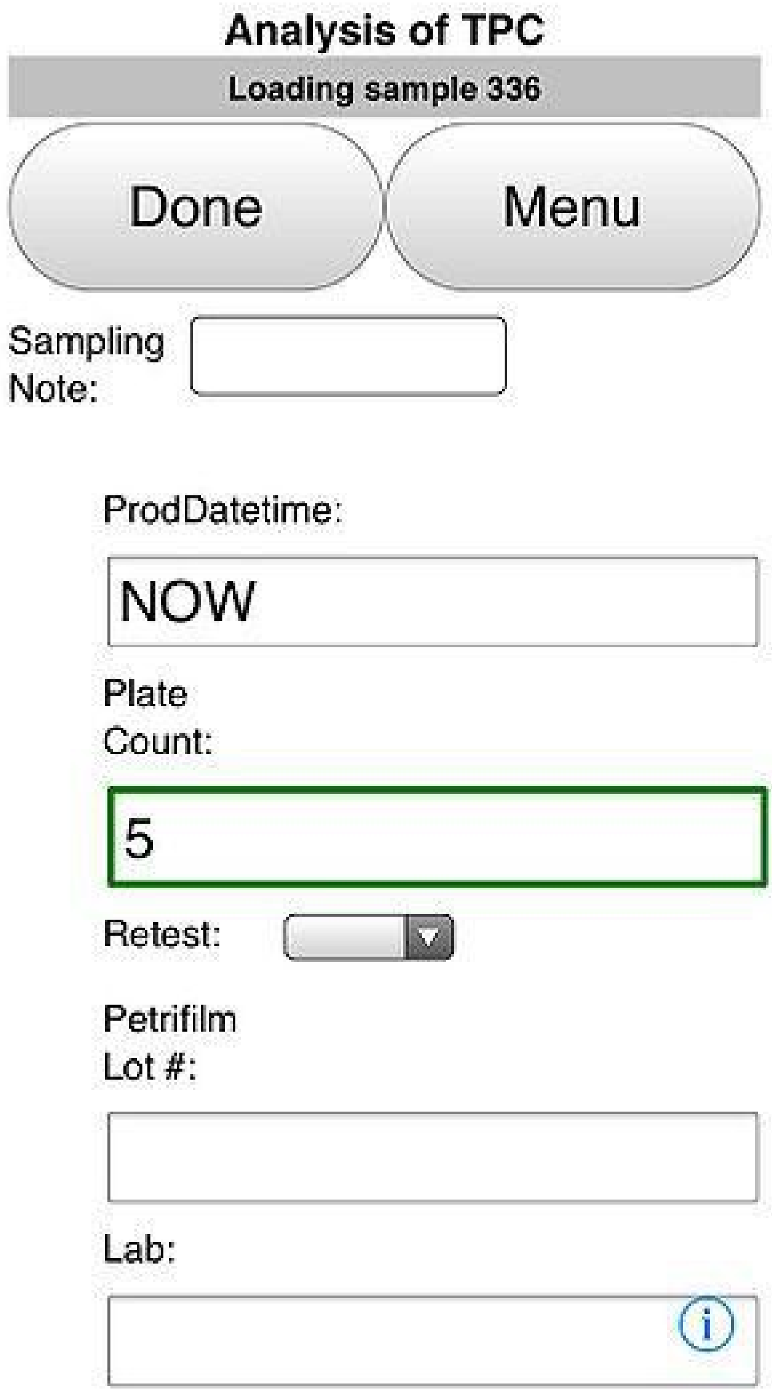
What do the result status colors mean on the scanner?

Black = Not Calculated or Error

Red = Unacceptable (out of spec)

Orange = Warning

Green = Acceptable (within spec)



How to change the date range(reduce clutter) in the work order queue.

There is a menu to the left of the display panel in the *Work Orders* tab.

User is the login user. Make sure to check this box instead of *All* to see work orders specific to you.

Date Range drop-down list shows time periods during which work orders were assigned. For example, selecting “Last Month” will list all work orders created in the previous month. There is an option to set the date range to *Custom*. For this option, users can set a date range by specifying the start and end date. To the right of the date box is a calendar icon for easier date selection.

Select the plant from the drop-down to narrow down the work order list to one plant.

By checking the *Show Closed* box, users can view all completed and pending work orders within the date range set.

The laser is not working on the Linea Pro.

Sometimes when the application is running for a long time, it needs to be restarted.

1) Double-click on the **Home** button on the iPod. The only button on the face of the iPod.

2) Swipe the ITrackLX app up to remove it from history.

3) Press on the *ITrack LX* app from the home screen and resume with logging in.

Most often the above steps will fix the laser, if not try the following:

Detach the iPod from the Linea Pro, then reattach.

Recently created work order is not showing in the *Home Screen* queue.

Click **Refresh** in the *Home Screen* to the left of the work order queue to see the new work order.

How to reorder columns in the work order screen.

There are two types of columns in the work order screen: constant columns and analysis option columns. The **Constant Columns** are those columns which are always the same in the work order screen: Attn, Analysis, Locations, Tag#, Sampling Comments, Testing Comments, Performed On, Performed By, etc. The **Analysis Option Columns** are those columns which are set up in the *Analysis Management* screen for each sample.

In the work order screen, click & drag the column header to where you like to place the column. The blue vertical line is an indicator of where the column will be dropped if you let go of the click.

Note: Analysis option columns are *always* placed after the **Locations** column. If you would like to move one of the Constant or non-analysis option columns from the very right of the work order screen to the beginning of the work order, make sure to place the columns before the **Locations** column to save the new order. If you place the column right after the **Locations** column, this order will not be saved as the analysis option columns will go in between the **Locations** column and the other column.

How to change ordering of analysis options in the work order screen.

There are two types of columns in the work order screen: constant columns and analysis option columns. The **Constant Columns** are those columns which are always the same in the work order screen: Attn, Analysis, Locations, Tag#, Sampling Comments, Testing Comments, Performed On, Performed By, etc. The **Analysis Option Columns** are those columns which are set up in the *Analysis Management* screen for each sample.

The analysis option ordering is managed in the **Analysis Management** screen under the *Main Menu*.

Click on the Analysis Name.

Click, drag, and drop the option name where you like it to go. The bold black line indicates where the option will be dropped.

How to change time or datetime fields to military format.

Presage uses the Windows User Locale settings to figure out what format to display/edit times and datetimes in. To do this, change the windows date/time display format, and Presage will follow suit.

1) Click **Start** or the Windows button on the lower-left.

2) Click **Control Panel**.

3) Click **Clock, Language, and Region**.

4) Click **Change the date, time, or number format** under *Region and Language*.

5) Set *Short time:* as **HH:mm** and the *Long time:* as **HH:mm:ss**.

6) Click **Apply**, then **OK**.

7) Now reopen Presage.

Part of data not exported to the .csv file.

The number of samples exported will depend on the number of items listed in the first page of *Sampling History*. By default, this screen lists 1000 items. If there are more than 1000 samples, there will appear two arrows (one pointing to the left and one to the right) with the text **Next 1000 Samples** on the bottom of the screen.

To increase the limit on number of items to show in this list, click on **Configuration** under the *Main Menu*, then **Settings**.

Click **Sampling** under the *Category* list, and type in 10,000 (more or less) under the *Value* (second column).

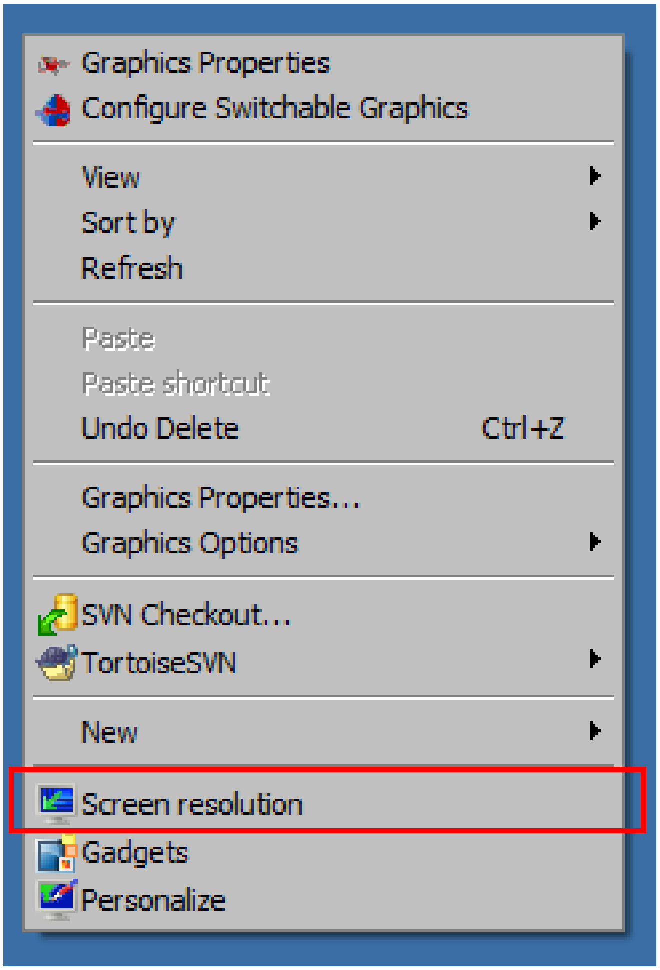
Click **Apply**.

How to Fix Overlapping in Presage Screens

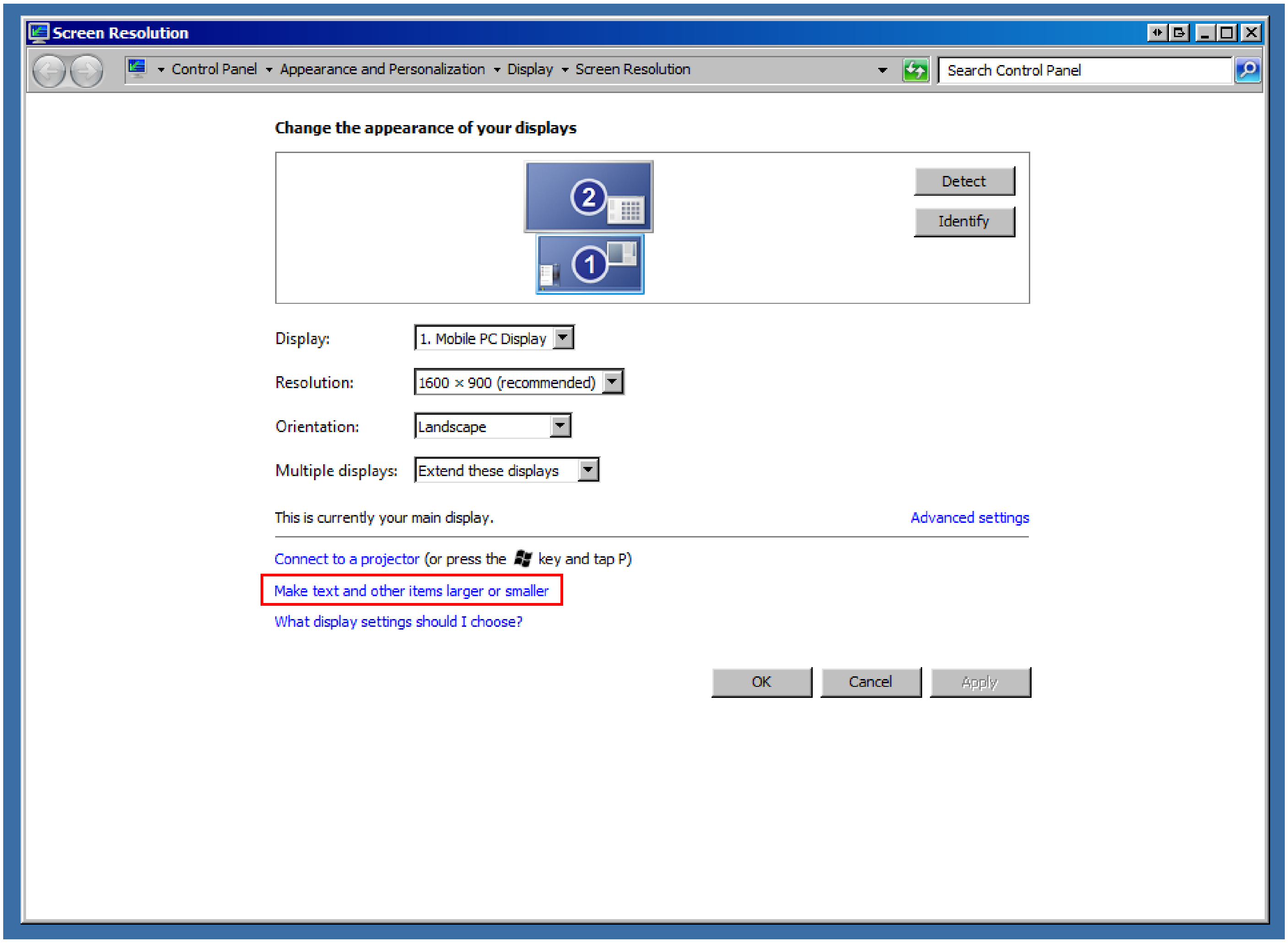
ITrack and Presage are formatted to have the default text size used, if a different text size is set on the computer then you will need to change the text size to 100%:

Changing DPI

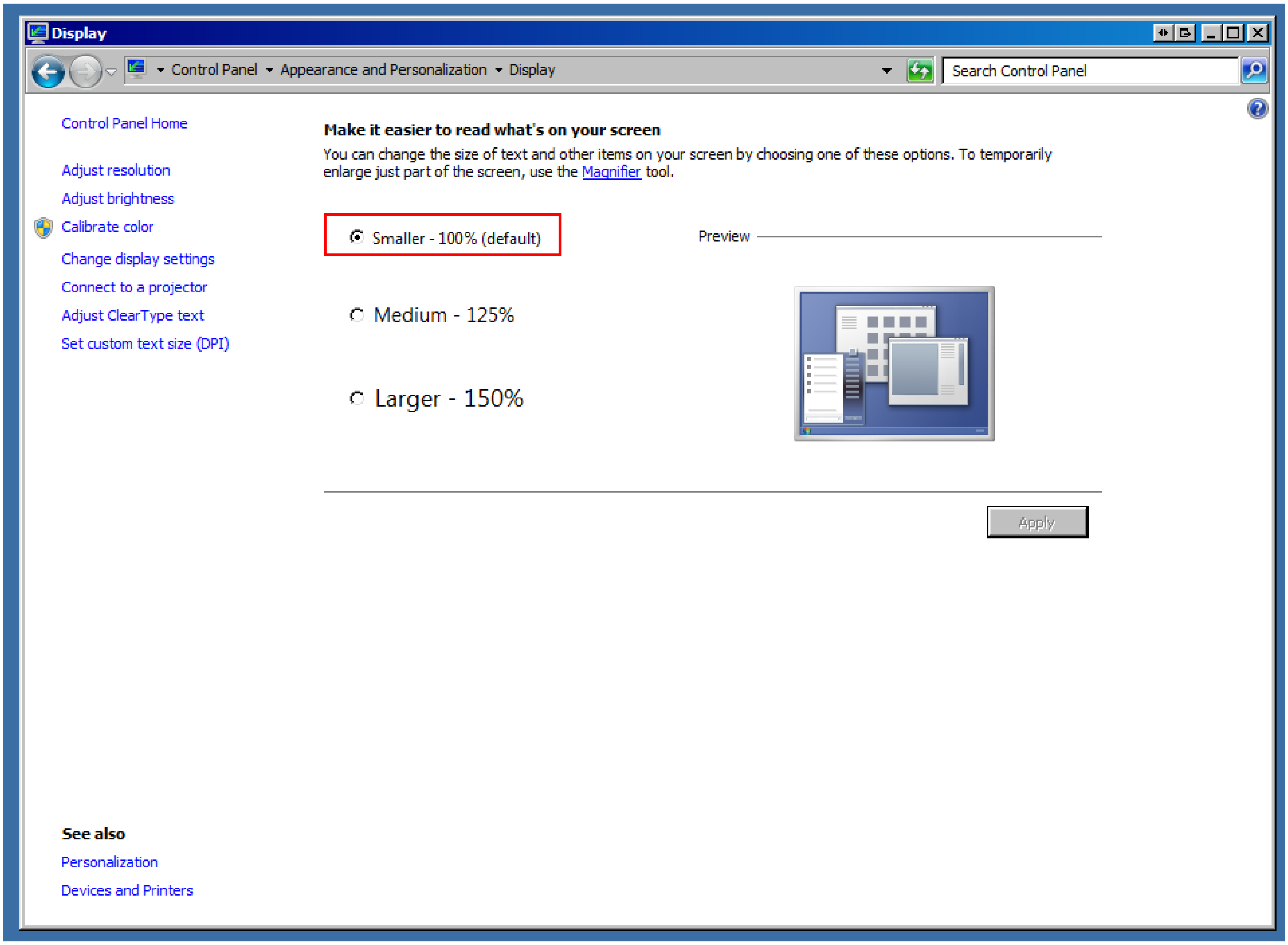
On your Windows Desktop right-click on an open area of the screen. A menu will appear, please select the 'Screen resolution' option.



This will cause a window to pop up. Click on the 'Make text and other items larger or smaller' and a new set of options will appear in the window.



Select the 'Smaller - 100% (default)' option and click the 'Apply' button.

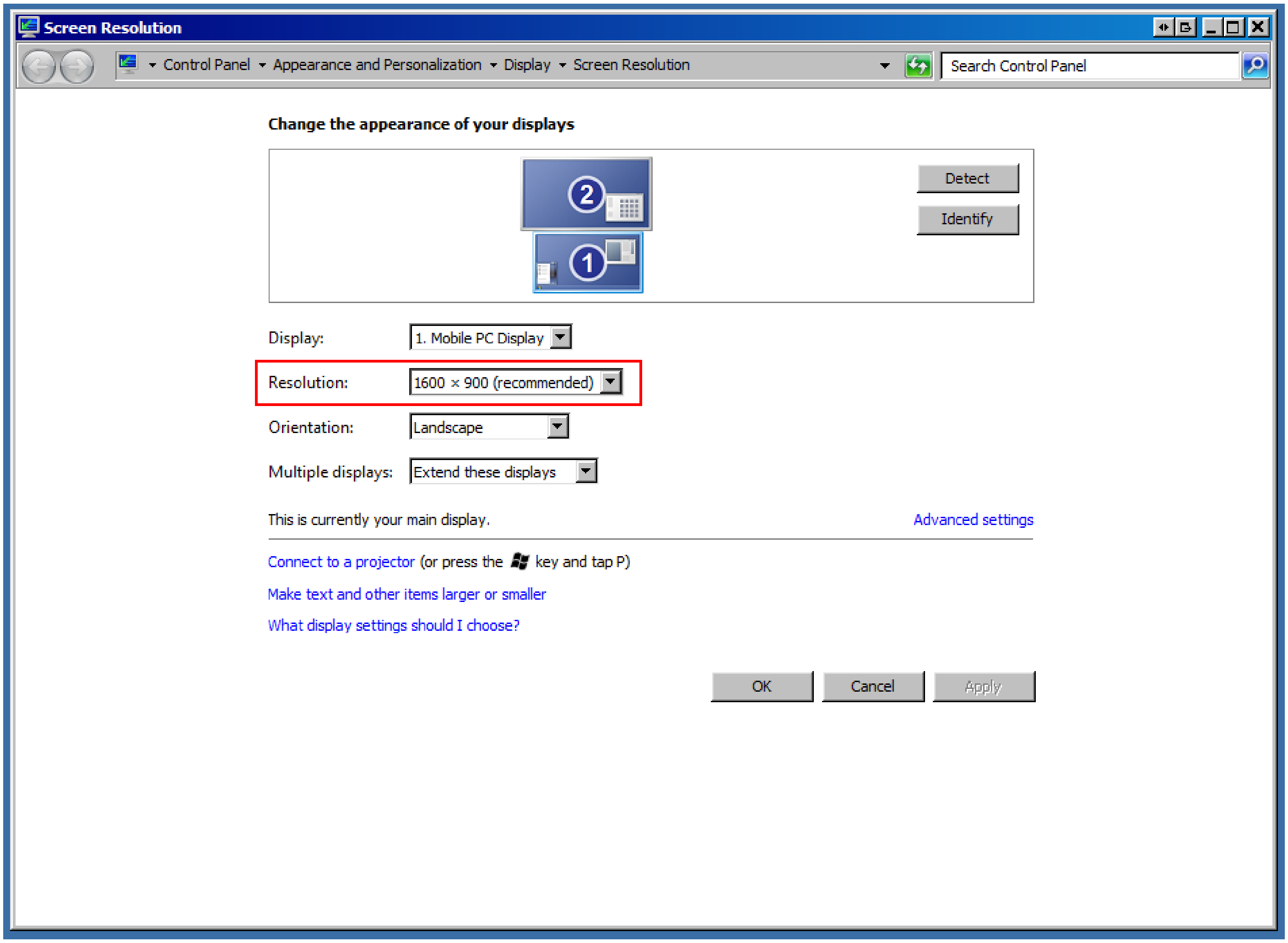


You will need to log out of your Windows User and log back in for the changes to take effect.

Changing Screen Resolution

After changing the DPI on a computer, the user may find the text and icons too small, in that case you will need to adjust the screen resolution, which is similar yet distinct from the DPI of the computer.

To adjust the screen resolution, right-click on an open area of the Desktop and you will see the menu appear again. Select 'Screen Resolution' as before, but once the window pops up click on the 'Resolution' drop down and adjust that until the user finds a suitable size for their icons and text.


Screen Resolution
Control Panel ▸ Appearance and Personalization ▸ Display ▸ Screen Resolution
Search Control Panel
Change the appearance of your displays
2
1
Detect
Identify
Display: 1. Mobile PC Display
Resolution: 1600 × 900 (recommended)
Orientation: Landscape
Multiple displays: Extend these displays
This is currently your main display.
Advanced settings
Connect to a projector (or press the key and tap P)
Make text and other items larger or smaller
What display settings should I choose?
OK
Cancel
Apply