CoreStar

EddyVISION®32 Tutorial
Release 5.0

September 2003

Covering: EddyAdmin, Acquisition, Analysis, MakeComp, & DBMS

Table of Contents:

1. Preface 6

2. Contact Information 7

3. Installation 8

4. Updating HASP Keys 9

5. Importing Pre v5.0 Files Using EddyAdmin 10

6. Creating a Project 12

7. Creating a Component File 17

8. Adding Landmarks to a Component File 27

9. Creating an Inspection Plan 32

10. Acquiring Data 43

11. Copying Cal Groups to Transfer Media 67

12. Copying Cal Groups from Transfer Media 68

13. Creating a History File 70

14. Analyzing Eddy Current Data (*excludes AutoVISION*) 72

15. Uploading Reports to the Database 108

16. Plotting Results on a Tubesheet Map...110

17. Printing Results..121

Appendix A: Understanding Pitch and Pitch Angles....................................125

Appendix B: More About Acquisition and Analysis 128

Appendix C: More about MakeComp 133

Appendix D: More about Auto-locate..134

Appendix E: Understanding Queries 135

Appendix F: Project Tips..139

List of Figures:

Figure 6-1. Project Table - Sites ... 13
Figure 6-2. Completed EddyAdmin Dialog ... 15
Figure 7-1. Tutorial Component (FWH3A) ... 17
Figure 7-2. Parameters Dialog - Parameters Tab ... 19
Figure 7-3. Parameters Dialog - Tube Labels Tab ... 19
Figure 7-4. Completed Group Parameters Dialog for Section 1 ... 22
Figure 7-5. Adding Group Dialog - Section 1 ... 23
Figure 7-6. Auto Tick Param Dialog ... 24
Figure 7-7. Completed Group Parameters Dialog for Section 2 ... 26
Figure 8-1. Parameters dialog - Landmarks Tab Showing Landmark Set 0 ... 30
Figure 8-2. Parameters dialog - Landmarks Tab Showing Landmark Set 1 ... 31
Figure 9-1. Periphery Plan in DBMS Screen ... 33
Figure 9-2. Using the List Dialog Manually ... 35
Figure 9-3. Repair Attributes Dialog ... 36
Figure 9-4. Bounding Existing Plugged Tubes Using the Box Dialog ... 38
Figure 9-5. Pattern Fill Dialog ... 39
Figure 9-6. Using the Pattern Fill Dialog on a Selected Section ... 40
Figure 9-7. Plan Dialog with Sample Plan Loaded ... 42
Figure 10-1. Current Operator Type Display Box ... 44
Figure 10-2. Plan Dialog with Plan Loaded in Acquisition ... 45
Figure 10-3. Test Options Dialog ... 47
Figure 10-4. OMNI Test Config Dialog ... 49
Figure 10-5. Ethernet Pusher Config ... 50
Figure 10-6. Summary Dialog ... 53
Figure 10-7. Acquisition Screen - Differential Channel Setup ... 55
Figure 10-8. Acquisition Screen - Absolute Channel Setup ... 57
Figure 10-9. Acquisition Screen - Setting Differential Span Levels ... 58
Figure 10-10. Acquisition Screen - Setting Absolute Span Levels ... 59
Figure 10-11. Setup Dialog - Storing the Cal Setup ... 60
Figure 10-12. Plan List at Bottom of Test Panel ... 62
Figure 10-13. Plan List and Updated Tube Encode for Next Tube (Split Screen Capture) ... 63
Figure 10-14. Fly Buttons Used for Quick Messages ... 64
Figure 11-1. Cal Admin Dialog - Archiving/Copying a Cal Group ... 67
Figure 11-2. Cal Admin Dialog - After Archiving/Copying a Cal Group ... 68
Figure 12-1. Cal Admin Dialog - Fetching a Cal Group ... 69
Figure 12-2. Cal Admin Dialog - After Fetching a Cal Group ... 69
Figure 13-1. History Dialog - Creating a History File in DBMS ... 71
Figure 14-1. Operator Type Display Box ... 73
Figure 14-2. Options Dialog - Report Tab ... 74
Figure 14-3. Options Dialog - Display Tab ... 75
Figure 14-4. Options Dialog - Print Tab ... 75
Figure 14-5. Analysis Screen - Differential Channels - Rotation Setup ... 77
Figure 14-6. Analysis Screen - Absolute Channels - Rotation Setup ... 78
Figure 14-7. Tube Suppport Signal Prior to Mixing ... 79
Figure 14-8. Setup Dialog - Mix Tab After Mixing Out TSP ... 81
Figure 14-9. Setup Dialog - Mix Tab After Mixing Out Noise ... 82
Figure 14-10. Analysis Screen - Setting Relative Voltage ... 83
Figure 14-11. Analysis Screen - Curve Dialog - Entering the Phase of a Calibration Flaw ... 86
Figure 14-12. Analysis Screen - Curve Dialog - Entering the As-built Depth of a Calibration Flaw ... 87
Figure 14-13. Analysis Screen - Curve Dialog – Completing a Phase Curve ... 89
Figure 14-14. Analysis Screen - Curve Dialog - Entering the As-Built Depth and Volts Measurement of a Calibration Flaw ... 92
Figure 14-15. Analysis Screen - Curve Dialog – Completing a Volts Curve ... 93

Figure 14-16. Options - Report Tab - Check NDD Code 94
Figure 14-17. Report Dialog - Using an Embedded Header 95
Figure 14-18. Auto-locate - Identifying the Tube End 97
Figure 14-19. Auto-locate - Training the Tube End 98
Figure 14-20. Auto-locate - Training the Tubesheet Interface 99
Figure 14-21. Auto-locate - Training Tube Support Structures 100
Figure 14-22. Auto-locate - Training More Tube Support Structures 101
Figure 14-23. Analysis Screen - Making Calls Using Macro Buttons 103
Figure 14-24. History Dialog 104
Figure 14-25. Selecting a Defect Code From the Popup List & Making a Call 105
Figure 14-26. Modifying Report Entries Using the Report Update Dialog 106
Figure 14-27. Globally Editing Text Values in a Report Using the Global Edit Dialog 107
Figure 15-1. Import Report Dialog 109
Figure 16-1. Executing a Query in the Query Dialog 111
Figure 16-2. Results of a Query Plotted on a Tubesheet Map 112
Figure 16-3. Choose Symbol Dialog 112
Figure 16-4. Symbol Color Dialog 113
Figure 16-5. Changing the Description for an Entry in the Legend 113
Figure 16-6. Executing a Query That Returns Specific Inspection Results 114
Figure 16-7. Moving and Rearranging Legend Entries 115
Figure 16-8. Map Header Dialog 117
Figure 16-9. DBMS Options Dialog - Options Tab 117
Figure 16-10. DBMS Options Dialog -Margins Tab 118
Figure 16-11. Report Dialog Displayed By Double-clicking a Tube on a Tubesheet Map 120
Figure 17-1. DBMS Print Options Dialog - Headers Tab 122
Figure 17-2. DBMS Print Options Dialog - Footers Tab 123
Figure 17-3. DBMS Print Options Dialog - Options Tab 123
Figure A-1. Equilateral Tube Pitch Designs 125
Figure A-2. Non-equilateral Tube Pitch Design 126
Figure B-1. Manual Landmarks 130

1. Preface

The minimum requirements to run EddyVISION 32 release 5.0 are:

- Windows NT 4.0 – SP 6a, Windows 2000, or Windows XP – SP 1. Windows 95/98 and Windows ME are not supported.
- A three button mouse. Certain features are not available to users with a two button mouse.
- 64 MB of RAM.
- 1024 x 768 pixel display. (A screen resolution of 1280 x 1024 is recommended for analysis).
- 200 MHz Pentium-type processor.
- 100MB free hard disk space.

In this manual, the word ***Click*** refers to a left mouse click. Other mouse clicks will be explicitly stated. Tube-sheet maps may print out very slowly under certain printers if the ***Vector Graphics*** option is active. Enable raster graphics to speed up map printing.

2. Contact Information

CoreStar's Corporate office is located at:

CoreStar International Corp
1044 Sandy Hill Road
Irwin, PA 15642
Phone: (724) 744-4094
FAX: (724) 744-4093

Questions or comments on this document should be addressed to:

Dan Spake
Southeast Regional Office Division Manager
CoreStar International Corporation
PO Box 655
Huntersville, NC 28070-0655

Email: spake@corestar-corp.com
Tel: (704) 895-4680
Cell: (704) 453-8352
Fax: (704) 895-4681

CoreStar has a 24-hour support number (888) 533-9403.

3. Installation

The user can remove EddyVision 32 from their system via the standard ***Add/Remove*** programs feature of the ***Control Panel***.

To install the software:

`STEP 1:` Insert the program CD-ROM into CD-ROM drive. The install software should automatically start.

`STEP 2:` Follow the instructions during the setup process. Unlike previous versions of EddyVision, the target directory for rev 5.0 defaults to the standard windows program files directory (typically ***`C:\Program Files`***). Users can select another directory if they prefer.

`STEP 3:` Reboot the computer. After setup is complete, some operating systems may or may not ask that you restart the computer; however, it's strongly recommended that you do. Shortcuts for ***EddyAdmin***, ***CoreStar*** (Analysis & Acquisition), ***DBMS***, and the **MakeComp** software will appear on the desktop.

4. Updating HASP Keys

In v5.0, the ***magic_cookie*** file has been eliminated. The necessary enable codes are stored in the physical ***HASP*** key itself. So once the key has been properly updated, it will work on any system with v5.0 without the need to maintain the proper cookie file on all systems. The physical ***HASP*** keys themselves have not changed. After updating, each key will still work with pre v5.0 software versions provided an updated ***magic_cookie*** file resides in the ***\bin*** directory of the pre v5.0 install.

To update a HASP key to work with v5.0:

STEP 1: In ***EddyAdmin***, select **Util | Show Keys**. The ***HASP Keys*** dialog displayed shows the list of ***HASP*** keys that are currently attached to any parallel port of the computer. A module is enabled if it is toggled green. The OMNI key is special and applies only to CoreStar portable tester units. There is no physical ***HASP*** key associated with these.

STEP 2: To update a key, click the ***UPDATE KEY*** button located to the right of the key of interest, i.e., Analysis, DBMS, etc.

STEP 3: A dialog appears with 8 fields each containing room for 4 hexadecimal digits (i.e. digits 0 to 9 and letters A to F).

The simplest way to enter the codes is to load them from a key file. The one installed from the CD is stored in:

c:\Program Files\CoreStar\EddyVision 5.0\HASP\eddyvision.key.

To load the codes from the key file:

STEP 1: Click the ***LOAD FILE*** button.

STEP 2: Select the ***eddyvision.key*** file. If the current key is in the file, the update codes will be filled in automatically.

NOTE:	If the enable codes for your key(s) are not in the ***.key*** file, contact CoreStar and we can either FAX or e-mail you the proper enable codes, which you can then type in.

5. Importing Pre v5.0 Files Using EddyAdmin

These functions are used to import pre v5.0 files into the v5.0 install directory. Furthermore, these functions are mainly used when v5.0 is first installed and setup. Later, when tables, auto-rules, defects lists, or report macros are 'imported' into currently opened projects, these will be the initial default files available for import. This tutorial assumes the pre 5.0 install directory is **`c:\corestar`**.

To import pre v5.0 lookup tables (***EddyAdmin*** tables):

STEP 1: Start ***EddyAdmin*** by double-clicking the ***EddyAdmin*** icon on the desktop.

STEP 2: Select ***Util | Import Pre 5.0 | Lookup Tables***.

STEP 3: Open the pre v5.0 install directory, i.e., ***`c:\corestar`***, and click ***OK***.

The pre v5.0 ***EddyAdmin*** tables (***`.tbl`***) are copied into:

`c:\Program Files\CoreStar\EddyVision 5.0\user\tables\`.

These are now referred to as the ***Master Lookup Tables***.

To import pre v5.0 ***auto-rules*** for use with ***AutoVISION***:

STEP 4: Select ***Util | Import Pre 5.0 | Rules***.

STEP 5: Open the pre v5.0 install directory, i.e., ***`c:\corestar`***, and click ***OK***.

The pre v5.0 ***rule files*** (.***`rul`***) are copied into:

`c:\Program Files\EddyVision 5.0\user\auto_rules\.`

NOTE: The import utility looks for the following directory path:

`C:\corestar\config\rule_base\`

To import pre v5.0 ***defect lists***:

STEP 6: Select ***Util | Import Pre 5.0 | Defect Lists***.

STEP 7: Open the pre v5.0 install directory, i.e., ***c:\corestar***, and click ***OK***.

The pre v5.0 ***defect_lists*** files (.***def***) are copied into:

c:\Program Files\EddyVision 5.0\user\defect_lists\.

> **NOTE:** The previous import utility looks for the following directory path:
>
> ***C:\corestar\config\defect_lists\.***

To import pre v5.0 ***report macros***:

STEP 8: Select ***Util | Import Pre 5.0 | Report Macros***.

STEP 9: Open the pre v5.0 install directory, i.e., ***c:\corestar***, and click ***OK***.

The pre v5.0 ***report macro*** files (.***rep***) are copied into:

c:\Program Files\EddyVision 5.0\user\report_macros\.

> **NOTE:** The previous import utility looks for the following directory path:
>
> ***C:\corestar\config\report_macros\.***

6. Creating a Project

In EddyVision v5.0, a project is simply a directory, which contains all of the information associated with a given component. There are several ways to create a project. This procedure will show you how to create a new project from 'scratch'.

STEP 1: Create a directory on your system named ***C:\Projects*** using the ***Windows File Explorer*** or ***Manager***.

STEP 2: Start ***EddyAdmin*** by double-clicking the ***EddyAdmin*** icon on the desktop.

STEP 3: Select ***Project* | *New*** from the menu bar.

STEP 4: In the ***New Project*** dialog, browse to ***C:\Projects***.

STEP 5: In the ***Project Name*** box, type in the name of the project and click ***OK***. This should be the name of the component to be inspected. In this example, we'll be inspecting ***Feedwater Heater-3A***. Let's use ***FWH3A*** as the name.

> **NOTE:** ***EddyAdmin*** will create the following directory for the project:
>
> ***C:\Projects\FWH3A***
>
> A log file will be displayed that will show exactly which files were created or copied. For example, the ***Master Lookup*** tables stored in:
>
> ***c:\Program Files\CoreStar\EddyVision 5.0\user\tables***
>
> were copied to the new project.

STEP 6: Fill in the attributes of the new project. Click in the ***Site Code*** box to display the available site code.

STEP 7: Click on the desired site code. If the site code you desire is not present, you may either edit the ***Components*** (site) table, or import the ***Master Components*** table containing the desired site codes and components.

For the tutorial, we recommend you edit the projects ***Components*** table. You can skip this procedure and import the ***Master Component*** table if desired. To edit the ***Components*** table:

STEP 1: Click ***Edit | Project Lookup Tables***.

STEP 2: Click the ***Components*** tab.

STEP 3: Click the **** (asterisk)*** above the vertical scroll bar. A new row appears at the bottom of the ***Components*** table.

STEP 4: Click in each field of the new row and type in the following information:

SITE CODE = CIC, COMPONENT = FWH3A, OWNER = ABC POWER GROUP, TZ (Timezone) = -4, UNIT = 2, then click ***OK***.

The entry should appear as shown in Figure 6-1.

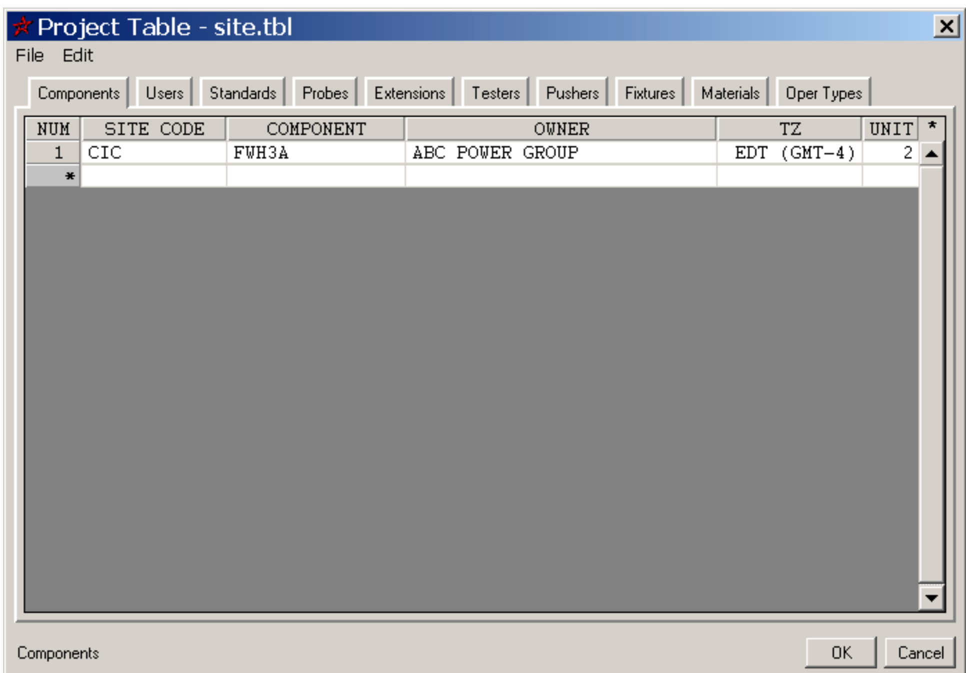


Figure 6-1. Project Table - Sites

STEP 5: Skip to step 6 in the next sequence.

To import the ***Master Components*** table:

STEP 1: Click ***Edit | Project Lookup Tables***.

STEP 2: Click the ***Components*** tab.

STEP 3: Select ***File | Import Master Components Table***.

STEP 4: Click ***Yes*** on the message box.

STEP 5: Click ***Cancel*** to undo or ***OK*** to accept the import of the ***Master Components Table***.

NOTE: The previous 5 steps imports only the ***Master Components*** table from:

c:\Program Files\CoreStar\EddyVision 5.0\user\tables\site.tbl

into the currently opened project.

STEP 6: Select ***CIC*** as the ***Site Code*** on the main ***EddyAdmin*** dialog.

STEP 7: Ensure that the ***Data Format*** for ***DSET 1*** is set to ***CoreStar***.

STEP 8: Ensure that the ***Data Directory*** for ***DSET 1*** is set to ***ecdata\\.***

STEP 9: In the ***Outage*** field, type ***2RF05*** as the outage name.

HINT: In the outage name ***2RF05***, '05' is 'zero five'. Also, '2' is for the unit, 'RF' means 'refuel', and '05' is the 5th outage since startup. This is an example of one of the many possible naming conventions.

STEP 10: The ***Date*** field can contain the first day of the outage, today's date, or an earlier date. Let's enter today's date in the format: MM/DD/YYYY.

STEP 11: Click the ***DBMS*** list box and choose the version of MS Access that will be used to manage any databases created with this project. For the purpose of this tutorial, select ***Access 97***.

NOTE: EddyVISION 32 can work with any of the three Access database formats shown in the ***DBMS*** list box. The choice made will determine the Access format used when a new database is created in this project.

STEP 12: In the ***Tester*** list box, choose ***OMNI-100***.

STEP 13: In the ***Pusher*** list box, choose ***Track Drive***.

STEP 14: Click the ***History*** button to enable the use of a history file. The button turns green.

STEP 15: Set the ***Table Rows*** to ***5***.

STEP 16: Set the ***Table Fields*** to ***11***.

STEP 17: Click the ***Show Oper Type*** button to enable it. The button turns green.

STEP 18: Click the ***Require Logon*** button to enable it. The button turns green.

STEP 19: ***Optional***: In the ***Project Notes*** area at the bottom of the ***EddyAdmin*** dialog, you may enter a description for the project. The final ***EddyAdmin*** dialog should appear as shown in Figure 6-4.

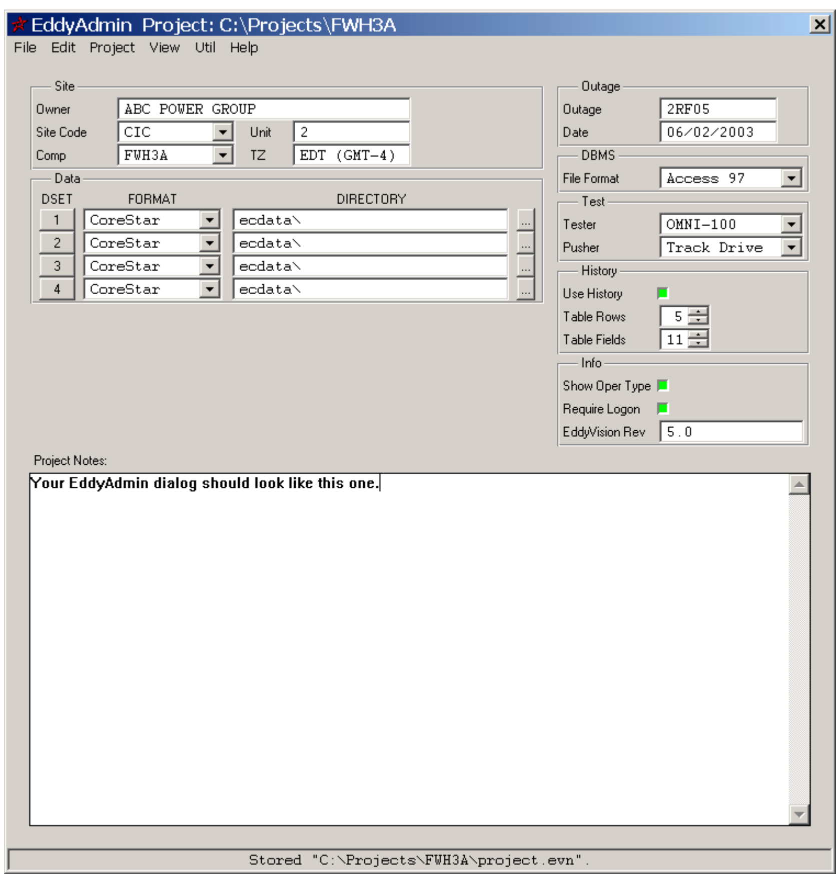


Figure 6-4. Completed EddyAdmin Dialog

The following Steps demonstrate how to import existing database, map, and component files into the current project; however, please skip to step 7. We will be creating new versions of these files for tutorial purposes.

STEP 1: Click ***Project* | *Import Databases***.

STEP 2: Select the desired ***CoreStar*** database file(s) (***.mdb***) from the source directory and click ***OK***. An import log screen appears. The name of the copied database(s) is not altered in any way.

STEP 3: Click ***Project* | *Maps***.

STEP 4: Select the desired ***CoreStar*** map file(s) (***.map***) from the source directory and click ***OK***. An import log screen appears. The name of the copied map file(s) is not altered in any way.

STEP 5: Click ***Project* | *Import Component***.

STEP 6: Select the desired ***CoreStar*** component file (***.cmp***) from the source directory and click ***OK***. An import log screen appears. The name of the copied component file is changed to ***project.cmp***.

STEP 7: Exit ***EddyAdmin***.

STEP 8: You will be asked if you wish to save changes. Choose ***Yes***.

TIP:	You may return to ***EddyAdmin*** anytime to make desired edits to a given project.

7. Creating a Component File

The component file is one of the most important parts of good inspection planning. The author cannot express its significance enough. Take the time to create a component file that meets the needs of the inspection – especially in terms of tube identification.

For this tutorial, we will create a U-tubed feedwater heater showing both halves (legs) of the tube bundle. The intent is to inspect only the straight-lengths on each leg and we will setup the component file accordingly. Figure 7-1 shows the component that we'll build during this tutorial.

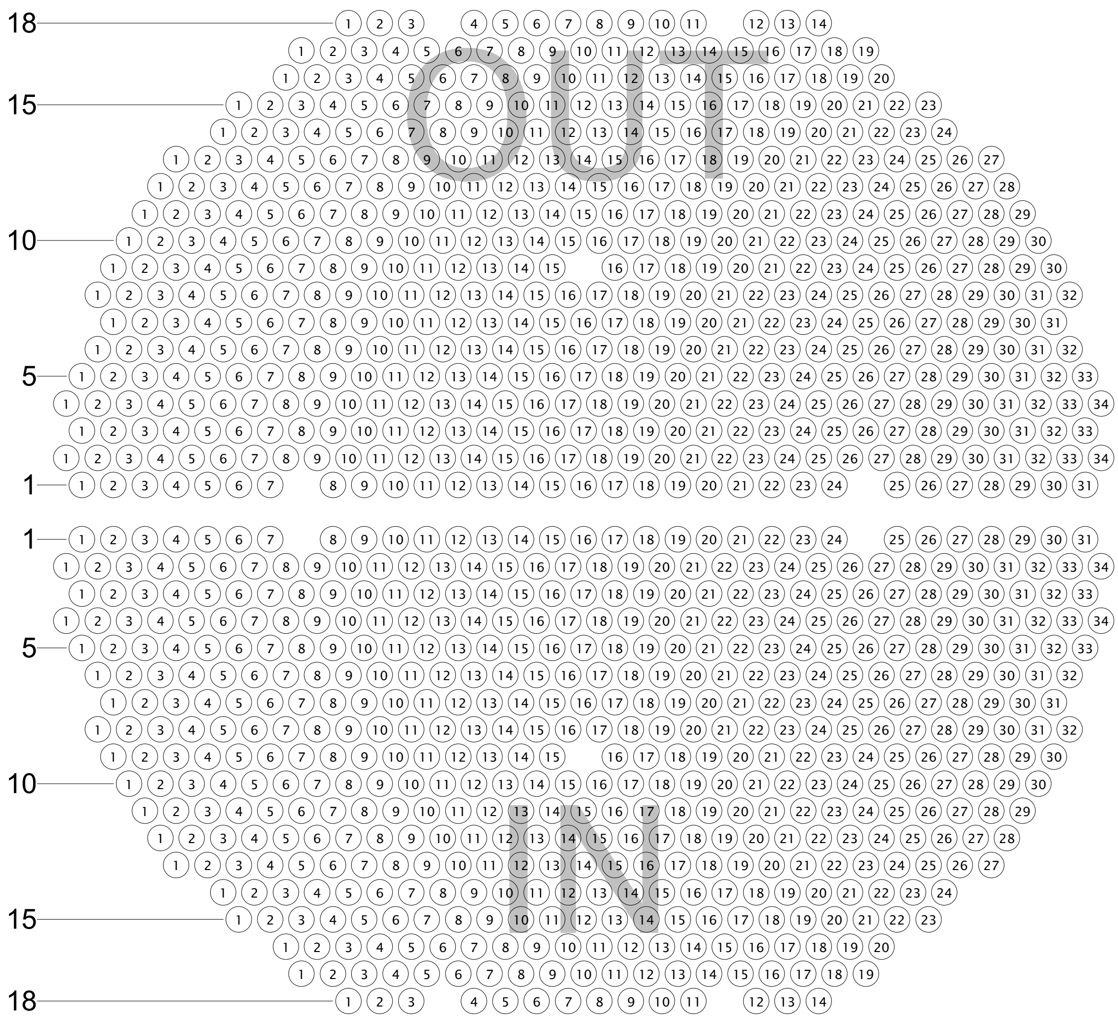


Figure 7-1. Tutorial Component (FWH3A)

To build the component shown in Figure 7-1:

STEP 1: Double-click the ***MakeComp*** icon on the desktop. The ***FWH3A*** project should open. If not, select ***File | Open Project***, select ***c:\Projects\FWH3A***, and click ***Open***.

STEP 2: Select ***File | New Component***.

STEP 3: Click the ***Parameters*** button on the main screen.

STEP 4: Click the ***Parameters*** tab on the ***Parameters*** dialog.

STEP 5: In the ***Component Type*** field, type ***Feedwater Heater***.

STEP 6: In the ***Model Name*** field, type ***FWH3***.

TIP:	This component can be used for say feedwater heaters 3A, 3B, and 3C; therefore, it only has to be built once and shared between the other projects.

STEP 7: In the ***Row Name*** field, type ***ROW***.

STEP 8: In the ***Col Name*** field, type ***TUBE***.

STEP 9: In the ***Sec Name*** field, type ***PASS***.

STEP 10: In the ***Inlet Name*** field, type ***INLET***.

STEP 11: In the ***Outlet Name*** field, type ***OUTLET***.

STEP 12: Enable ***Draw Sec Labels*** and ***Use Tube Labels*** by clicking the button beside each. The button turns green.

STEP 13: If necessary, disable ***Has Ubends*** by clicking the button beside it. The button turns gray (not green) and appears raised.

NOTE:	***Has Ubends*** is turned on **only** when building a U-tubed component for a project where the U-bends will be inspected **and** only half the component will be shown on the map. This feature calculates the length of each U-bend for each tube.

The ***Parameters*** tab of the ***Parameters*** dialog should now appear as shown in Figure 7-4.

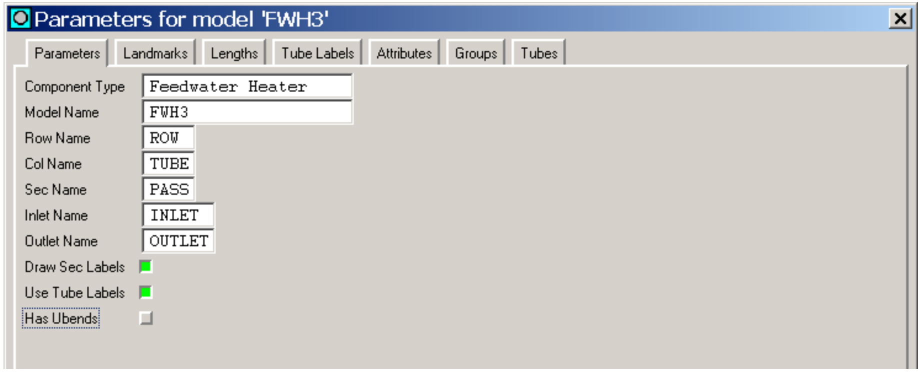


Figure 7-4. Parameters Dialog - Parameters Tab

STEP 14: Click the ***Tube Labels*** tab.

STEP 15: Click the **** (asterisk)*** at the top of the scroll bar for the ***Sec Labels*** 3 times. This creates 3 blank rows beginning with ***0*** (***zero***).

STEP 16: Click in the ***Sec Label*** field for ***Sec 1*** and type ***IN***.

STEP 17: Click in the ***Sec Label*** field for ***Sec 2*** and type ***OUT***. The ***Sec Label*** fields on the ***Tube Labels*** tab should now appear as shown in Figure 7-7.

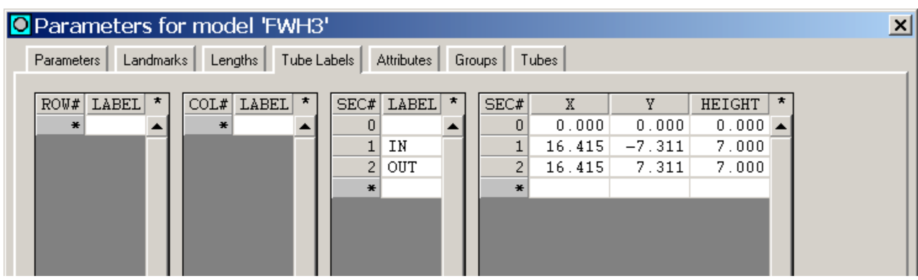


Figure 7-7. Parameters Dialog - Tube Labels Tab

NOTE: ***Sec 0 (zero)*** may be used at the discretion of the user; however, it's easier to keep track of the sections of multi-section components if the first section used is ***1***.

STEP 18: Click ***OK*** on the ***Parameters*** dialog.

NOTE: We'll come back and enter the landmarks later.

Now that the preliminary parameters are setup, let's add the first group of tubes for the component:

STEP 1: Ensure that ***View | Flip Horizontal*** and ***View | Flip Vertical*** are both disabled (i.e. *no checkmark beside either one*).

STEP 2: Click the ***Add Group*** button at the top of the main ***MakeComp*** screen.

STEP 3: Click the ***Parameters*** button to display the ***Group Parameters*** dialog.

STEP 4: In the ***Num Row*** field, type ***23***. (i.e., about 5 more rows than each group of tubes.)

STEP 5: In the ***Num Col*** field, type ***37***. (i.e., about 3 more tubes than the longest row.)

STEP 6: In the ***First Row*** field, type ***1***.

STEP 7: In the ***Group Angle*** field, type ***0***.

STEP 8: In the ***X Offset*** field, type ***0***.

STEP 9: In the ***Y Offset*** field, type ***0***.

STEP 10: In the ***Tube Pitch*** field, type ***0.938***.

STEP 11: In the ***Pitch Angle1*** field, type ***60***.

STEP 12: In the ***Pitch Angle2*** field, type ***60***.

HINT: ***Appendix A*** contains detailed information regarding ***Tube Pitch*** and ***Pitch Angle*** values required by ***MakeComp***.

STEP 13: In the ***Tube ID*** field, type ***0.652***.

STEP 14: In the ***Tube OD*** field, type ***0.750***.

STEP 15: Click the ***Material*** list box. Note that the list box is empty. This is because there are no entries in the ***Materials*** project lookup table. Let's add an entry.

STEP 16: Click ***OK*** on the ***Group Parameters*** dialog, but leave the ***Adding Group*** dialog open.

STEP 17: Click ***Start | Programs | CoreStar | EddyAdmin***.

STEP 18: If necessary, select ***File | Open Project*** and open the ***FWH3A*** project.

STEP 19: Select ***Edit | Project Lookup Tables***.

STEP 20: Click the ***Materials*** tab.

STEP 21: Click the **** (asterisk)*** above the vertical scroll bar. A new row appears at the bottom of the ***Materials*** table.

STEP 22: Click in the ***Name*** field and type ***304SS***.

STEP 23: Click in the ***Conductivity*** field and type ***2.5***.

STEP 24: Click ***OK*** on the ***Project Table*** dialog.

STEP 25: Select ***File | Save***.

STEP 26: Select ***File | Exit***.

TIP:	Use the preceding steps to add or edit entries to any of the 10 ***Project Lookup Tables***. Simply choose the lookup table of interest.

STEP 27: Return to the ***Adding Group*** dialog in ***MakeComp***, click the ***Parameters*** button to display the ***Group Parameters*** dialog.

STEP 28: Click the ***Material*** list box and select ***304SS***.

STEP 29: Click in the ***Sec*** field until ***1*** is displayed.

STEP 30: Ensure the ***Land Set*** field is set to ***0 (zero)***.

STEP 31: Click the ***Numbering*** list box and select ***Row (Skip Spaces)***.

STEP 32: Click the ***Origin*** list box and select ***Upper Left***.

STEP 33: The ***Group Parameters*** dialog should not look like Figure 7-10. Click ***OK*** on the ***Group Parameters*** dialog.

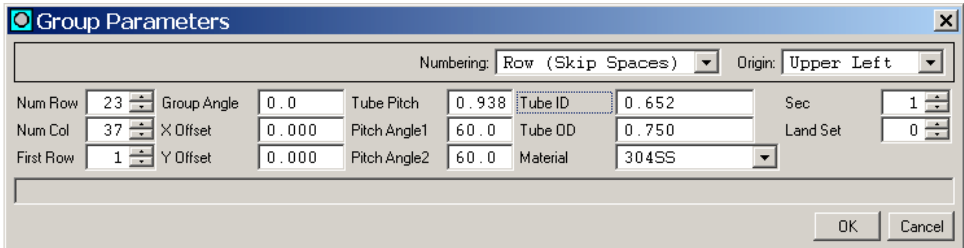


Figure 7-10. Completed Group Parameters Dialog for Section 1

A building grid now appears based on the settings in the ***Group Parameters*** dialog. We'll now enable the group of tubes that exist in the lower (inlet pass) half of the feedwater.

STEP 1: Look at Figure 7-1. Counting the blank spaces in the first row on the lower half (IN), decide on the total number of tubes in row #1. You should come up with ***33***.

STEP 2: Click on the building grid in the *upper left* corner so that ***Grid row=1 col =1*** is displayed in the status bar are the bottom of the ***Adding Group*** dialog.

HINT: On the building grid, rows always are horizontal.

STEP 3: On the numeric keypad, type ***33*** and press the ***Insert*** key on the keyboard. 33 filled (black) tubes should now be displayed for row #1 from left to right.

TIP: If you are not getting the expected results, verify the entries in the ***Group Parameters*** dialog per the preceding sequence of steps.

STEP 4: Look at Figure 7-1. Decide on the total number of tubes in row #2. You should come up with ***34***.

STEP 5: Click on the building grid so that ***Grid row=2 col =1*** is displayed in the status bar at the bottom of the ***Adding Group*** dialog.

STEP 6: On the numeric keypad, press the **** (asterisk)*** key. 34 filled tubes should now be displayed for row #2.

TIP: Using the **** (asterisk)*** on the numeric keypad as described above is a function called ***Symmetrical Insert***. Use it for building groups of tubes where the tube pattern is symmetrical or *balanced* such as this feedwater heater tubesheet map.

STEP 7: Click on the building grid so that the red crosshairs are over the first tube in row #3 (***Grid row=3 col =1***). Relative to the first tube in row #2, you could say, "This tube is down and in one tube from the previous tube".

STEP 8: On the numeric keypad, press the *** (asterisk)*** key. 33 filled tubes should now be displayed for row #3.

STEP 9: Repeat the two steps above for the remaining rows of the lower half of the component before continuing to the next step.

STEP 10: The ***Adding Group*** dialog should now look like the one shown in Figure 7-12. Click ***OK*** on the ***Adding Group*** dialog. ***MakeComp*** returns to the main screen and displays only the enabled tubes on the building grid.

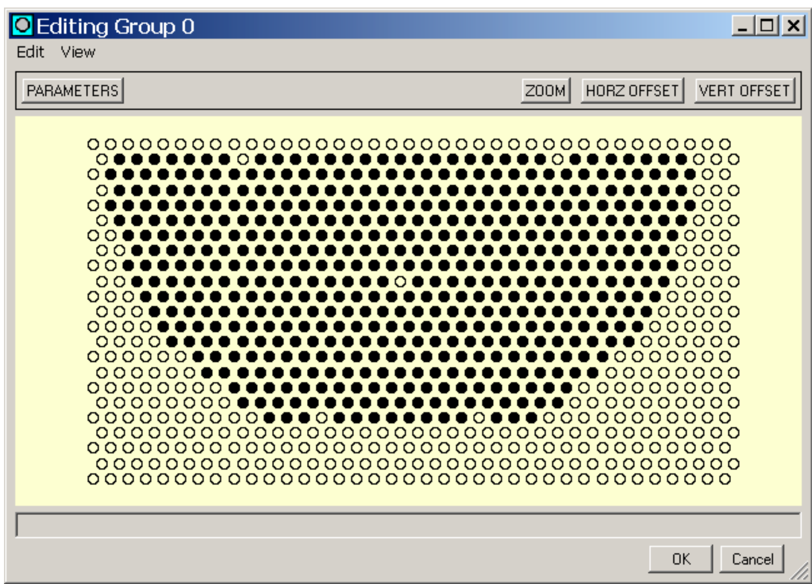


Figure 7-12. Adding Group Dialog - Section 1

Let's temporarily add row tick marks to the map. The row ticks will help us to quickly identify and remove the blank tubes later.

STEP 1: Select ***Edit | Auto Compute Ticks***.

STEP 2: In the ***Row Ticks*** section of the ***Auto Tick Param*** dialog, enable (green) ***Used***, ***Align Ends***, ***First Always***, and ***Last Always***.

STEP 3: In the ***Row Ticks*** section of the ***Auto Tick Param*** dialog, click in the ***Do First/Last By*** field until ***SEC*** is displayed.

STEP 4: In the ***Row Ticks*** section of the ***Auto Tick Param*** dialog, click in the ***Inc*** field until ***5*** is displayed.

STEP 5: In the ***Row Ticks*** section of the ***Auto Tick Param*** dialog, click in the ***Tick Length*** field until ***1.0*** is displayed.

STEP 6: In the ***Row Ticks*** section of the ***Auto Tick Param*** dialog, click in the ***Max Length*** field until ***5.0*** is displayed.

STEP 7: In the ***Row Ticks*** section of the ***Auto Tick Param*** dialog, click in the ***Closest Allowed Tube*** field until ***1.0*** is displayed.

STEP 8: In the ***Col Ticks*** section of the ***Auto Tick Param*** dialog, disable (gray) ***Used***. The remaining settings in this section may be disregarded. The ***Auto Tick Param*** dialog should now look like Figure 7-14.

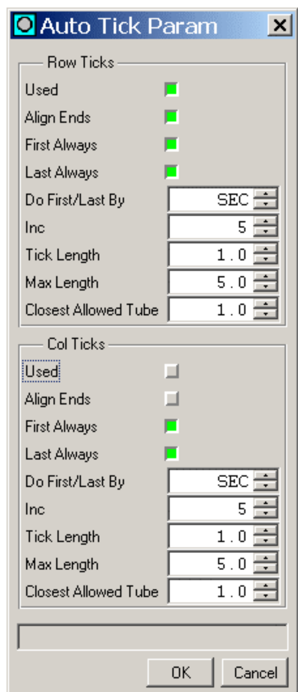


Figure 7-14. Auto Tick Param Dialog

STEP 9: Click ***OK*** on the ***Auto Tick Param*** dialog. Row ticks now appear on every 5th row of the tube group, plus the *first* and *last* rows. Compare the tubesheet map on-screen to the lower half of Figure 7-1.

TIP:	Press ***ESC*** on the keyboard to auto-fit the tubesheet map to the screen.

NOTE:	The section watermark (***IN***) will not appear on-screen. Watermarks only appear on the printed map.

To remove the blank tubes:

STEP 1: Click the 8th tube in row #1. A green cursor box appears around the tube.

STEP 2: Press the ***Shift*** key on the keyboard and click the 8th tube from the end of row #1. A green cursor should now appear only around these two tubes.

STEP 3: Select ***Edit | Delete Selected Tubes***. The selected tubes disappear from the map.

TIP:	If the wrong tubes are deleted, simply select ***Edit	Undo Delete Selected Tubes*** (*Ctrl+Z*) and try again.

STEP 4: Click the center tube (*IN-9-16*) in row #9. A green cursor box appears around the tube.

STEP 5: Select ***Edit | Delete Selected Tubes***. The selected tube disappears from the map.

STEP 6: Click the 4th tube in row #18. A green cursor box appears around the tube.

STEP 7: Press the ***Shift*** key on the keyboard and click the 4th tube from the end of row #18. A green cursor should now appear only around these two tubes.

STEP 8: Select ***Edit | Delete Selected Tubes***. The selected tubes disappear from the map. A total of 504 tubes should now be displayed in the lower left side of the ***MakeComp*** screen.

STEP 9: Select ***File | Save***.

We're about done creating the basic component. Let's create the other half of the component by using the ***Dupe Group*** feature as follows:

STEP 1: Click any tube in the map. A green cursor box appears around the tube.

STEP 2: Right-click the selected tube and select ***Edit Group*** in the popup. The tube group is opened in the ***Editing Group*** dialog (same as the ***Adding Group*** dialog when adding a new tube group).

STEP 3: Select ***Edit | Dupe Group***. The dialog's title changes to ***Adding Group 1***.

STEP 4: Click the ***Parameters*** button. The ***Group Parameters*** dialog appears.

STEP 5: Select ***Lower Left*** in the ***Origin*** list box.

STEP 6: Click in the ***Sec*** field until ***2*** is displayed.

STEP 7: Click in the ***Land Set*** field until ***1*** is displayed.

STEP 8: The ***Group Parameters*** dialog should look like Figure 7-16. Click ***OK*** on the ***Group Parameters*** dialog.

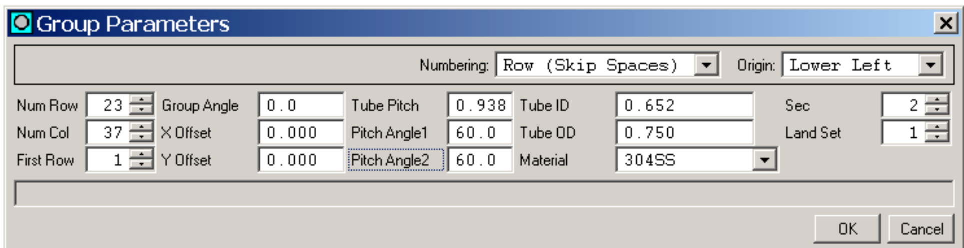


Figure 7-16. Completed Group Parameters Dialog for Section 2

TIP:	If the new tube group appears staggered or zigzagged, simply click the ***Vert Offset*** button in the upper right of the ***Adding Group*** dialog once.

STEP 9: Click ***OK*** on the ***Adding Group*** dialog. The complete map is now displayed on the main ***MakeComp*** screen.

STEP 10: Select ***Edit | Auto Compute Ticks***.

STEP 11: Click ***OK*** on the ***Auto Tick Param*** dialog. The tubesheet map on-screen should now match the one shown in Figure 7-1.

STEP 12: Select ***Edit | Auto Compute Sec Labels***.

STEP 13: Click the ***Parameters*** button on the main ***MakeComp*** screen.

STEP 14: Click the ***Tube Labels*** tab.

STEP 15: Click in the ***Height*** field for ***Sec 1*** and type *7*.

STEP 16: Click in the ***Height*** field for ***Sec 2*** and type *7*.

STEP 17: Click ***OK*** on the ***Parameters*** dialog. When printed, the watermarks will now appear centered in each pass at a reasonable font size.

STEP 18: Select ***File* | *Save***.

STEP 19: Proceed to the next the mini-tutorial, ***Adding Landmarks to a Component File***.

TIPS:	Please see ***Appendix C*** for more information regarding ***MakeComp***.

8. Adding Landmarks to a Component File

In this tutorial, unlike the 'real world', the landmarks and their associated distances are readily available. Many times, however, landmark information is difficult to obtain. In those instances, don't forget that you have a powerful tool available to you called ***eddy current***. Simply physically measure the straight-length, setup a scale in the analysis software, and review eddy current recordings of selected tubes in both halves of the component. Using the scale, landmark distances can easily be measured, noted, and entered accordingly into the component file.

Because the landmarks are typically different in the lower half (*inlet pass*) of a horizontal feedwater heater vs. the upper half (*outlet pass*), each half must be treated as an independent tube bundle with its own landmark set.

Table 8-1 and Table 8-2 each show a list of the landmarks for each landmark set used for each section or *pass*. Each table shows all the values required to setup each landmark set. We will refer to this table while entering the landmarks for the component. So keep it handy.

Table 8-1. Landmarks for Tutorial Component - Section IN (Landmark Set 0)

LAND#	Landmark	NAME	TYPE	LEG	POS(ition)	ANGLE	FB INT	FIRST	LAST	REQ
1	Inlet Tube End	TEI	TEH	Inlet	0.000			1	18	Y
2	Inlet Tubesheet	TSI	TSH	Inlet	6.563			1	18	Y
3	Baffle Plate	B01	BAF	Inlet	30.376			1	18	N
4	Baffle Plate	B02	BAF	Inlet	54.376			1	18	N
5	Baffle Plate	B03	BAF	Inlet	78.376			1	18	N
6	Subcooler End Plate	SEP	SP	Inlet	104.063			1	18	Y
7	TSP	C03	SP	Inlet	156.376			1	18	Y
8	TSP	C04	SP	Inlet	210.376			1	18	Y
9	TSP	C05	SP	Inlet	264.376			1	18	Y
10	TSP	C06	SP	Inlet	315.376			1	18	Y
11	TSP	C07	SP	Inlet	364.376			1	18	Y
12	TSP	C08	SP	Inlet	415.376			1	18	Y
13	TSP	C09	SP	Inlet	469.376			1	18	Y
14	TSP	C10	SP	Inlet	523.376			1	18	Y

Table 8-2. Landmarks for Tutorial Component – Section OUT (Landmark Set 1)

LAND#	Landmark	NAME	TYPE	LEG	POS(ition)	ANGLE	FB INT	FIRST	LAST	REQ
1	Outlet Tube End	TEO	TEH	Inlet	0.000			1	18	Y
2	Outlet Tubesheet	TSO	TSH	Inlet	6.563			1	18	Y
3	TSP	H01	SP	Inlet	54.376			1	18	Y
4	TSP	H02	SP	Inlet	102.376			1	18	Y
5	TSP	H03	SP	Inlet	156.376			1	18	Y
6	TSP	H04	SP	Inlet	210.376			1	18	Y
7	TSP	H05	SP	Inlet	264.376			1	18	Y
8	TSP	H06	SP	Inlet	315.376			1	18	Y
9	TSP	H07	SP	Inlet	364.376			1	18	Y
10	TSP	H08	SP	Inlet	415.376			1	18	Y
11	TSP	H09	SP	Inlet	469.376			1	18	Y
12	TSP	H10	SP	Inlet	523.376			1	18	Y

NOTES:

- No values are required in either the ***Angle*** or ***FBI NT*** columns.
- ***POS*** values are cumulative from the tube end.
- Since each section is treated independently, ***Inlet*** must be used in the ***LEG*** column in both landmark sets – even in ***Landmark Set 1*** for the outlet section.
- The ***Cold*** leg of a feedwater heater is the inlet, while the ***Hot*** leg is the outlet; hence, the ***C*** and ***H*** used in the ***Names***.
- Landmark types ***TEC*** and ***TSC*** are only used in either straight-tubed components OR in U-tubed components where only one-half of the component is used as the tubesheet map, i.e., steam generators.

Let's get started by entering the landmarks for ***Landmark Set 0***.

STEP 1: Assuming the tutorial project (***FWH3A***) is opened in ***MakeComp***, click the ***Parameters*** button at the top of the ***MakeComp*** screen.

STEP 2: Click the ***Landmarks*** tab on the ***Parameters*** dialog.

STEP 3: In the ***Set*** list box, select ***0***.

HINT:	We've already assigned ***Landmark Set 0*** to tube ***Group 0***, which was assigned to ***Section 1***. All this was done in the ***Group Parameters*** dialog for the lower half of the component.

STEP 4: Click in the ***Name*** field of the first row of the ***Landmark*** table and type ***TEI*** (*Tube End Inlet*).

TIP:	The landmark ***Name*** can be any user-defined term. This is the landmark label that will appear in the landmark strip in the analysis software.

STEP 5: Click in the ***Type*** field of the first row of the ***Landmark*** table and type ***TEH***.

NOTE:	***TEH, TSH, SP, BAF, BW, AVB, FB, RB, VS, TSC,*** and ***TEC*** are reserved *landmark types*. For auto-locate to work properly during analysis, ***TEH*** must be the first landmark type entered at the top of all landmark tables.

STEP 6: Click in the ***Leg*** field of the first row of the ***Landmark*** table and type the letter ***I***. ***INLET*** will automatically be entered by the software.

STEP 7: Click in the ***Pos(ition)*** field of the first row of the ***Landmark*** table and type ***0***.

STEP 8: Leave all ***Angle*** and ***FB Int*** fields blank for the tutorial.

STEP 9: Click in the ***First*** field of the first row of the ***Landmark*** table and type ***1***. This defines the first row where this landmark is expected to be present.

STEP 10: Click in the ***Last*** field of the first row of the ***Landmark*** table and type ***18***. This defines the last row where this landmark is expected to be present.

NOTE:	The component shown in Figure 7-1 has 18 rows of tubes in each half. The default value of ***999*** for the ***Last*** field would be acceptable as well.

STEP 11: Click in the ***Req(uired)*** field of the first row of the ***Landmark*** table and type ***Y(es)***. This forces auto-locate to label this landmark for every tube in the **First - Last** range of rows.

STEP 12: Click the **** (asterisk)*** above the vertical scroll bar. A new row appears in the ***Landmark*** table.

STEP 13: Continue entering the remaining landmarks for ***Landmark Set 0*** while referring to Table 8-1. When you have completed this step, continue to the next series of steps to enter the landmarks for ***Landmark Set 1*** (upper half). When you have finished, the ***Landmarks*** tab of the ***Parameters*** dialog should look like the one shown in Figure 8-1.

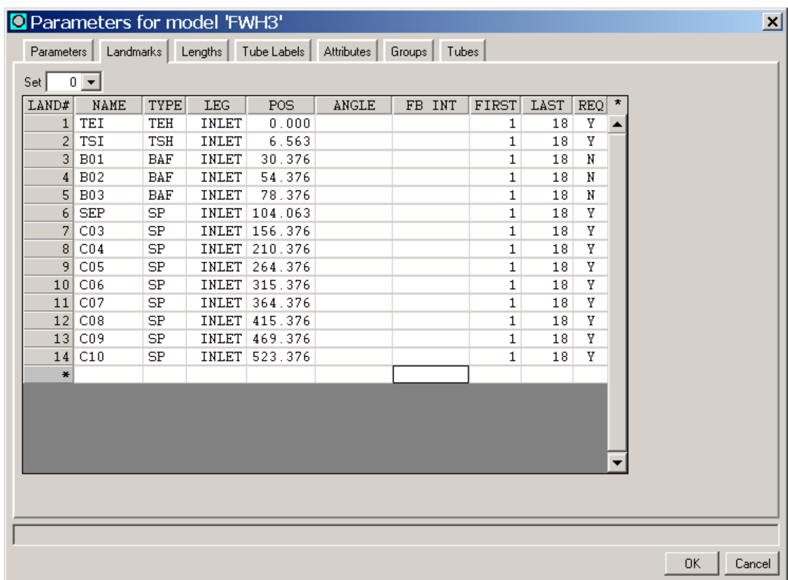


Figure 8-1. Parameters dialog - Landmarks Tab Showing Landmark Set 0

To enter the landmarks for ***Landmark Set 1***:

STEP 1: Ensure that the ***Landmarks*** tab of the ***Parameters*** dialog is currently being displayed.

STEP 2: In the ***Set*** list box, select ***1***. A blank landmark table will appear.

HINT: We've already assigned ***Landmark Set 1*** to tube ***Group 1***, which was assigned to ***Section 2***. All this was done in the ***Group Parameters*** dialog for the upper half of the component.

STEP 3: Enter all the landmarks shown in Table 8-2 using the same techniques as in the previous series of steps for entering the landmarks in ***Landmark Set 0***.

STEP 4: When you have finished, the ***Landmarks*** tab of the ***Parameters*** dialog should look like the one shown in Figure 8-3. Click ***OK*** on the ***Parameters*** dialog.

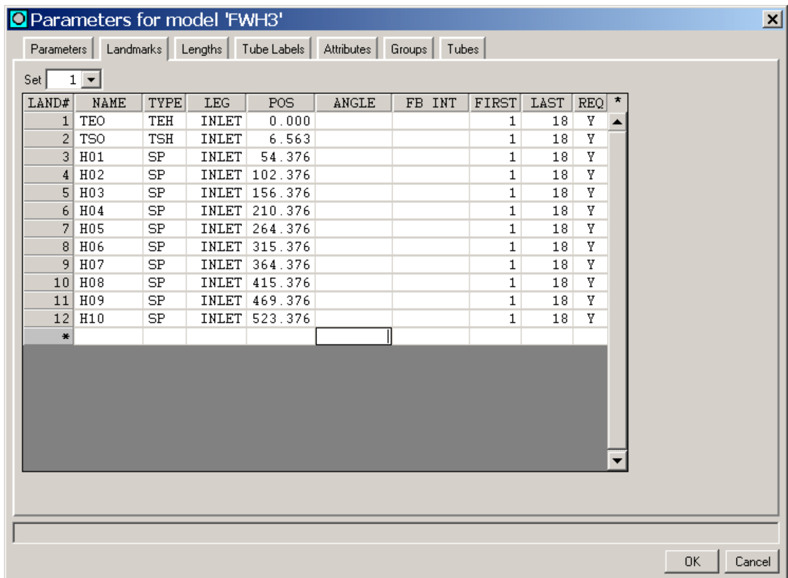


Figure 8-3. Parameters dialog - Landmarks Tab Showing Landmark Set 1

Finally, we should double-check the landmark set assignment as follows:

STEP 1: Right-click on the tubesheet map and select ***Select Landmark Set | 0*** from the popup menu. All tubes assigned to ***Landmark Set 0*** should be highlighted. If not, right-click on the tubesheet map and select ***Reset Landmark Sets*** from the popup menu.

STEP 2: Right-click on the tubesheet map and select ***Select Landmark Set | 1*** from the popup menu. All tubes assigned to ***Landmark Set 1*** should be highlighted.

STEP 3: Select ***File | Save*** on the main menu bar of ***MakeComp***.

STEP 4: Exit ***MakeComp***.

9. Creating an Inspection Plan

This topic provides steps to create a typical inspection plan for the component we'll be inspecting. The criteria for our inspection sample include:

- All periphery tubes in both passes.
- Around existing plugged tubes.
- A pattern sample across the inlet pass.
- All tubes still in-service with historical damage.

The resulting inspection plan file (***.pln***) is used by acquisition as an electronic checklist. The acquisition software, in conjunction with the plan file, keeps track of the status of all tubes on the plan until acquisition is completed. A graphical display of the tubesheet map with the current inspection plan status is readily available at anytime during acquisition. Additionally, uploading the plan(s) to the database will provide you with the ability to query for missing tube records during database closeout.

Although there are different ways to add tubes to an inspection plan, the method shown in this tutorial simply plots symbols on all the tubes of interest before adding them to the plan all at once. You will develop your own favorite method as you become more accustomed to the use of the software.

Before we create an inspection plan, we need to create a new database to use for our project. To create a new database:

STEP 1: Double-click the ***DBMS*** icon on the desktop. The ***FWH3A*** project should open. If not, select ***File | Open Project***, select ***c:\Projects\FWH3A***, and click ***Open***.

STEP 2: Select ***Database | New Database***.

STEP 3: On the ***Save As*** dialog that appears, enter ***project.mdb*** as the name for the new database and click ***Save***. The software returns to the main ***DBMS*** screen with the new database automatically opened (*connected*).

To create the first part (*all periphery tubes in both passes*) of the inspection plan:

STEP 1: Click on tube ***IN-1-1*** so that a green box cursor highlights the tube.

STEP 2: Shift+click all remaining periphery tubes in the inlet pass, i.e., "connect the dots", so that they're all highlighted green. (*Only 1 tube in around*).

TIP:	Be careful to not let go of the Shift key; otherwise, clicking once will clear all your selections and you'll have to start over.

STEP 3: Click the ***List*** button at the upper right of the ***DBMS*** screen.

STEP 4: On the ***DBMS List*** dialog, select ***Edit | Add Selected***, and click ***OK***.

STEP 5: Click ***No*** on the "Do you wish to save changes?" message box. A symbol (*probably a red letter 'A'*) is now plotted on each of the tubes you had selected.

STEP 6: Right-click entry number ***1*** in the # column of the legend and click ***Ignore Section*** in the popup to enable it. When enabled, a checkmark appears next to ***Ignore Section***. The same tubes as in the inlet pass now appear with symbols in the outlet pass as shown in Figure 9-1 below.

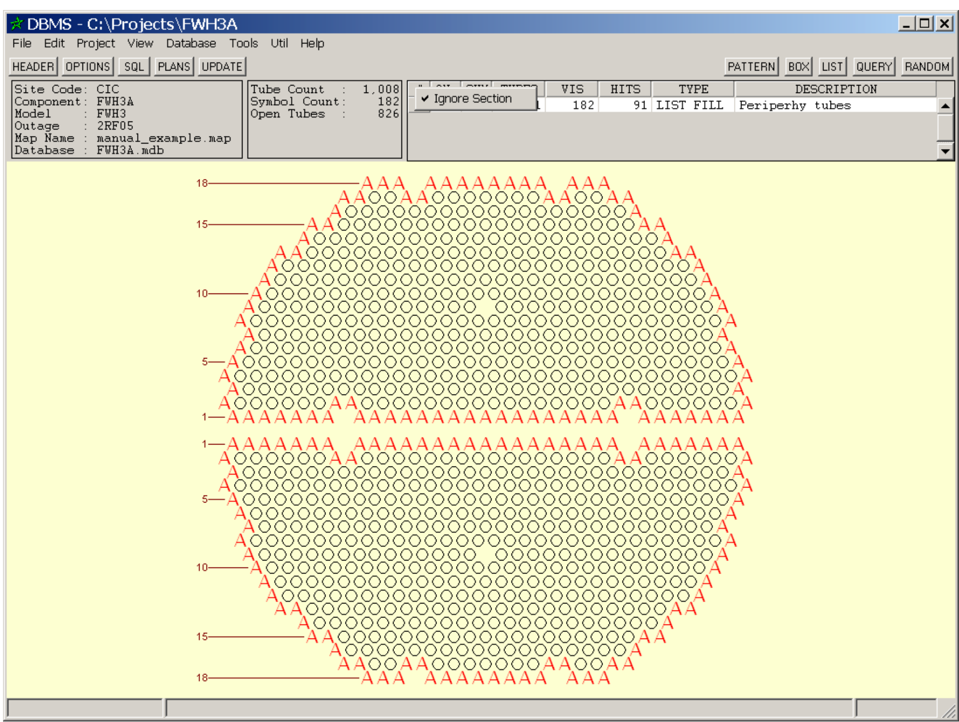


Figure 9-1. Periphery Plan in DBMS Screen

To create the second part (*around existing plugged tubes*) of the inspection plan, we first need to add the plugged tubes manually. If this was an existing database with historical records including plugged tubes, we could simply query the *repairs* table to quickly plot the plugged tubes.

To add the plugged tubes to the *repairs* table, we'll type them into the ***List*** dialog for the inlet pass only. You could select them as well – much in the same way that we selected the periphery tubes in the preceding steps.

STEP 1: Click the ***List*** button at the upper right of the ***DBMS*** screen.

STEP 2: On the ***DBMS List*** dialog, click in the ***Pass (Sec)*** field of the first row and type ***1***, and then press ***Enter***. ***IN*** is displayed and the cursor moves to the ***Row*** field.

STEP 3: In the ***Row*** field of the first row, type ***2***, and then press ***Enter***. The cursor moves to the ***Tube*** (***Col)*** field.

STEP 4: In the ***Tube*** (***Col)*** field of the first row, type ***20***, and then press ***Enter***. The cursor moves to the ***Pass (Sec)*** field of the second row.

STEP 5: Continue entering the following 11 tubes using the same technique:

- IN-2-22.
- IN-7-16.
- IN-7-20.
- IN-7-24.
- IN-9-5.
- IN-11-21.
- IN-13-12.
- IN-13-23.
- IN-14-2.
- IN-15-5.
- IN-16-12.

Afterwards, the ***List*** dialog should look like the one shown in Figure 9-3.

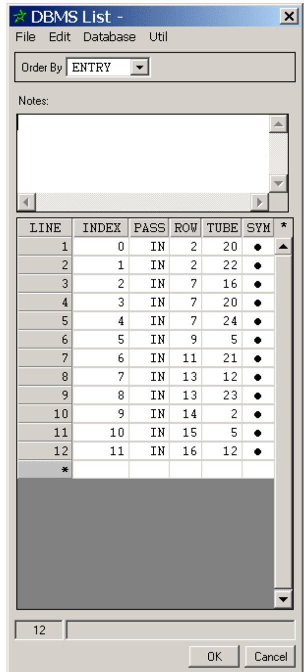


Figure 9-3. Using the List Dialog Manually

STEP 6: Click ***OK*** on the ***List*** dialog. A total of 12 tubes should now be displayed on the inlet pass (*lower*) of the tubesheet map.

STEP 7: Right-click entry number ***2*** in the # column of the legend and click ***Ignore Section*** in the popup to enable it. The same plugged tubes as in the inlet pass now appear with symbols in the outlet pass.

STEP 8: Turn off the periphery tube symbols by clicking the ***Yes*** in the ***ON*** column of the legend for the first entry. Only the plugged tubes in both passes should now be displayed.

STEP 9: Right-click the tubesheet map and select ***Repairs***. The ***Repairs*** dialog appears.

STEP 10: On the ***Repairs*** dialog, click ***Schedule | Tubes With Symbols***. The ***Repair Attributes*** dialog appears.

STEP 11: In the ***Outage*** field, type ***LEGACY***. These are historical plugged tubes that we're adding to the ***repairs*** table of the database, *NOT* new plugged tubes based on the results of the current outage. You'll add those after the inspection when recommending tubes for plugging.

STEP 12: In the ***Repair Type*** list box, select ***Plug***.

STEP 13: In the ***Type*** list box, type ***Pop-a-plug***.

STEP 14: Enable the ***Permanent*** button (*green*).

STEP 15: In the ***Material*** list box, select ***SS***.

STEP 16: In the memo area at the bottom of the ***Repair Attributes*** dialog, type ***Plugged prior to 2RF05***.

NOTE: The remaining fields of the ***Repair Attributes*** dialog are optional. Use them at your discretion.

STEP 17: Your ***Repair Attributes*** dialog should look like the one shown in Figure 9-5. Click ***OK*** on the ***Repair Attributes*** dialog. 24 plugged tubes now exist in the *repairs* table of the database and are visible in the ***Repairs*** dialog.

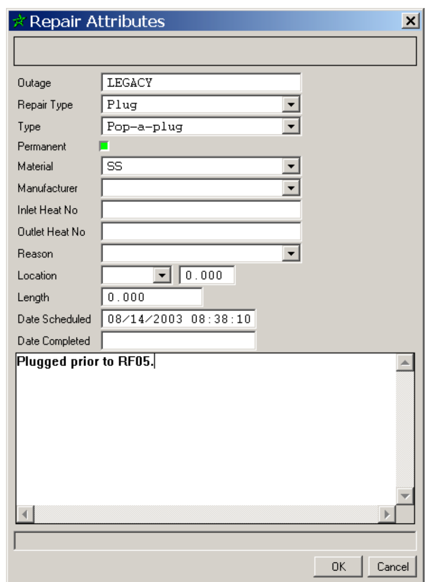


Figure 9-5. Repair Attributes Dialog

STEP 18: Select ***Complete | All Scheduled Repairs***. This step 'cleans up' the entries for the previously plugged tubes so that they are not flagged as pending to be plugged (*scheduled*).

STEP 19: Click ***OK*** on the ***Repairs*** dialog. This action finalizes and accepts these new records into the *repairs* table of the database.

Now that we have records (*plugs*) in the ***repairs*** table of our new database, we can quickly create the second part (*around existing plugged tubes*) of the inspection plan as follows:

STEP 1: Click the button for entry number ***2*** in the # column of the legend so that it appears depressed, and then press ***Delete*** on the keyboard. Click ***Yes*** on the message box. This was the temporary list fill we created for the plugged tubes, but we no longer need it.

STEP 2: Click the ***Query*** button is the upper right of the ***DBMS*** screen. The ***Query*** dialog appears.

STEP 3: Click ***File | New***.

STEP 4: Type in the following query - exactly:

```
SELECT sec, row, col
FROM repairs
WHERE repair_type = 'PLUG'
```

STEP 5: Click the ***Execute*** button. The list of plugged tubes appears at the bottom of the ***Query*** dialog. Click ***OK*** to plot the tubes on the tubesheet map.

STEP 6: In the legend, right-click the symbol (*probably the letter 'B'*) for the newly plotted plugged tubes.

STEP 7: On the ***Choose Symbol*** dialog, click the ● symbol (*6th row down – 3rd from the end*).

STEP 8: Right-click the large colored button on the left side of the ***Choose Symbol*** dialog and pick the color black, and then click ***OK*** on the ***Color*** dialog.

STEP 9: Click ***OK*** on the ***Choose Symbol*** dialog. The symbols for the plugged tubes should now appear as black filled circles on the tubesheet map.

STEP 10: Right-click any plugged tube and select ***Select | Symbol*** from the popup menu. Green cursor boxes should now appear on all the plugged tubes.

STEP 11: Click the ***Box*** button at the upper right of the ***DBMS*** screen.

STEP 12: Enter a value of ***1.200*** in the ***Box*** dialog and click ***OK***. New symbols should now appear surrounding or bounding the plugged tubes (See Figure 9-7). You can experiment with the value to get the results you desire.

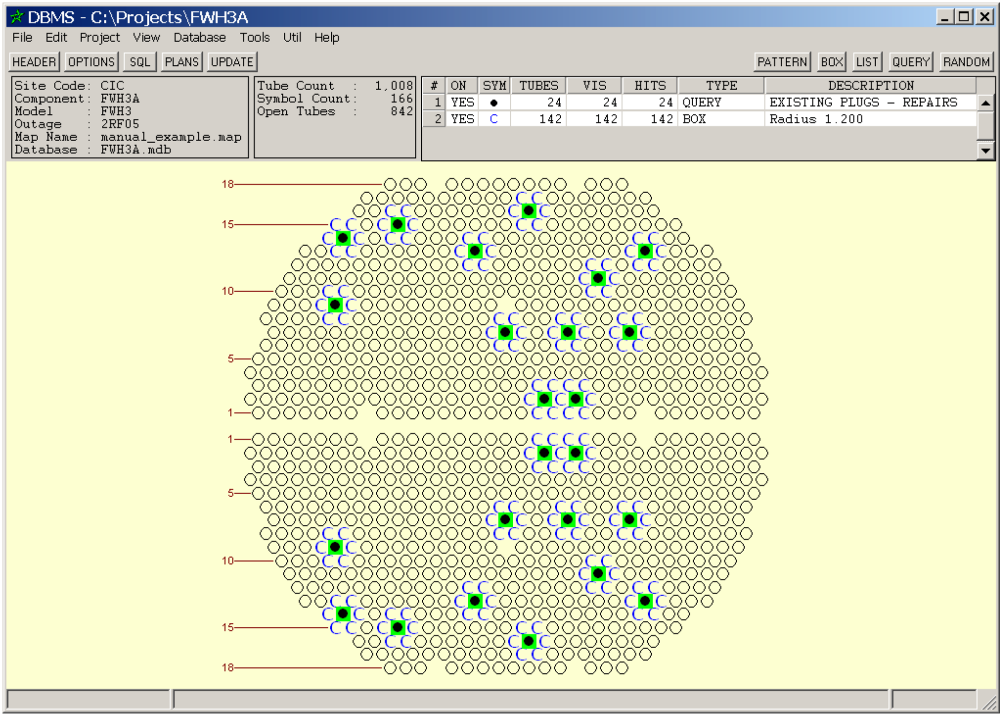


Figure 9-7. Bounding Existing Plugged Tubes Using the Box Dialog

To create the third part (*a pattern sample across the inlet pass*) of the inspection plan:

STEP 1: In the legend, turn on all entries. There should be only 3 entries, i.e., periphery tubes, plugged tubes, and tubes around plugs.

TIP: You can quickly toggle all legend entries on and off by clicking the column header ***ON*** in the legend.

NOTE: Don't worry if some of the inspection plan tubes overlap. A tube will be added to an inspection only once regardless of the number of symbols on it.

STEP 2: Right-click on the inlet pass (*lower half*) and select ***Select | Section***.

STEP 3: Click the ***Pattern*** button is the upper right of the ***DBMS*** screen.

STEP 4: In the ***Region*** list box, select ***Selected Tubes***.

STEP 5: In the ***Tube*** list box, select ***Open Tubes Only***.

STEP 6: Enable both ***Include First Tube*** and ***Include Last Tube***. Both buttons will turn green. This forces the pattern fill to include the first and last tubes in each row regardless of the pattern sequence.

STEP 7: Set the ***First Row*** field to ***1***.

STEP 8: Set the ***Row Inc*** field to ***2***.

STEP 9: Set the ***First Col*** field to ***1***.

STEP 10: Set the ***Col Inc*** field to ***2***. The ***Pattern Fill*** dialog should now look like Figure 9-9.

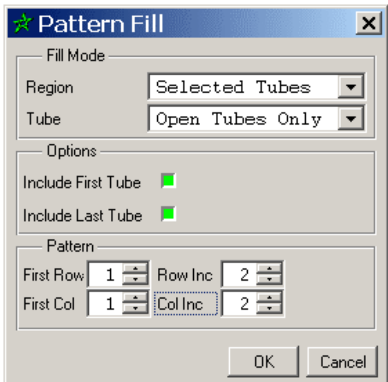


Figure 9-9. Pattern Fill Dialog

STEP 11: Click ***OK*** on the ***Pattern Fill*** dialog. A 2x2 pattern of 82 tubes should now appear in inlet pass only (See Figure 9-11).

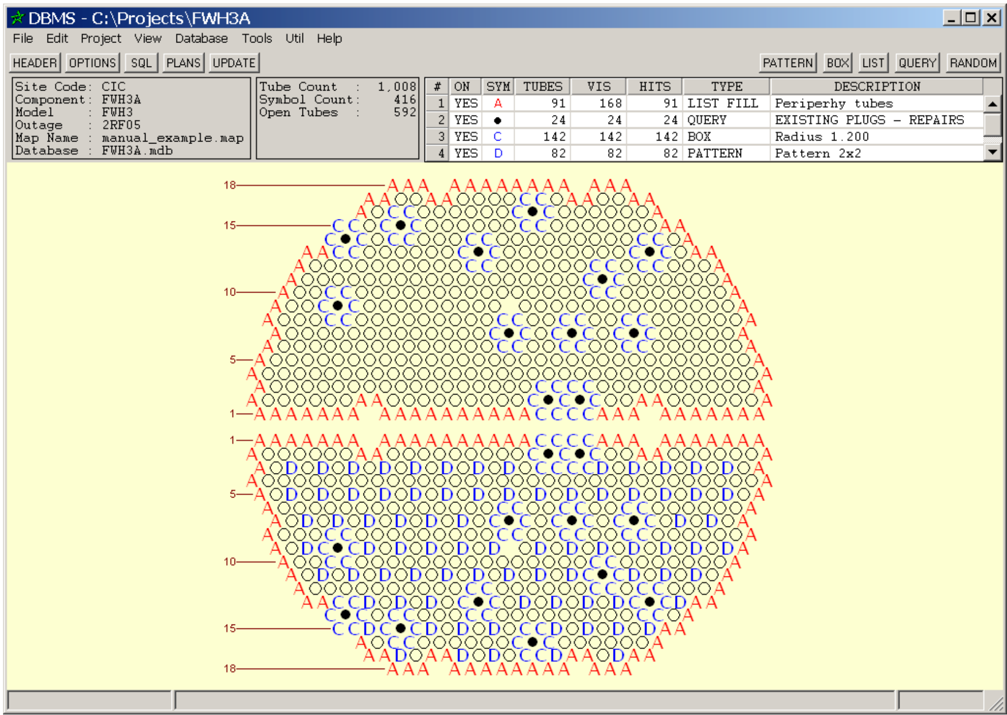


Figure 9-11. Using the Pattern Fill Dialog on a Selected Section

STEP 12: Click anywhere on the map to clear the section selection.

To create the fourth and final part (*all tubes still in-service with historical damage*) of the inspection plan, follow the steps below.

NOTE:	Since this is a new database, no previous inspection data is present; however, the steps are provided for your reference in developing future inspection plans.

STEP 1: Click the ***Query*** button is the upper right of the ***DBMS*** screen. The ***Query*** dialog appears.

STEP 2: Click ***File | New***.

STEP 3: Type in the following query (which assumes the prior outage is 2RF04):

```
SELECT outage, sec, row, col, volts, phase, defect,
pcnt, loc_land, loc_off, beg_test, end_test
FROM report
WHERE pcnt >= 20
AND outage = '2RF04'
ORDER BY sec, row, col;
```

STEP 4: Click the ***Execute*** button. The list of all tubes with eddy current calls ≥20% appears at the bottom of the ***Query*** dialog. Click ***OK*** to plot the tubes on the tubesheet map.

All tubes that we want to inspect and the plugged tubes should now be plotted on the map. A total symbol count of 416 tubes should be displayed in the upper center area of the ***DBMS*** screen as shown in Figure 9-11. Now that we have identified all the tubes to be inspected for ***2RF05***, let's finish up creating the inspection plan.

STEP 1: Since plugged tubes are not typically included on an inspection plan, turn on all legend entries, except for the plugged tubes.

STEP 2: Click the ***Plans*** button in the upper left of the ***DBMS*** screen. The ***Plan*** dialog appears.

STEP 3: Select ***File | New Plan***.

STEP 4: In the ***Order By*** list box, select ***ROW***.

STEP 5: The ***Probe, Beg Test,*** and ***End Test*** list boxes are optional. Leave them blank for now.

STEP 6: In the ***Description*** field enter, "*2RF05 sample*."

STEP 7: Select ***Edit | Load Symbols***. The ***Plan*** dialog should now appear as shown in Figure 9-13.

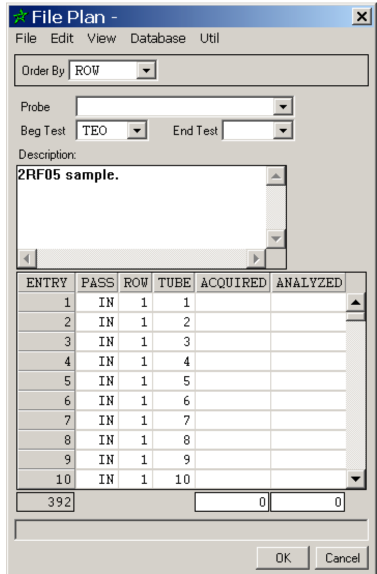


Figure 9-13. Plan Dialog with Sample Plan Loaded

`STEP 8:` Select ***Database* | *Upload Plan***.

`STEP 9:` In the ***Plan Name*** field, enter *"2RF05-1"*. This action copies the plan to the database only, which is handy for querying later.

TIP:	Multiple inspection plans can be created for the same project, i.e., more than one material, wall thickness, size, plans for different probes, etc.

`STEP 10:` Select ***File* | *Save As***.

`STEP 11:` The ***File Name*** field should automatically show *"2RF05-1"* – if not, simply type it in, and then click ***Save***.

`STEP 12:` Click ***OK*** on the ***Plan*** dialog.

A typical inspection plan now exists in the database for querying later in the ***DBMS*** software, while a plan file (***`pln`***) exists in the ***`\plans`*** directory of this project for use with the acquisition software.

10. Acquiring Data

This topic provides continuing steps for the tutorial project to acquire eddy current data using the acquisition software module of EddyVISION32 in conjunction with a CoreStar OMNI-100 tester.

In order to prepare for the following steps, setup the tester for a typical acquisition procedure by connecting a set of bobbin probes (MF) for differential and absolute data acquisition. A typical calibration standard, simulated tube support, and reference section of tubing material will be required. The component for our fictitious project contains Type 304 stainless steel tubes measuring 0.750-inch OD x 0.049-inch Wall.

Ensure there is an ***ID*** entry in the ***Users*** table of the project lookup tables for you by following these steps:

STEP 1: Start ***EddyAdmin*** and open the tutorial project (***FWH3A***).

STEP 2: Select ***Edit | Project Lookup Tables***.

STEP 3: Click the ***Users*** tab.

STEP 4: If no user ***ID*** entry exists for you, click the **** (asterisk)*** above the scroll bar. A new row appears at the bottom of the ***Users*** table.

STEP 5: Click in the ***ID*** field of the new row.

STEP 6: Type your user ***ID***. The ***ID*** can be anything you want to use.

> **TIP:** A common user ***ID*** format is:
>
> ***[First letter of lastname]+[Last 4 digits of SSN]***
>
> For example, the user ***ID*** for John Doe who's SSN is 123-45-6789 would be *D6789*.

STEP 7: Click in the ***User Name*** field of the new row.

STEP 8: Type your name.

STEP 9: Click in the ***Level*** field of the new row.

STEP 10: Type your NDE level of certification, i.e., I, II, IIA, III, etc.

STEP 11: Click ***OK***.

`STEP 12:` Select ***File* | *Save***.

`STEP 13:` Exit ***EddyAdmin***.

To start the acquisition software and logon:

`STEP 1:` Start ***CoreStar*** by double-clicking the "yellow star" icon. The ***User Logon*** dialog appears. This software module provides both data acquisition and analysis functions.

`STEP 2:` In the ***User ID*** list box, select your ***User ID***. The remaining fields of the ***User Logon*** dialog will auto-complete.

`STEP 3:` Click ***OK***. The acquisition/analysis software screen appears.

`STEP 4:` Select ***Screen* | *Test***.

TIPS:

- You may design up to 16 different screens by selecting ***Edit* | *Screens***.
- Click the ***Options*** button, click the ***Colors*** tab, then *right-click* in any field under ***DSET 1*** to select desired colors of the various screen components.

In the extreme upper right corner of the acquisition screen, there is a box, which displays the current operator type, i.e., ***PRI***, ***SEC***, ***ACQ***, etc. We need to set this to ***ACQ*** for acquisition. All subsequent setup and report files will be encoded as ***ACQ*** as part of the filename.

To set the operator type as ***ACQ***:

`STEP 1:` Click the ***Options*** button at the top of the acquisition screen. The ***Options*** dialog appears.

`STEP 2:` Click the ***Report*** tab.

`STEP 3:` In the ***Analysis Type Dset 1*** list box, select ***ACQ***.

`STEP 4:` Click ***OK***. The operator type box on the main acquisition screen now shows ***ACQ*** as shown in Figure 10-1.

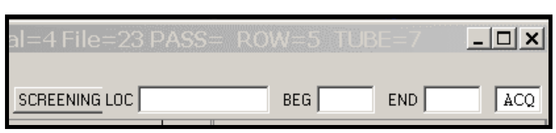


Figure 10-1. Current Operator Type Display Box

Next, let's load the inspection plan that we created earlier. Many of the acquisition settings and controls discussed in the following steps are accessed via the ***Test Panel***. The ***Test Panel*** is the gray area located on the right side of the acquisition screen immediately adjacent to the rightmost Lissajous.

To load the inspection plan:

STEP 1: Click the ***Plan*** button located in the ***Test Panel***. The ***Plan*** dialog appears.

STEP 2: Disable ***Serpentine***. The little box beside ***Serpentine*** turns gray (*not green*).

NOTE: Enabling ***Serpentine*** (*snake-like*) modifies the inspection path so that at the end of each row (*or column*), the next tube will be the last tube in the next row (*or column*). Not recommended when using manual probe positioning.

STEP 3: In the ***Order By*** list box, select ***Row***.

STEP 4: Select ***File | Open Plan***. A dialog appears displaying the available plan files (***`pln`***) within the current project.

STEP 5: Select ***2RF05-1.pln***. This is the plan we created in an earlier topic.

STEP 6: Click ***Open***. The planned tubes appear with a blue letter "A" over each one as shown in Figure 10-3.

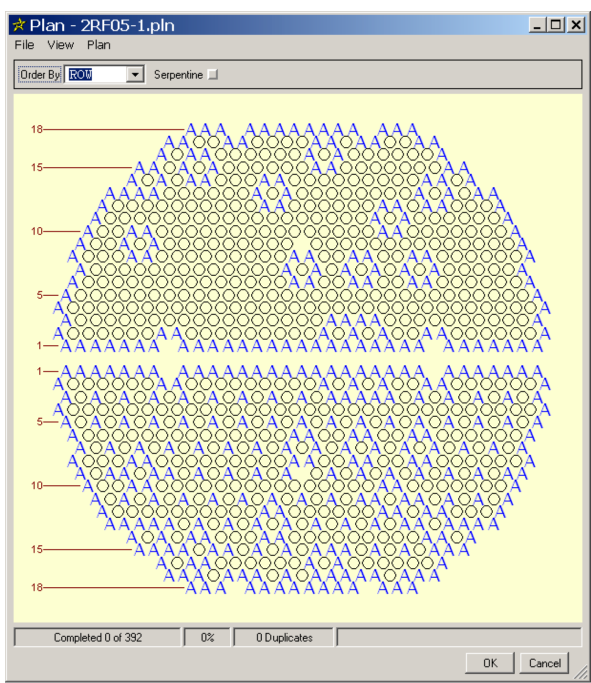


Figure 10-3. Plan Dialog with Plan Loaded in Acquisition

STEP 7: Select ***View | Path***. Based on the selection in the ***Order By*** field, a red line shows the computed path for the inspection.

STEP 8: Click ***OK***. The plan now appears as a list at the bottom of the ***Test Panel***.

NOTES:

- To ***Zoom In*** on the map in the ***Plan*** dialog, ***Ctrl+Click***.
- To ***Zoom Out*** on a map in the ***Plan*** dialog, ***Ctrl+Right-click***.
- To ***Auto-zoom*** a map in the ***Plan*** dialog, ***Ctrl+Middle-click***.
- To ***Move*** or ***Pan*** a map in the ***Plan*** dialog, ***Middle-click+Drag***.
- You may view the ***Plan*** dialog at anytime to review the progress and the available statistics of the inspection.

We now need to set some basic test options as follows:

STEP 1: Click the ***Test Options*** button located in the ***Test Panel***. The ***Test Options*** dialog appears.

STEP 2: Enable both ***Tube Must Exist*** and ***Warn If Done***.

STEP 3: Set the ***Cal Warning*** field to ***10 min***.

STEP 4: Set the ***Cal Timeout*** field to ***04:00*** (i.e 4 hours and 0 minutes).

STEP 5: Set the ***Min Free Disk*** field to ***10 MB***.

STEP 6: Set the ***Max Cal Size*** field to ***Unlimited***.

STEP 7: Disable the ***Disable Pusher*** option (*not green*).

STEP 8: Set the ***Acquire During*** field to ***Pull***.

STEP 9: Set the ***Display Buffer Size*** field to ***50,000 pts***.

STEP 10: Set the ***Stripchart Scale*** field to ***40 pts/pix***.

STEP 11: Set the ***Refresh Interval*** field to ***0.0 sec***.

STEP 12: Enable the ***Auto Balance*** option (*green*).

STEP 13: Enable the ***Auto Scale*** option (*green*).

STEP 14: Disable the ***Auto Locate*** option (*not green*).

STEP 15: Disable the ***Auto Analyze*** option (*not green*).

STEP 16: The ***Test Options*** dialog should now look like Figure 10-5. Click ***OK***.

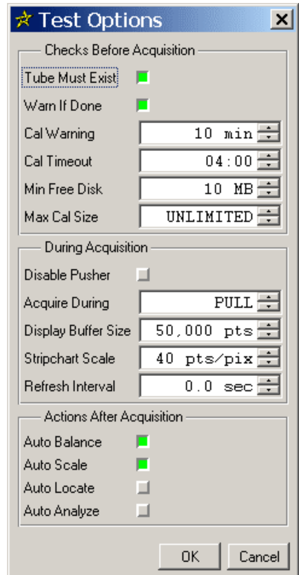


Figure 10-5. Test Options Dialog

To set the tester's configuration, i.e., inspection frequencies, sample rate, etc., follow these steps:

STEP 1: Click the ***Test Config*** button located in the ***Test Panel***. The ***OMNI Test Config*** dialog appears.

STEP 2: Click the ***Options*** button.

STEP 3: Enable the ***Advanced Config*** option (*green*) and click ***OK***.

STEP 4: Click ***Cancel*** on the ***OMNI Test Config*** dialog.

STEP 5: Click the ***Test Config*** button again. The ***OMNI Test Config*** dialog appears, but this time in ***Advanced Config*** mode.

STEP 6: Set the ***Sample Rate*** field to ***2,000***.

STEP 7: Set the ***Filter Cutoff*** field to ***2.5Khz***.

STEP 8: On the ***Drivers*** panel (*lower left*), ensure that tab ***1*** is selected.

STEP 9: On the ***Coil*** panel (*lower right*), ensure that tab ***1*** is selected.

STEP 10: In the ***Coil*** panel, click the first 4 buttons under ***Coil 1***. The buttons will change to green. Corresponding *frequency slots* will appear in the ***Drivers*** panel as well. ***Coil 1*** in this configuration will provide the differential channels for our inspection. If necessary, click the ***Coil 1*** tab until the letter ***'D'*** (*differential*) appears in the 4 green buttons.

STEP 11: In the ***Coil*** panel, click the first 4 buttons under ***Coil 3***. The buttons will change to green. ***Coil 3*** in this configuration will provide the absolute channels for our inspection. If necessary, click the ***Coil 3*** tab until the letter ***'A'*** (*absolute*) appears in the 4 green buttons.

STEP 12: On the ***Drivers*** panel, *click and hold* on the left side of the decimal in ***Slot 1*** under the ***Freq*** column to set the value to ***300***.

TIPS:	• ***Shift+clicking*** or ***Shift+right-clicking*** in increment or decrement fields will change the value in tens vs. units. • ***Middle-clicking*** in increment or decrement fields will reset the value to zero.

STEP 13: Ensure that the range for all 4 frequency slots is set to ***KHz***. *Click or right-click* to change as necessary.

STEP 14: On the ***Drivers*** panel, *click and hold* on the left side of the decimal in ***Slot 2*** under the ***Freq*** column to set the value to ***150***.

STEP 15: On the ***Drivers*** panel, *click and hold* on the left side of the decimal in ***Slot 3*** under the ***Freq*** column to set the value to *75*.

STEP 16: On the ***Drivers*** panel, *click and hold* on the left side of the decimal in ***Slot 4*** under the ***Freq*** column to set the value to ***35***.

STEP 17: Enable the ***Auto Config*** option (*green*). The ***Delay*** and ***Gain*** values in the ***Drivers*** panel will be calculated automatically. In addition, the ***Time Used***, ***Time Avail***, and ***Oversamples*** values are calculated as well.

STEP 18: *Shift+right-click* the ***Gain*** column header until all gain values are ***75.0%***.

NOTE:	Each time the ***Auto Config*** option is toggled, the ***Delay*** and ***Gain*** values are maximized depending on the frequency.

STEP 19: Select ***File | Save As***.

STEP 20: In the ***File name*** box, type ***FWH3*** and click ***Save***. This configuration file (***cfg***) can now easily be recalled and/or shared with other users.

STEP 21: Click the ***Test Link*** button. A message should appear in the status bar at the bottom of the ***OMNI Test Config*** dialog indicating that the TCP/IP network link between the tester and PC is okay. If not, please refer to the technical manual for your tester to resolve the communication error, and then return to this step in the tutorial.

STEP 22: The ***OMNI Test Config*** dialog should now appear setup as shown in Figure 10-7 below. Click ***OK*** on the ***OMNI Test Config*** dialog.

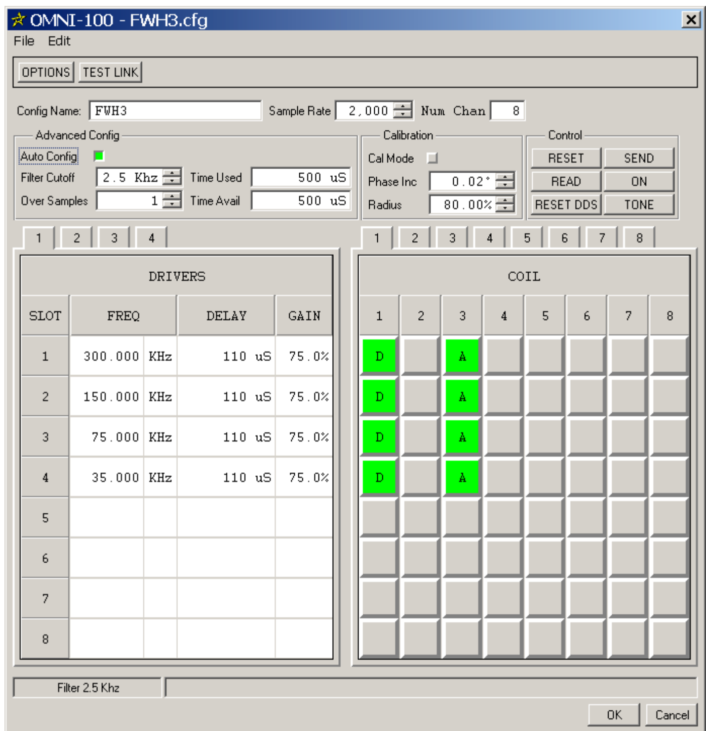


Figure 10-7. OMNI Test Config Dialog

To setup the ***TrackDrive*** (probe-pusher's) configuration:

NOTE:	If you don't have a probe-pusher, you may select ***No Pusher*** in ***EddyAdmin*** and skip the following set of steps at your option.

STEP 1: Click the ***Pusher Config*** button on the ***Test Panel***.

STEP 2: On the ***Ethernet Pusher Config*** dialog, select ***TrackDrive*** in the ***Mode*** list box.

STEP 3: Set the remaining parameters on the ***Ethernet Pusher Config*** dialog per the instructions in the technical manual for the ***TrackDrive***. The ***Ethernet Pusher Config*** dialog shown in Figure 10-9 contains typical settings. Click ***OK***.

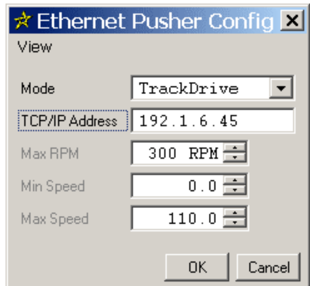


Figure 10-9. Ethernet Pusher Config

STEP 4: On the ***Test Panel***, click or right-click the ***Jog Spd*** button to display the desired jog speed measured in inches per second (ips). Typically set to ***6***.

STEP 5: Click or right-click the ***Push Spd*** button to display the desired push speed (ips). Typically set to ***40***.

STEP 6: Click or right-click the ***Acq Spd*** button to display the desired acquisition speed (ips). Typically set to ***24***.

Before we can acquire any data, we need to complete the ***Summary*** form and open the first cal group as follows:

STEP 1: Click the ***Summary*** button. The ***Summary*** dialog appears.

STEP 2: Ensure that the information contained in the 7 fields within the ***Site*** area of the ***Summary*** dialog (*upper left*) is correct. If not, these fields can only be edited via the ***EddyAdmin*** utility. The entries should match the information entered during the ***Creating a Project*** topic covered earlier.

STEP 3: Set the ***Cal Num*** field to ***1***.

STEP 4: Leave the ***Disk*** field to blank.

STEP 5: Select ***Inlet*** in the ***Leg*** list box.

STEP 6: The ***Material*** field value can be edited by selecting ***Edit | Tables***, while the selection for the ***Material*** field is accomplished in each ***Group Parameters*** dialog in ***MakeComp***.

STEP 7: The ***ID*** and ***OD*** fields can be edited via each ***Group Parameters*** dialog in ***MakeComp***.

STEP 8: Select your ***User ID*** in the first ***Operator ID*** list box.

STEP 9: Select another ***User ID*** in the second ***Operator ID*** list box. This would normally be the probe technician. If no other ***User ID*** is currently available, simply leave the field blank.

STEP 10: In the ***Standards*** area of the ***Summary*** dialog, select *ASME* in the first ***Type*** list box, *Wear Scar* in the second ***Type*** list box, and *N/A* in the third list box.

STEP 11: Select appropriate serial numbers for each corresponding calibration standard in the three ***SN*** list boxes.

> **HINT:** If no standards exist in the list boxes, select ***Edit | Tables*** on the ***Summary*** dialog, click on the ***Standards*** tab, and add the appropriate entries. Enter as many fictitious standards and serial numbers you want for tutorial purposes. Also, an *N/A* entry is handy to have in most tables as well. This modifies *only* the project lookup tables.

STEP 12: In the ***Probe*** area of the ***Summary*** dialog, select ***CA-600-ESH/MF*** in the ***Model*** list box. The corresponding ***Vendor*** field will auto-complete. If no entries exist in this or the list boxes in the remaining steps of this tutorial, follow the *Hint* above to add entries to the applicable tables.

STEP 13: In the ***Probe*** area of the ***Summary*** dialog, select ***4-pin*** in the ***Ext Type*** list box. The corresponding ***Vendor*** field will auto-complete. If no entries exist in the list box, follow the *Hint* above to add entries to the applicable tables.

STEP 14: In the ***Probe*** area of the ***Summary*** dialog, the ***Head Size*** field should have auto-completed when the value for the ***Model*** list box was selected above.

STEP 15: Enter ***85*** in the ***Shaft Length*** field. This is the length (*ft*) of the probe's poly.

STEP 16: Enter ***110*** in the ***Ext Length*** field. This is the total length (*ft*) of any probe extension cables.

STEP 17: Enter ***N/A*** in the ***Slip SN*** (*Slip-ring assembly serial number*) field.

STEP 18: Enter ***1234-0603*** in the probe ***Head SN*** field. This would normally be the serial number of the inspection probe head of a detachable-head probe.

STEP 19: Enter ***1302-0603*** in the probe ***Shaft SN*** field. This can be either the serial number of the inspection probe poly of a detachable-head probe or fixed-head probe.

STEP 20: Enter ***1235-0603*** in the probe ***Ref Head SN*** field. This would normally be the serial number of the reference probe head of a detachable-head probe.

STEP 21: Enter ***1303-0603*** in the probe ***Ref Shaft SN*** field. This can be either the serial number of the reference probe poly of a detachable-head probe or fixed-head probe.

STEP 22: The upper section of the ***Test Config*** area of the ***Summary*** dialog simply displays the details of the currently set inspection frequencies and channels of the tester configuration we set previously in the ***Tester Config*** dialog. None of these settings can be manually changed in the ***Summary*** dialog. Proceed to the next step.

STEP 23: The ***Config*** field should automatically show ***FWH3***. This is the name of the tester configuration we set previously.

STEP 24: The ***Rate*** field should automatically show ***2000***. This is the sample rate we previously set in the tester configuration.

STEP 25: The ***Speed*** field should automatically show ***24.00***. This is the acquisition probe-pusher speed we previously set in the ***Test Panel***.

STEP 26: The ***#CH*** field should automatically show ***8***. This is the number of channels currently set in the ***Tester Config*** dialog.

STEP 27: The ***OFF*** field should initially show ***0***. Once a file is played in from the ***TLIST***, it will show the data offset for that file.

STEP 28: The ***RPM*** field should be blank. If an RPC (*rotating pancake coil*) inspection was being setup, this field would show the ***RPC Spd*** (*rotation speed*) currently set in the ***Test Panel***.

STEP 29: Select ***Pull*** in the ***DIR*** list box. This is the probe direction during acquisition, which is typically on the ***Pull***.

STEP 30: The ***Source*** field should be currently ***blank***, while the ***Samples*** field shows a value of ***50,000***. For each file loaded from the ***TLIST***, the ***Source*** and ***Samples*** fields will show the data file format and total samples of the file, respectively. For example, ***CoreStar*** may be shown in the **Source** field and ***51,357*** in the ***Samples*** field for a given data file.

STEP 31: Enter ***CIC-300 Rev 3*** in the ***Procedure*** field.

STEP 32: Select ***OMNI-100/R*** in the ***Tester*** list box.

STEP 33: Select ***0111-0503*** in the ***SN*** list box that corresponds to the ***Tester***.

STEP 34: Select ***TD-100*** in the ***Pusher*** list box.

STEP 35: Select ***0051-0703*** in the ***SN*** list box that corresponds to the ***Pusher***.

STEP 36: Select ***N/A*** in both the ***Fixture*** and the corresponding ***SN*** list boxes.

STEP 37: Enter the following text in the comment area (*large white block*) at the bottom of the ***Summary*** dialog: ***EddyVISION32 v5.0 Tutorial***.

STEP 38: Ensure that the ***Cal Num*** is set to ***1***. The ***Summary*** dialog should now look like the completed one shown in Figure 10-11 below.

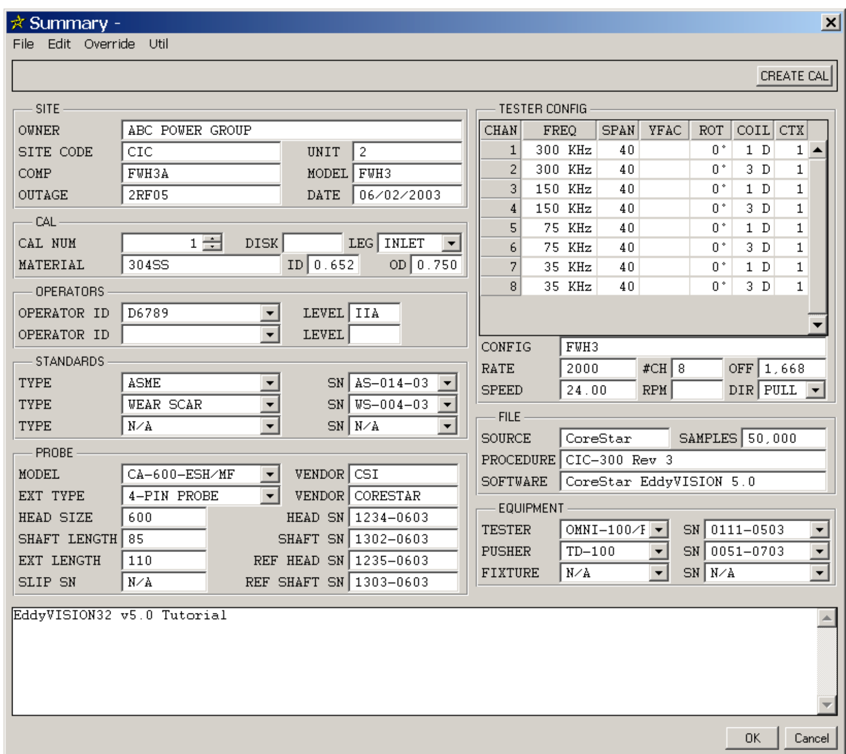


Figure 10-11. Summary Dialog

STEP 39: *Shift+click* the ***Create Cal*** button in the upper right of the ***Summary*** dialog. The ***Summary*** dialog disappears from the screen and cal group #1 is now open and ready to acquire data.

We're now ready to pull a preliminary calibration standard that will be temporarily stored in the computer's memory. We'll setup the signal phase angles (*rotations*), signal spans (*sizes*), update the ***Summary***, and then store our setup.

NOTE:	The data used in the following graphics are typical calibration standard runs. The inspection frequencies may not match the text in the steps.

STEP 1: With the reference probe in a reference section of tubing, place the inspection probe into a defect-free section of the ***ASME*** standard.

STEP 2: Click the ***Tester Off*** button. The button turns green, the label changes to ***Tester On***, and the long strip charts begin to scroll.

STEP 3: Press the **Space** bar to balance or *null*.

STEP 4: Position the probe at the far end of the ***ASME*** standard and smoothly pull the probe through the standard, then click the ***Tester On*** button to turn off the tester. A temporary data file of the standard now exists in the computer's memory.

STEP 5: Locate the ***100% TWH*** (*through-wall hole*) signal in the long strip chart.

STEP 6: Center and *trim down* on the ***100% TWH*** signal in the expanded strip chart so that it's clearly visible in the Lissajous window. Balance as required.

STEP 7: Middle-click in the ***Span*** box of any Lissajous to auto-fit the signal within the Lissajous.

NOTE:	The ***Span*** box is located at the upper left corner of any Lissajous.

STEP 8: Ensure that ***CH 1 300KHz C1*** is displayed in the title bar of the left Lissajous. If not, click the title bar of the Lissajous until ***CH 1*** is displayed.

STEP 9: Left-click once in the left Lissajous to measure the ***100% TWH*** signal ***Vpp*** (*Volts peak-to-peak*). Red balls will appear at either end of the signal. Volt and phase angle values are displayed in the measurement grid located at the lower left of the Lissajous.

NOTE:	Left-clicking twice in a Lissajous measures a signal ***Vmr*** (*Volts Max-Rate*). Green measurement balls appear in addition to the red ***Vpp*** ones.

STEP 10: Shift+left-click once in the ***Rotation*** box. The Lissajous displays 3 red vectors at 0°, 90°, and 40°.

NOTE:	The ***Rotation*** box is located at the upper right corner of any Lissajous. It contains a value with the degree symbol (°).

STEP 11: Left-click+drag in the Lissajous display and observe the phase angle in the measurement grid. Stop at 40°.

STEP 12: Repeat the 2 steps above until the ***100% TWH*** signal measures a phase angle of 40° for channels 1, 3, 5, and 7 (*all differential channels*) as shown in Figure 10-13 below.

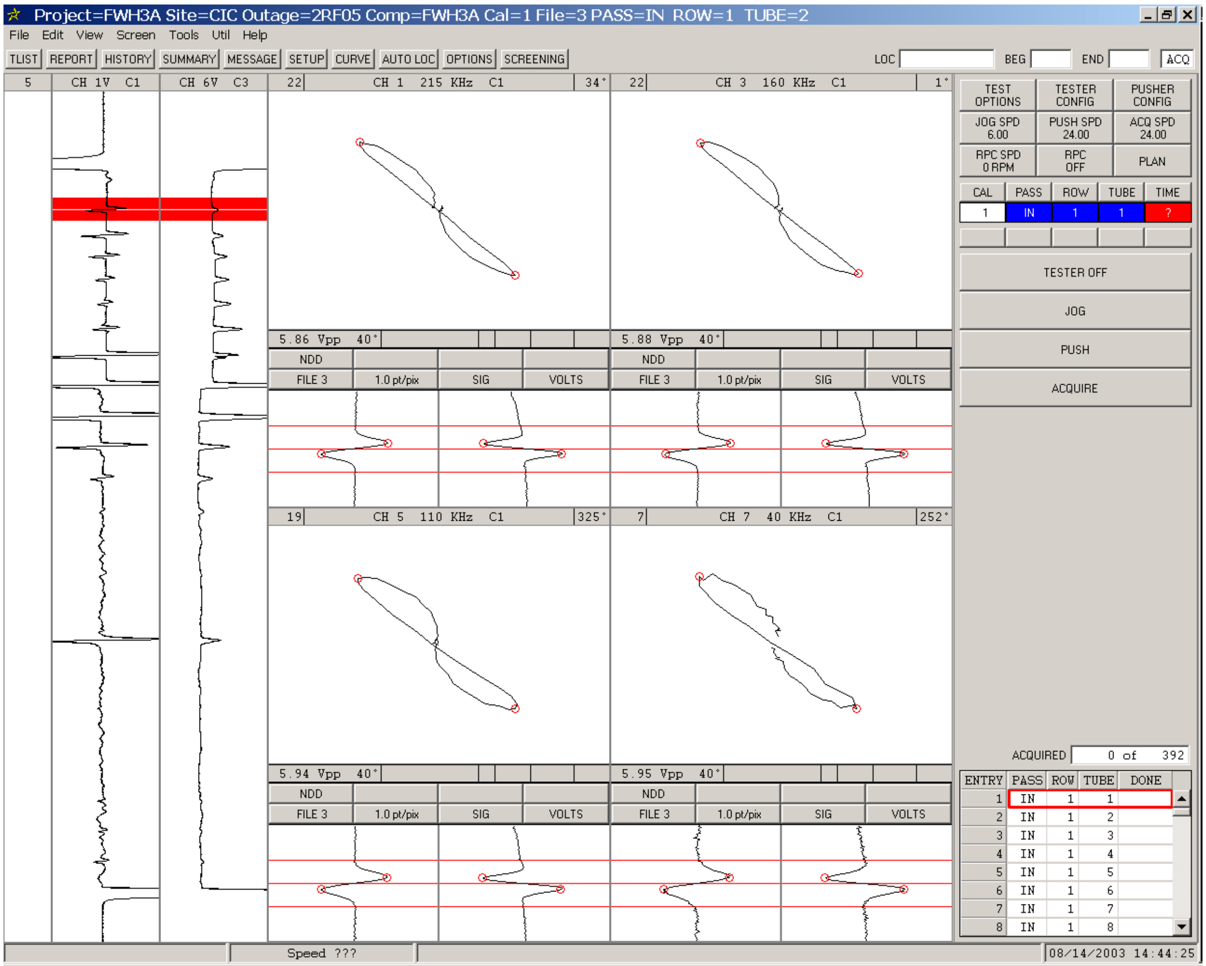


Figure 10-13. Acquisition Screen - Differential Channel Setup

TIP:	Be sure to re-measure the ***100% TWH*** signal ***Vpp*** in each Lissajous before setting the signal rotation.

STEP 13: Select ***CH 4 150KHz C3*** for the left Lissajous.

STEP 14: Locate the ***10% OD Groove*** signal in the long strip chart.

STEP 15: Center and *trim down* on the ***10% OD Groove*** signal in the expanded strip chart so that it's clearly visible in the Lissajous window. Balance as required.

STEP 16: Left- or right-click in the ***Span*** box of any Lissajous to adjust the signal size to fit within the Lissajous.

STEP 17: Shift+left-click once in the ***Rotation*** box. The Lissajous displays 3 red vectors at 0°, 90°, and 40°.

STEP 18: Left-click+drag in the Lissajous display and set the ***10% OD Groove*** signal vertical (*going up from null*). This is a preliminary setting and the rotation accuracy is not important at this time.

STEP 19: Repeat the 3 steps above until the ***10% OD Groove*** signal is vertical for channels 2, 4, 6, and 8 (*all absolute channels*).

STEP 20: Locate a typical probe motion (*lift-off*) signal in the calibration standard on ***CH 2 300KHz C3***. Open the expanded window so that the lift-off is clearly visible.

STEP 21: Repeat the applicable preceding steps, but this time set the rotation of the lift-off signal horizontal (*flat*) for channels 2, 4, 6, and 8. The final result for all absolute channels should be *lift-off horizontal with flaws going up* as shown in Figure 10-15.

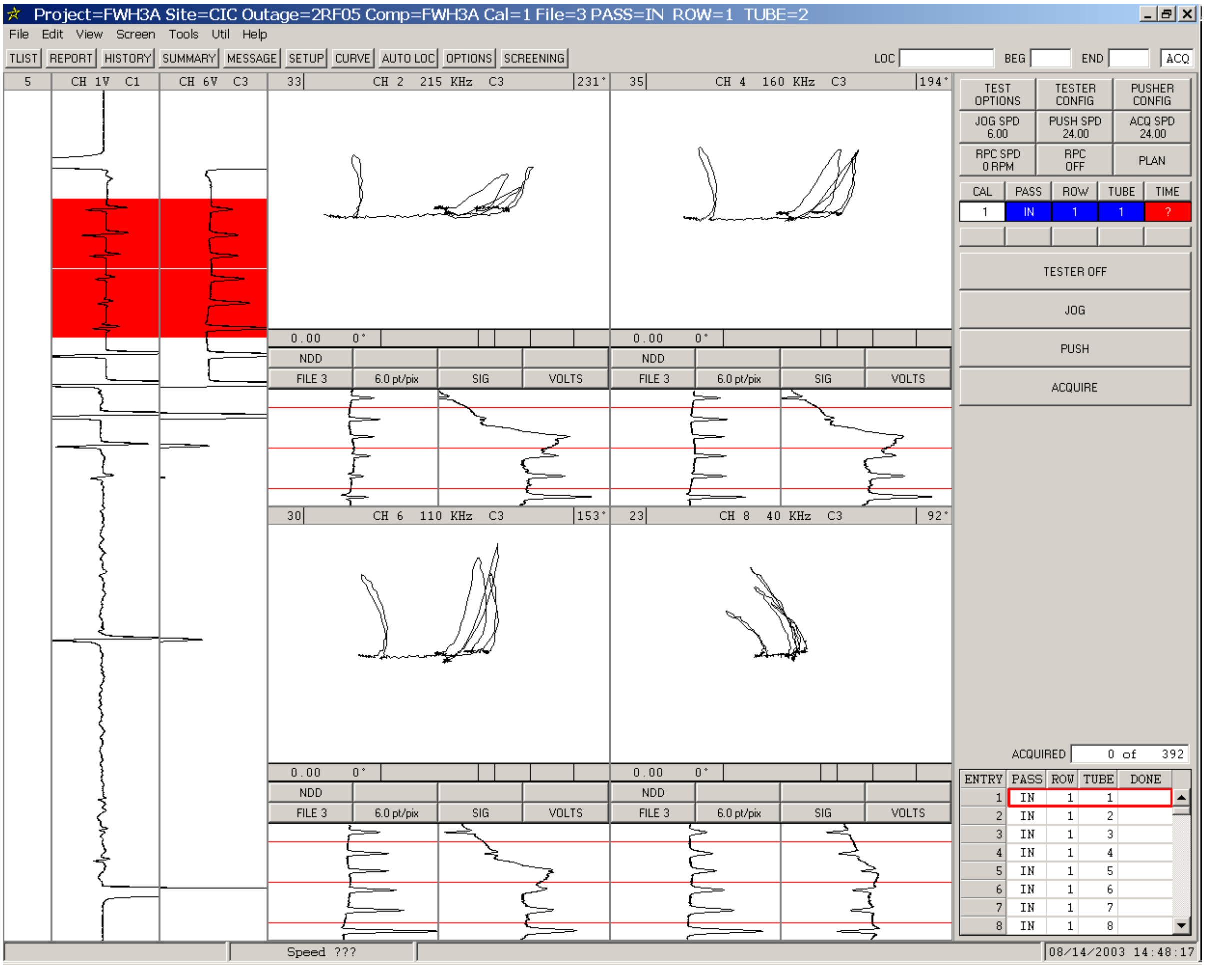


Figure 10-15. Acquisition Screen - Absolute Channel Setup

STEP 22: With the ***100% TWH*** signal clearly visible in a Lissajous for ***CH 1 300KHz C1***, Shift+left-click once in the ***Span*** box of the Lissajous. A red grid pattern appears in the Lissajous.

STEP 23: Left-click+drag in the Lissajous to obtain 6 vertical divisions for the ***100% TWH*** signal.

STEP 24: Repeat the 2 steps above for the remaining differential channels (*3, 5, & 7*). The signal ***Span*** or *size* should appear similar to Figure 10-17.

Note: The voltage grid shown in Figure 10-17 is for tutorial purposes only.

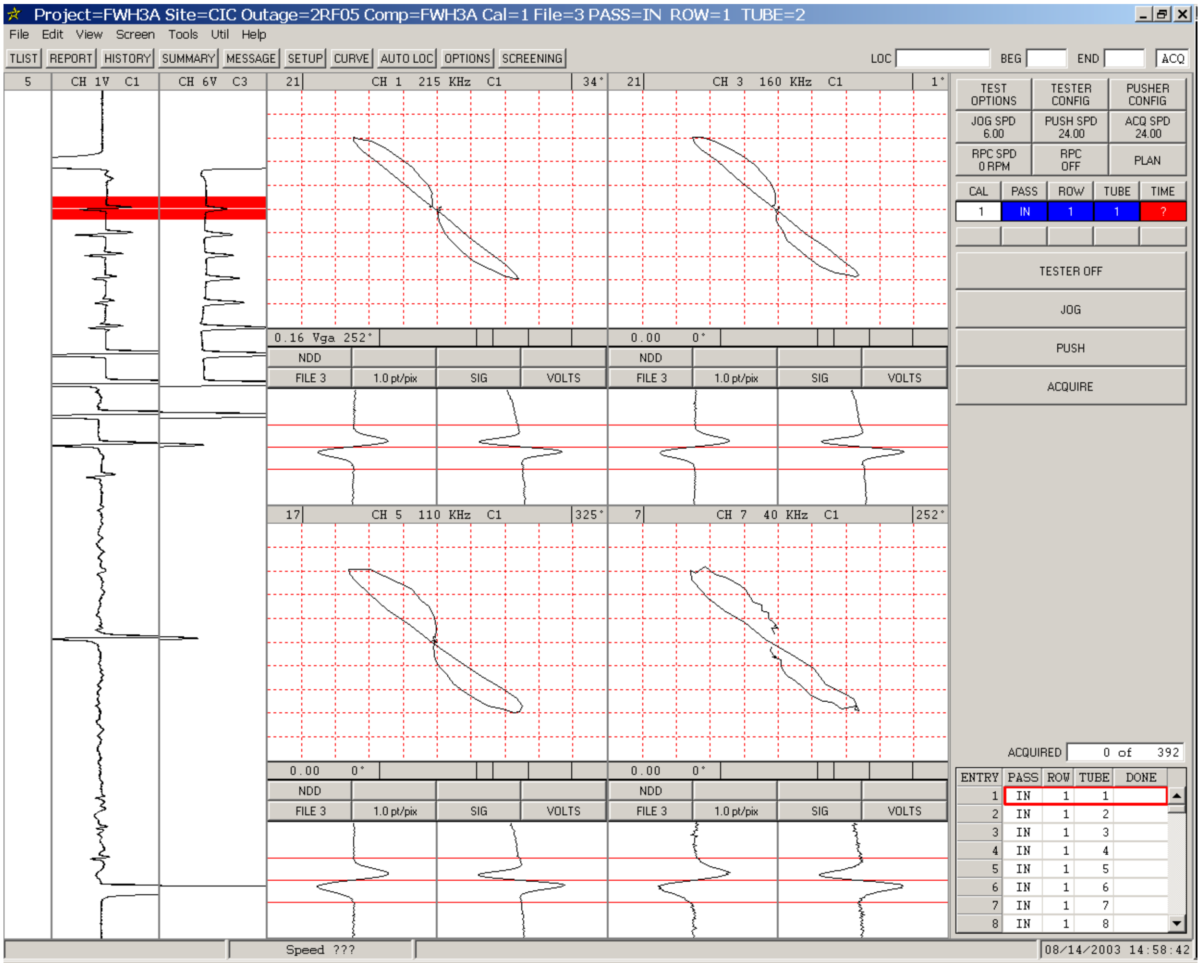


Figure 10-17. Acquisition Screen - Setting Differential Span Levels

STEP 25: With the ***10% OD Groove*** signal clearly visible in a Lissajous for ***CH 2 300KHz C3***, Shift+left-click once in the ***Span*** box of the Lissajous.

STEP 26: Left-click+drag in the Lissajous to obtain 4 divisions for the ***10% OD Groove*** signal.

STEP 27: Repeat the 2 steps above for the remaining absolute channels (*4, 6, & 8*). The signal ***Span*** or *size* should appear similar to Figure 10-19.

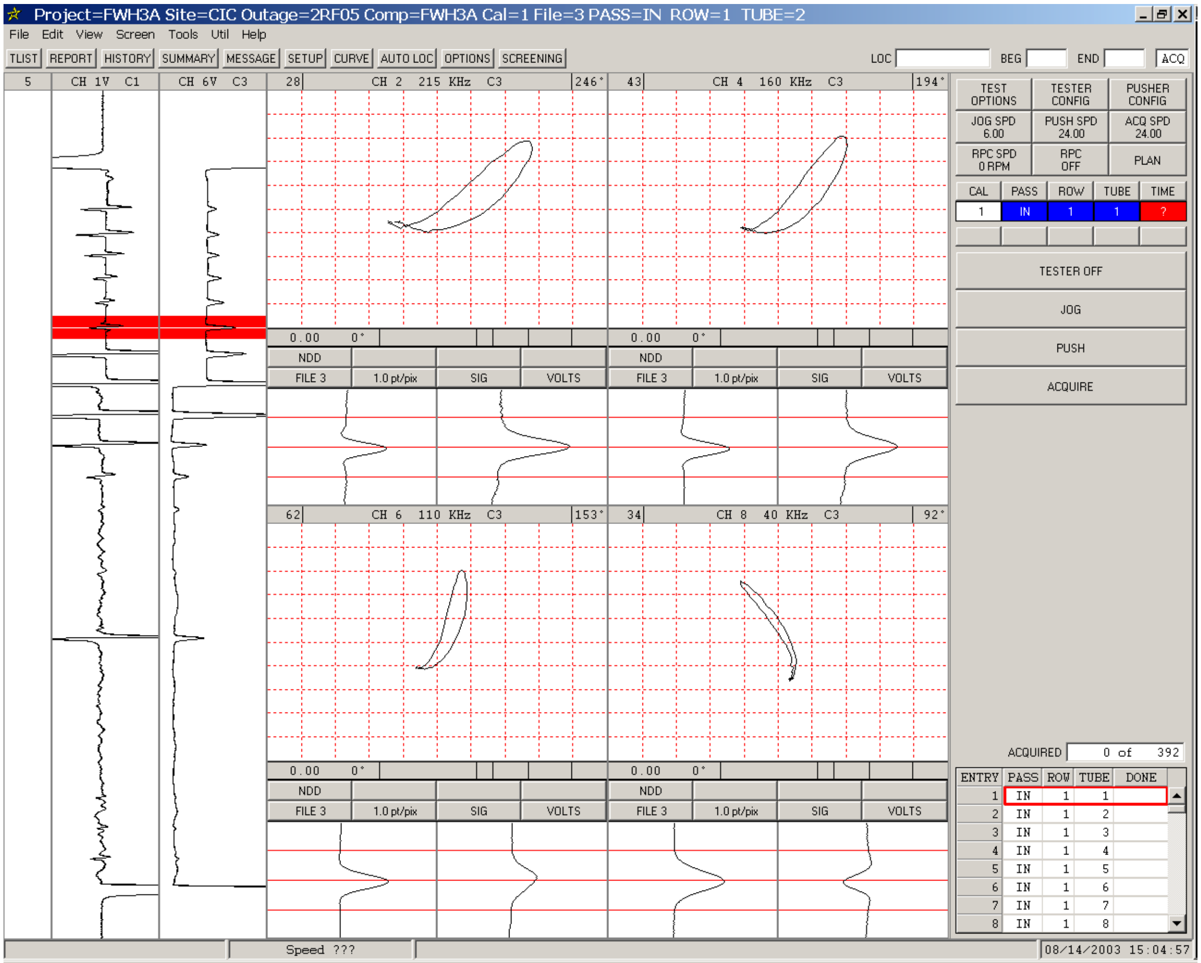


Figure 10-19. Acquisition Screen - Setting Absolute Span Levels

STEP 28: Click the title bars of the long strip charts so that ***CH 1V*** appears in the left, while ***CH 6V*** appears in the right.

STEP 29: Click the title bars of the Lissajous' so that ***CH 3*** appears in the left one, while ***CH 6*** appears in the right one.

STEP 30: Open the cursors in the expanded chart to about 50%.

STEP 31: Click the ***Summary*** button. The ***Summary*** dialog appears.

STEP 32: Select ***Util | Update Tester Config***.

STEP 33: Click ***OK***.

STEP 34: Click the ***Setup*** button along the top of the acquisition screen. The ***Setup*** dialog appears.

STEP 35: As shown in Figure 10-21, select ***Cal | Store Cal Setup***. A message appears in the ***Status Bar*** at the bottom of the main acquisition screen showing the name of and where the ***Setup*** file was saved.

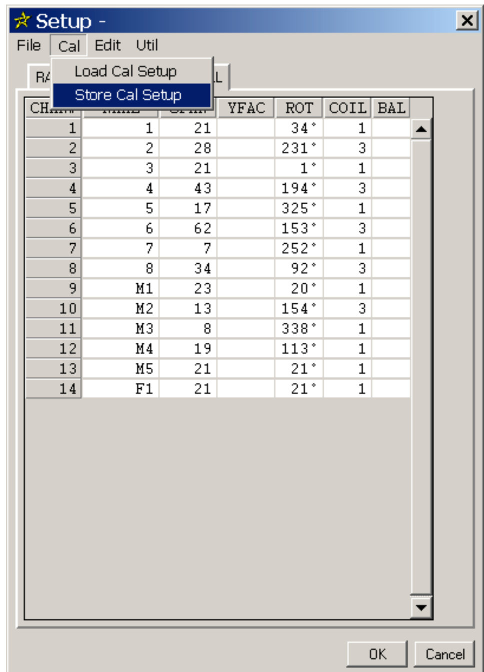


Figure 10-21. Setup Dialog - Storing the Cal Setup

STEP 36: Click ***OK*** on the ***Setup*** dialog.

NOTE:	The ***Setup*** file stores all the parameters we set during the preceding steps. If the ***Setup*** is inadvertently changed during acquisition, it can easily be reloaded via the ***Setup*** dialog by selecting ***Cal | Load Cal Setup***.

Now that we are calibrated, we're ready to record the calibration standards. A team of 2 operators is recommended for the following steps.

To record calibration runs:

STEP 1: Click the ***Message*** button along the top of the acquisition screen.

STEP 2: In the ***Message*** dialog, enter the following text: ***Start cal #1. ASME cal runs to follow***.

STEP 3: Click ***OK*** on the ***Message*** dialog to *write* the **`msg`** file to the cal group.

STEP 4: *Click and hold* the ***Jog*** button (*below the Tester button*) on the ***Test Panel***. The ***Jog*** button turns green and displays ***Jogging Forward***. Continue jogging out the probe until the probe head clears the conduit by about 3 feet, and then release the mouse button to stop the probe.

TIP: *Right-click and hold* the ***Jog*** button to jog the probe in the reverse direction.

STEP 5: Turn ***Tester On***.

STEP 6: Insert the probe into the ***ASME*** calibration standard so that the probe is located in a defect-free area, and then balance.

STEP 7: Manually push the probe all the way out the far end of the ***ASME*** standard so that the probe's coils are just outside the mouth of the tube.

STEP 8: Shift+click the ***Acquire*** button on the test panel. When the probe exits the near tube end, click the ***Acquire*** button again.

NOTES: When the ***Acquire*** button was Shift+clicked in the previous step, the following occurred:

- A tube ID of ***999 999*** was automatically encoded.
- The calibration timer was started per the ***Test Options*** dialog settings;
- The ***Acquire*** button turned green and displayed ***Storing***.
- The mouse pointer became a *cross-hair* and was *locked* within the Limits of the ***Acquire*** button.
- The probe-pusher pulled the probe at the specified ***ACQ SPD***.
- The cal run was stored as a data file named **`c:\project\FWH3A\ecdata\000001.cal\0000_999_999_000.DAT`**

STEP 9: Repeat the applicable preceding steps 2 more times for a total of 3 cal runs for the ***ASME*** standard. Three cal runs per standard is common practice.

STEP 10: Turn ***Tester Off***.

STEP 11: Add the following message: ***Wear Scar cal runs to follow***.

STEP 12: Repeat the applicable preceding steps to record 3 cal runs of the ***Wear Scar*** standard.

STEP 13: Turn ***Tester Off***.

We're now ready to start acquiring data per our inspection plan.

STEP 1: Turn ***Tester On***.

STEP 2: ***Jog*** out about 2 feet of the probe.

STEP 3: Insert the probe into the first tube on the inspection.

> **NOTE:** For tutorial purposes, use the calibration standard repeatedly for the next few steps to simulate acquiring data for several different tubes.

STEP 4: Click the ***Push*** button. The button turns green and changes to ***Pushing***. The probe-pusher begins to push the probe down the tube at the ***PUSH SPD***.

> **TIP:** Right-click the ***Push*** button to make the probe-pusher pull the probe at the ***ACQ SPD***. The button turns green and changes to ***Pulling.***

STEP 5: Watch the Lissajous and balance to keep the display within the Lissajous.

STEP 6: When the probe nears the far end of the tube, click the ***Push*** button to stop the probe-pusher, balance in a free-span area, and then ***Jog*** to the end of the tube. Stop jogging when you see the *air signal*.

STEP 7: If necessary, click tube number *IN-1-1* in the ***Plan*** list at the bottom of the ***Test Panel*** as shown in Figure 10-23. This tube should now appear encoded in the ***Test Panel*** with a blue background.

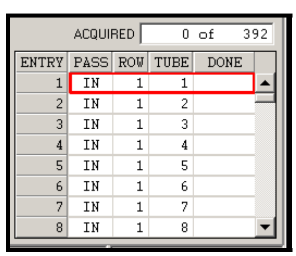


Figure 10-23. Plan List at Bottom of Test Panel

HINT:	A blue background means that the tube has not been acquired for the current plan. A red background indicates the tube has been acquired.

STEP 8: Click the ***Acquire*** button. The probe-pusher pulls the probe at ***ACQ SPD*** and the system begins to record data. You may balance during acquisition, as it has no effect on the recorded data.

STEP 9: When the probe exits the tube (air signal) at the near end, click the ***Acquire*** button to stop recording data and stop the probe-pusher. The next tube on the ***Plan*** list will automatically be selected and encoded as shown in Figure 10-25.

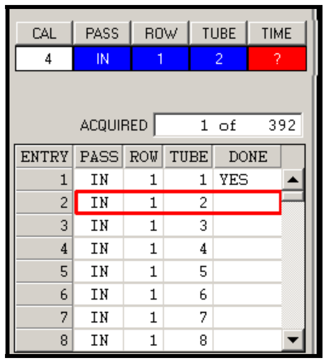


Figure 10-25. Plan List and Updated Tube Encode for Next Tube (Split Screen Capture)

STEP 10: Repeat this sequence of steps for the remaining tubes on the inspection plan or until the cal group reaches its maximum size. For tutorial purposes, let's acquire the first 5 tubes on the ***Plan***, <u>then</u> continue with the next topic.

Sometimes during acquisition, circumstances require the operator to add a message to the ***TLIST***. Common messages may be in regard to plugged, restricted (*partial length*), and obstructed (*can't insert the probe in the tube*). Version 5.0 has a new timesaving feature, which automates entering the most common messages.

In our test plan, let's say that tube *IN-1-6* is plugged and *IN-1-7* is restricted (*You should've already acquired the first 5 tubes per the preceding steps*). To use the new messaging feature:

STEP 1: If necessary, turn ***Tester Off***.

STEP 2: Select ***Edit* | *Defect List***. The ***Defect List*** dialog appears.

STEP 3: Check to see if both ***PLG*** and ***RST*** codes exist in the ***Defect List***. If so, skip to step #7, else continue with the next step.

STEP 4: Click the *** (asterisk)*** above the scroll bar. A new row appears at the bottom of the ***Defect List*** dialog.

STEP 5: Click in the ***Name*** field of the new row and type ***PLG***.

STEP 6: Click in the optional ***Description*** field of the ***PLG*** entry and type ***Plugged Tube***.

STEP 7: Click in the ***Yes/No*** fields for the ***PLG*** entry so that only ***Code***, ***Pop***, and ***Fly*** display ***Yes***. ***Fly*** is the main one to have enabled for auto-messaging.

STEP 8: Repeat the preceding steps as applicable to add an ***RST*** code. Enter ***Restricted*** for the description. Set the ***Yes/No*** fields identical to the ***PLG*** entry.

TIP: To delete a row in most *table-like* dialogs, click in any field of the row to be deleted and press ***Shift+Delete*** on the keyboard.

STEP 9: Click ***OK*** on the ***Defect List*** dialog to save the changes. Note that on the acquisition screen, these codes now appear on the first 2 buttons just below the tube encode fields in the ***Test Panel*** as shown in Figure 10-27. Up to 5 of these ***Fly*** buttons may be assigned.

STEP 10: Ensure that tube *IN-1-6* is selected in the ***Plan*** list and encoded.

STEP 11: Shift+click the ***PLG Fly*** button. The tester may be on or off for this step.

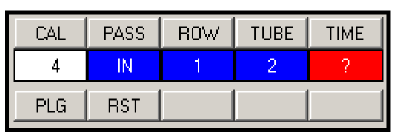


Figure 10-27. Fly Buttons Used for Quick Messages

NOTES: When the ***PLG Fly*** was Shift+clicked in the previous step, the following occurred:

- The tube was marked as ***Done*** in the ***Plan***.
- The next tube on the ***Plan*** was encoded.
- A data file containing the message ***PLG PASS=IN ROW=1 TUBE=6*** was added to the ***TLIST***. The file contains no raw data.
- The message was displayed in the status bar at the lower left of the main acquisition screen.

STEP 12: Ensure that tube *IN-1-7* is selected in the ***Plan*** list and encoded.

STEP 13: Shift+click the ***RST Fly*** button.

STEP 14: Since this tube is partially restricted, we need to acquire the available data for this tube. Double-click tube *IN-1-7* in the ***Plan*** list.

STEP 15: Click ***Yes*** in the message box that appears to change its status from ***Done*** to ***Not Done***.

STEP 16: Acquire the available data for tube *IN-1-7*. The ***TLIST*** now shows 2 entries for tube *IN-1-7*. The first notifies the analyst that *IN-1-7* is restricted, while the second entry contains the available eddy current data.

Sometimes during acquisition, it may be necessary to encode a tube, which is not on the inspection plan.

To manually encode a tube:

STEP 1: Click in the ***Pass*** (*Sec*), ***Row***, and ***Tube*** (*Col*) encode fields until the desired tube identification is displayed. A white background indicates that a tube is not on the inspection plan.

STEP 2: Acquire the data like any other tube. The tube will be appended to the plan.

During acquisition, it may be necessary to load or *play in* a tube file for review such as a calibration run.

To load a tube for review:

STEP 1: Turn ***Tester Off***.

STEP 2: Click the ***TLIST*** button along the top of the acquisition screen. The ***TLIST*** dialog appears.

STEP 3: Double-click any entry in the ***TLIST***. The software loads the file and displays it on the screen.

Let's end cal group #1 by adding a message to the ***TLIST***, recording *end-of-cal* standard runs, and closing the cal group. Afterwards, we'll start cal group #2.

To properly end a cal group:

STEP 1: Turn ***Tester Off***.

STEP 2: Click the ***Message*** button. The ***Message*** dialog appears.

STEP 3: Enter the following text: ***End of cal#1. ASME to follow***.

STEP 4: Click ***OK*** on the ***Message*** dialog to add the message to the ***TLIST***.

STEP 5: Acquire 3 cal runs of the ***ASME*** standard. Be sure to use ***Shift+Acquire*** to automatically encode ***999 999***.

STEP 6: Turn ***Tester Off***.

STEP 7: Click the ***Message*** button. The ***Message*** dialog appears.

STEP 8: Enter the following text: ***Wear Scar to follow***.

STEP 9: Click ***OK*** on the ***Message*** dialog to add the message to the ***TLIST***.

STEP 10: Acquire 3 cal runs of the ***Wear Scar*** standard. Be sure to use ***Shift+Acquire*** to automatically encode ***999 999***.

STEP 11: Turn ***Tester Off***.

STEP 12: Click the ***Summary*** button. The ***Summary*** dialog appears.

STEP 13: Shift+click the ***END CAL*** button in the upper right of the ***Summary*** dialog to close the cal. The button changes to ***CREATE CAL***.

STEP 14: Modify the ***Summary*** as required (*new probe SNs, operators' IDs, etc*.) for cal group #2.

STEP 15: Increment the ***Cal Num*** field to ***2***.

STEP 16: Shift+click the ***CREATE CAL*** button in the upper right of the ***Summary*** dialog to open cal group #2.

STEP 17: Continue acquiring data referring to all the steps in this topic until all tubes on the inspection plan have been recorded.

NOTE: For tutorial purposes, we are done with this topic. Proceed to the next topic.

11. Copying Cal Groups to Transfer Media

OK, so you're finished with cal group #1 and need to transfer it to say a 250Mb ZIP disk in order to take a copy to the data room where it will be copied to another PC and analyzed. At your option, you may simply use Windows Explorer to copy the directory for cal group #1 from the ***FWH3A*** project to the ZIP disk. Potential problems using this method is that cal group #1 may exist for more than one component during a given inspection and it's possible to lose track which is which on more than one ZIP disk. <u>If you use this method, be sure to label the ZIP disks very carefully</u>. Alternatively, it's recommended that you use the ***Cal Admin*** utility accessible in the ***TLIST***. The advantage is that this utility creates the project path on the ZIP disk (*or other media*) so that there's no mistake identifying the component the data belongs to.

To copy cal group #1 from the currently opened tutorial project to a *target disk*:

STEP 1: If not already running, start ***CoreStar***.

STEP 2: Open the ***FWH3A*** tutorial project.

STEP 3: Select the ***Test*** screen.

STEP 4: Click the ***TLIST*** button.

STEP 5: Select ***Cal | Admin***.

STEP 6: Click the entry for cal group #1.

STEP 7: In the ***Target Drive*** list box, select ***x:***, where ***x*** is the letter of the device that contains the transfer media on your computer, i.e., ZIP, JAZ, etc. The ***Cal Admin*** dialog should appear similar to Figure 11-1. The ***Target Drive*** selected in this example is ***y:***.

STEP 8: Click the ***Archive*** button. (*Click the **Archive** button at anytime to stop.*)

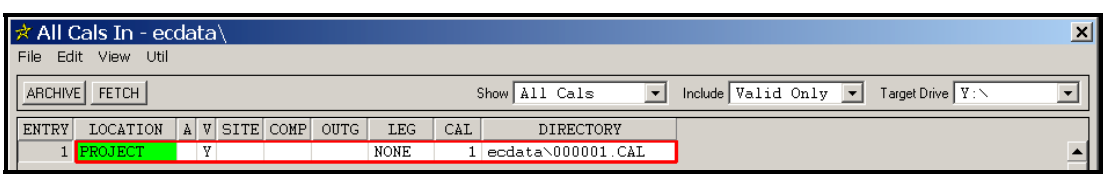


Figure 11-1. Cal Admin Dialog - Archiving/Copying a Cal Group

NOTES:	When the ***Archive*** button was clicked in the previous step, the following occurred: • The ***Archive*** button turned green. • The cursor was locked inside the ***Archive*** button. • The copying process (*from computer hard drive to target drive*) began. When the copying process completed, the ***Location*** field for cal group #1 turned to ***Both*** on a white background as shown in Figure 11-3.

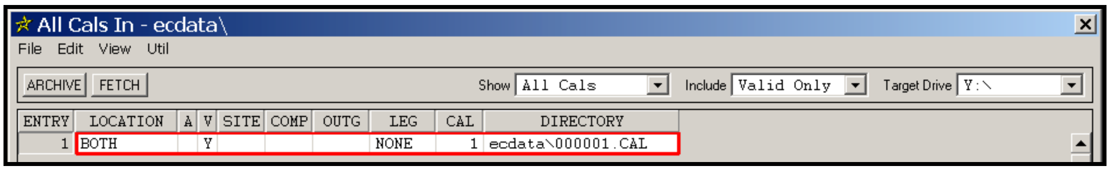


Figure 11-3. Cal Admin Dialog - After Archiving/Copying a Cal Group

STEP 9: Click ***OK*** to close the ***Cal Admin*** dialog.

12. Copying Cal Groups from Transfer Media

Before copying a cal group to another computer for analysis using the ***Cal | Admin*** utility, the same project must exist on the analysis computer as the cal group to be copied.

To copy a cal group(s) from transfer media for the tutorial project to an analysis computer:

STEP 1: If not already running, start ***CoreStar*** on the analysis computer.

STEP 2: Open the ***FWH3A*** tutorial project.

STEP 3: Select the ***Main Analysis*** screen.

STEP 4: Click the ***TLIST*** button.

STEP 5: Select ***Cal | Admin***.

STEP 6: In the ***Target Drive*** list box, select ***x:***, where ***x*** is the letter of the device that contains the transfer media on your computer, i.e., ZIP, JAZ, etc. An entry for cal group #1 should appear with ***Target*** on a yellow background in the ***Location*** field.

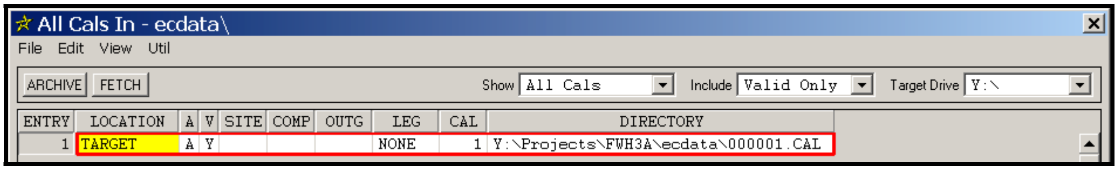


Figure 12-1. Cal Admin Dialog - Fetching a Cal Group

STEP 7: Click the entry for cal group #1. The ***Cal Admin*** dialog should appear similar to Figure 12-1.

STEP 8: Click the ***Fetch*** button. (*Click the* ***Fetch*** *button at anytime to stop*.)

NOTES: When the ***Fetch*** button was clicked in the previous step, the following occurred:

- The ***Fetch*** button turned green.
- The cursor was locked inside the ***Fetch*** button.
- The copying process (*from media to computer hard drive*) began.

When the copying process completed, the ***Location*** field for cal group #1 turned to ***Both*** on a white background as shown in Figure 12-3. This means that cal group #1 exists on the ***Target*** (*source*) drive as well as the local hard drive.

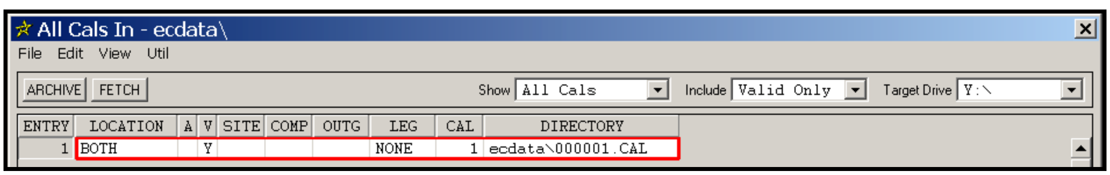


Figure 12-3. Cal Admin Dialog - After Fetching a Cal Group

STEP 9: Click ***OK*** to close the ***Cal | Admin*** dialog.

13. Creating a History File

During data analysis, history files are very useful in determining if previous eddy current calls have changed. The ***DBMS*** module of ***EddyVISION32*** allows you to create a custom history file from previous report entries that reside in a given project's database. The history file can then be conveniently referenced manually by the analyst or automatically by the analysis software.

IMPORTANT: ***For the remainder of this tutorial***, you will be working with a sample project named ***LPFH16A***. We will assume the project is in the following directory:

`c:\projects\LPFH16A`

To create a ***History*** file for use during data analysis:

STEP 1: Start ***DBMS***.

STEP 2: Open the newly copied project, which should be located in: ***`c:\projects\LPFH16A`***.

STEP 3: Select ***Database*** | ***Open Database*** and double-click ***`LPFH16A.mdb`*** in the ***Open*** dialog.

TIP: The currently opened database is always displayed in the site information area in the upper left of the ***DBMS*** screen just below the buttons.

STEP 4: Select ***Database*** | ***Create History.*** The ***History*** dialog appears.

STEP 5: Select ***Database*** | ***Fetch History.***

STEP 6: In the ***Outage Filter*** section, click the ***Keep Latest Selected Outage*** check box (*green*).

STEP 7: Click the ***Use*** column header to turn all the ***Use*** fields to ***Yes*** for all outages shown.

STEP 8: In the ***Defect Filter*** section, click the ***Use*** column header to turn all ***Use*** fields to ***No*** for all the defect codes shown.

STEP 9: Click the ***Use*** fields for ***%***, ***ADS***, and ***DNT*** so that only these 3 fields show ***Yes***. The ***Keep*** box in the upper right of the ***History*** dialog shows that 61 records of the total available in the database will be used for the history file. Compare your ***History*** dialog to Figure 13-1.

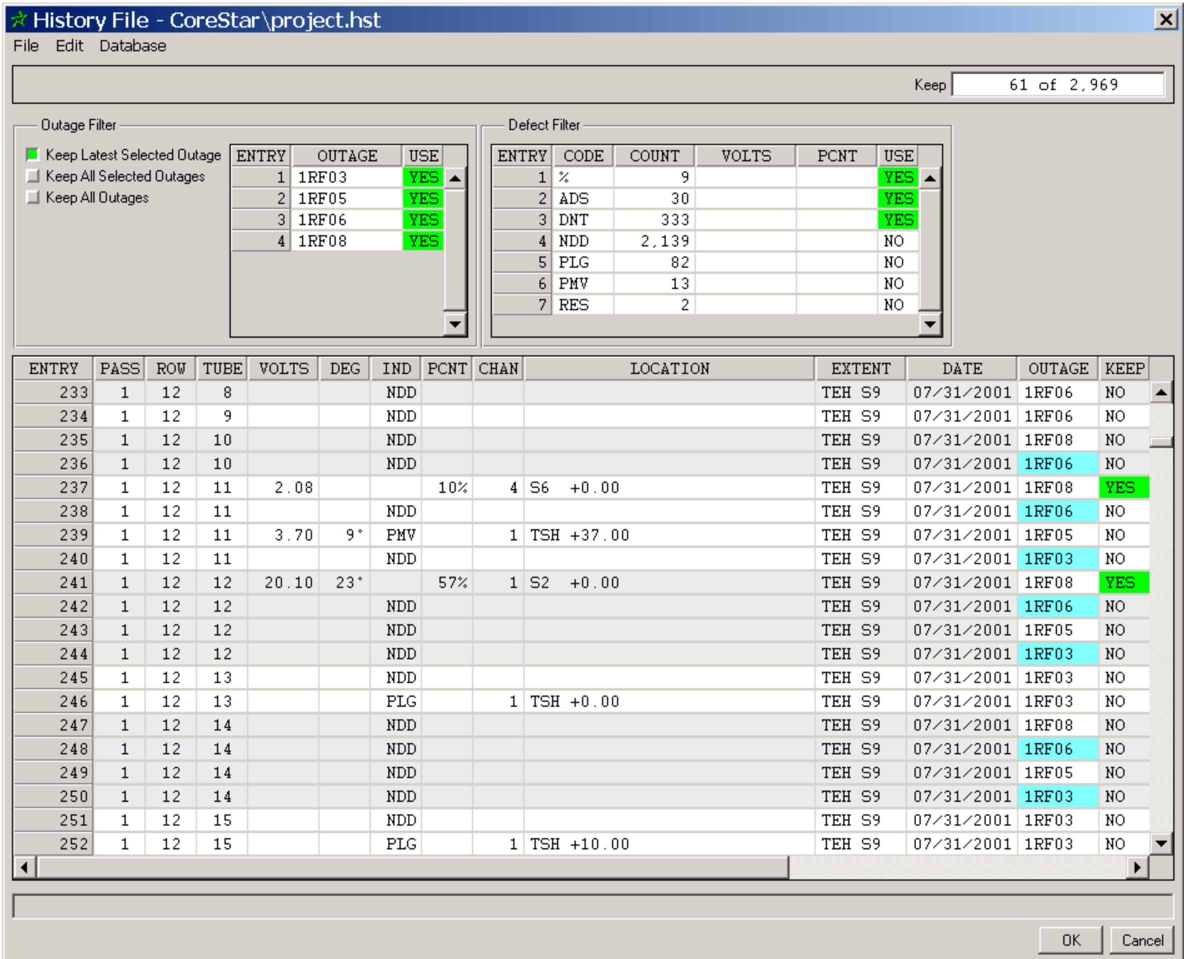


Figure 13-1. History Dialog - Creating a History File in DBMS

NOTE: Clicking in the ***Volts*** and/or ***Pcnt*** fields for a given defect code will provide additional filtering to *narrow down* the makeup of the history file.

STEP 10: Select ***File | Save As***.

STEP 11: Click ***Save*** to accept the recommended file name of:

LPFH16A\history\CoreStar\project.hst.

STEP 12: Click ***OK*** on the ***History*** dialog.

NOTES:
- The ***History*** file may be *tweaked* at anytime during an inspection. Just be sure to copy the *tweaked* ***History*** file to all EddyVISION32 workstations for the specific project.
- The preceding steps show how to make a typical history file; however, what you include and/or exclude in a history is up to you.
- You can make multiple history files if desired. One for bobbin calls last outage, another for RPC calls, etc. Just give them different filenames.

14. Analyzing Eddy Current Data (*excludes AutoVISION*)

This topic will provide the necessary steps to follow using the manual analysis software to perform typical eddy current bobbin data analysis. It is not intended to instruct the user in the eddy current method, but rather simply how to setup and use the analysis software. Be sure to use the ***LPFH16A*** project for following steps.

The first thing we need to do is start the analysis software module as follows:

STEP 1: Start ***CoreStar*** by double-clicking the "yellow star" icon. The ***User Logon*** dialog appears. This software module provides both data analysis and acquisition functions.

STEP 2: In the ***User ID*** list box, select ***D6789*** (*John Doe*) or ***S1234*** (*Jane Smith*) as appropriate. The remaining fields of the ***User Logon*** dialog will auto-complete.

STEP 3: Click ***OK***. The analysis/acquisition software screen appears.

STEP 4: Select ***Screen* | *ANA-3L/3S*** or simply press ***F1*** on the keyboard.

TIPS:
- You may design up to 16 different screens by selecting ***Edit* | *Screens***.
- Click the ***Options*** button, click the ***Colors*** tab, then *right-click* in any field under ***DSET 1*** to select desired colors of the various screen components.

In the extreme upper right corner of the acquisition screen, there is a box, which displays the current operator type, i.e., PRI, SEC, ACQ, etc. We need to set this to ***PRI*** for primary analysis. All subsequent setup and report files will be encoded as ***PRI*** as part of the filename.

To set the operator type as ***PRI***:

STEP 1: Click the ***Options*** button at the top of the analysis screen. The ***Options*** dialog appears.

STEP 2: Click the ***Report*** tab.

STEP 3: In the ***Analysis Type Dset 1*** list box, select ***PRI***.

STEP 4: Click ***OK***. The operator type box on the main analysis screen now shows ***PRI*** as shown in Figure 14-1.

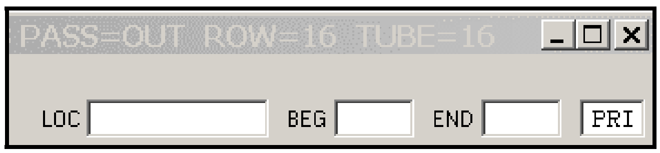


Figure 14-1. Operator Type Display Box

Next, let's load the ***History*** file that we created in the previous topic. In addition, we'll set the analysis software to automatically display historical calls from our ***History*** file when a tube with history is loaded from the ***TLIST***.

To load the ***History*** file:

STEP 1: Click the ***History*** button located along the top of the analysis screen. The ***History*** dialog appears.

STEP 2: Select ***File* | *Open***. An ***Open*** dialog appears.

STEP 3: Select and open the ***History*** file (***`hst`***) within the current project located in: ***`c:\projects\LPFH16A\history\CoreStar\project.hst.`***

STEP 4: Click ***OK***.

TIP:	This ***History*** file will remain loaded for this project – even when the project is closed and re-opened – until a different ***History*** file is loaded.

To make the software display the ***History*** dialog automatically when there's a historical call for the current tube:

STEP 1: Click the ***Options*** button at the top of the analysis screen. The ***Options*** dialog appears.

STEP 2: Click the ***Report*** tab.

STEP 3: Enable the ***Check History*** option. The box turns *green*.

STEP 4: The ***Report*** tab should look like Figure 1-3. Click ***OK***.

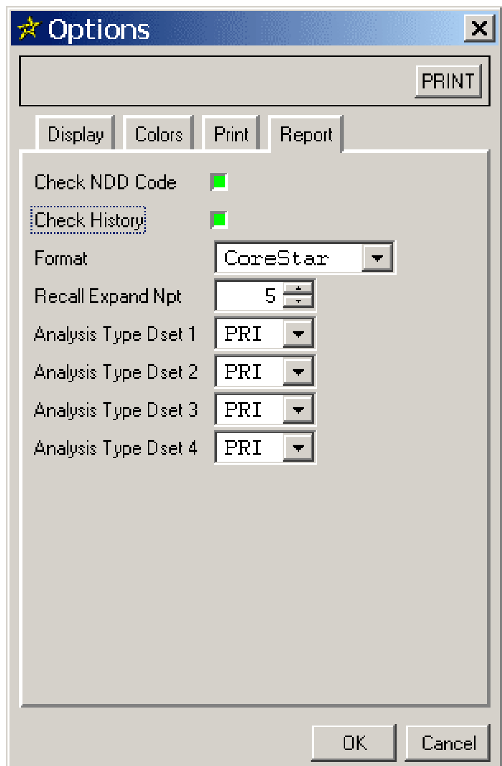


Figure 1-3. Options Dialog - Report Tab

Before we begin data analysis, we need to select a few basic options as follows:

STEP 1: Click the ***Options*** button at the top of the analysis screen. The ***Options*** dialog appears.

STEP 2: Click the ***Display*** tab.

STEP 3: Modify the settings to match the ones shown in Figure 1-5.

STEP 4: Click the ***Print*** tab.

STEP 5: Modify the settings to match the ones shown in Figure 1-7.

STEP 6: Click ***OK***.

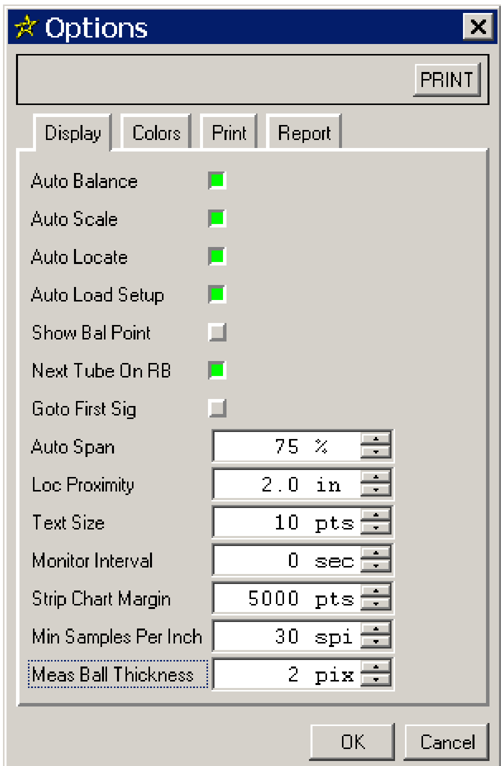


Figure 1-5. Options Dialog - Display Tab

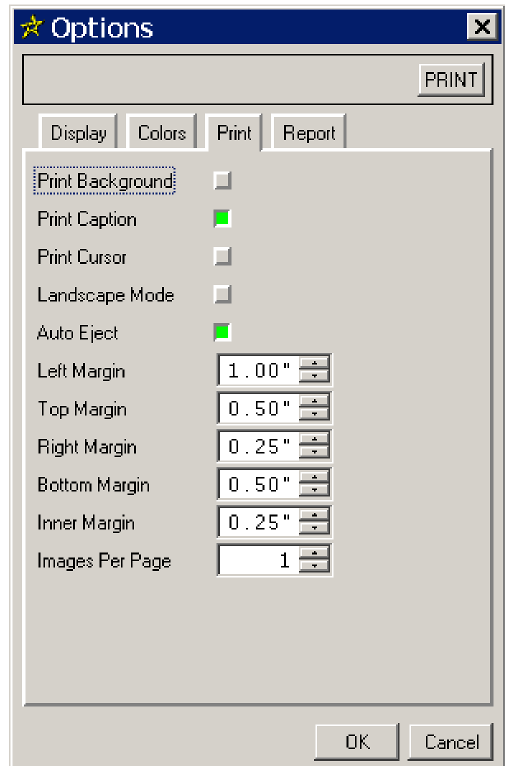


Figure 1-7. Options Dialog - Print Tab

We're now ready to play in a calibration standard run and begin setting up for data analysis:

STEP 1: Click the ***TLIST*** button at the top of the analysis screen. The ***TLIST*** dialog appears.

STEP 2: In the ***Cal*** list box, select ***2*** to *open* cal group #2. A listing of data files appears in the ***TLIST***.

STEP 3: Double-click the 3rd cal run encoded ***999 999*** (*Entry 3*).

STEP 4: Click the title of each long strip chart and Lissajous until they are set as follows:

- Left Chart: CH 1.
- Center Chart: CH 3.
- Right Chart: CH 6.
- Left Lissajous: CH 1.
- Center Lissajous: CH 3.
- Right Lissajous: CH 5.

STEP 5: Locate the ***100% TWH*** (*through-wall hole*) signal in the long strip chart. In this cal run, the ***100% TWH*** is the first signal from the top of the long chart.

STEP 6: Center and *trim down* on the ***100% TWH*** signal in the expanded strip chart so that it's clearly visible in the Lissajous window. Balance as required.

TIP:	Please see ***Mouse and Keyboard Shortcuts for Acquisition and Analysis*** in ***Appendix B***.

STEP 7: Middle-click in the ***Span*** box of any Lissajous to auto-fit the signal within all the Lissajous windows.

NOTE:	The ***Span*** box is located at the upper left corner of any Lissajous. Click in the box to increase or right-click to decrease the signal size.

STEP 8: Ensure that ***CH 1 600KHz C1*** is displayed in the title bar of the left Lissajous. If not, click the title bar of the Lissajous until ***CH 1*** is displayed.

STEP 9: Left-click twice in the left Lissajous to measure the ***100% TWH*** signal ***Vmr*** (*Volts Max-Rate*). Red (***Vpp***) & green (***Vmr***) measurement balls appear at either end of the signal. Volt and phase angle values are displayed in the measurement grid located at the lower left of the Lissajous.

NOTE:	Left-clicking once in a Lissajous measures a signal ***Vpp*** (*Volts peak-to-peak*). Only red measurement balls appear at either end of the signal.

STEP 10: Shift+left-click once in the ***Rotation*** box. The Lissajous displays 3 red vectors at 0°, 90°, and 40°.

NOTE:	The ***Rotation*** box is located at the upper right corner of any Lissajous. It contains a value with the degree symbol (°).

STEP 11: Left-click+drag in the Lissajous display and observe the phase angle in the measurement grid. Stop at 40°.

STEP 12: Repeat the 2 steps above until the ***100% TWH*** signal measures a phase angle of 40° for channels 1, 3, 5, 7, and 9 (*all differential channels*) as shown in Figure 1-9.

Note: The mix channel shown in Figure 1-9 will be created and calibrated during this topic.

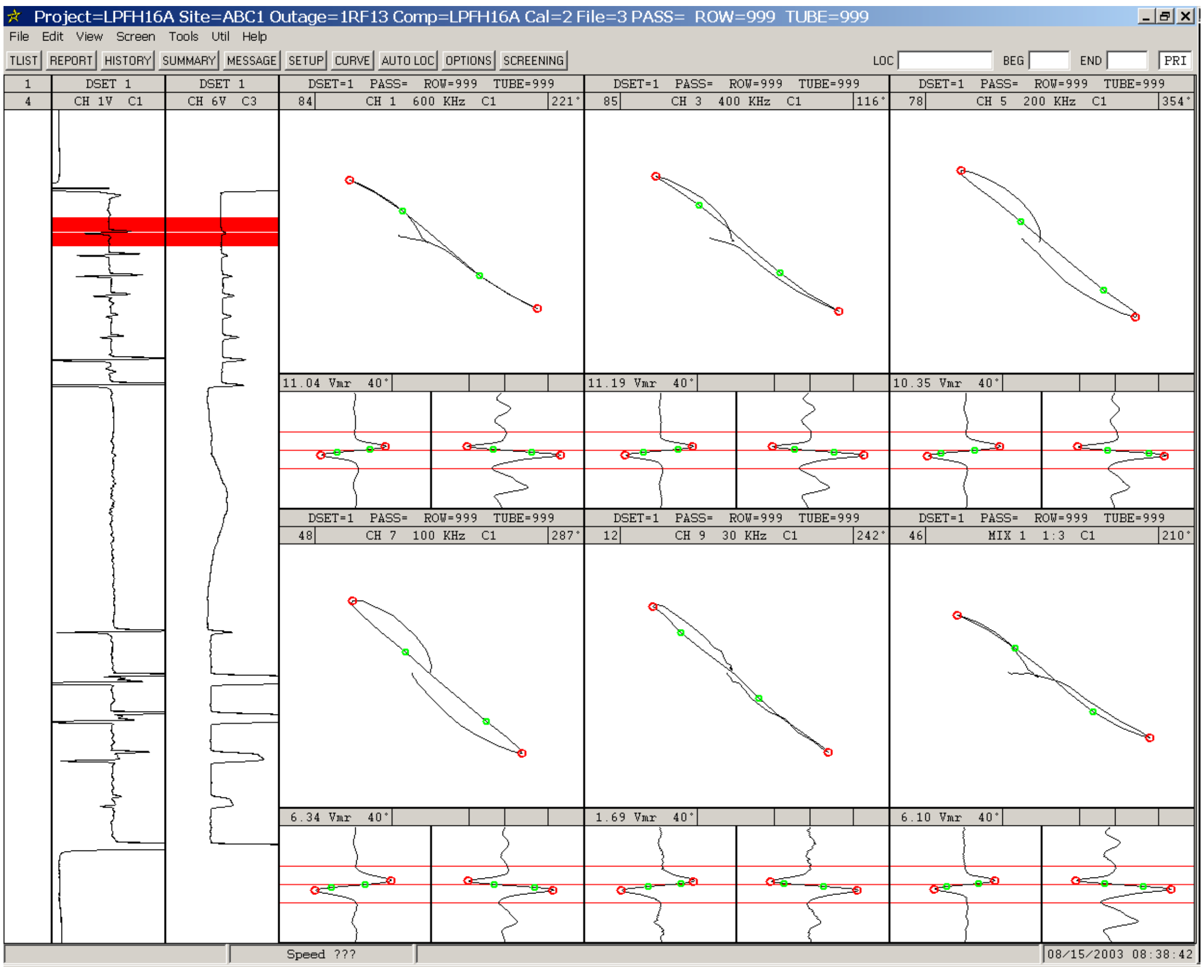


Figure 1-9. Analysis Screen - Differential Channels - Rotation Setup

TIP:	Be sure to re-measure the ***100% TWH*** signal ***Vmr*** in each Lissajous before setting the signal rotation.

STEP 13: Ensure the ***100% TWH*** signal is still displayed in the Lissajous.

STEP 14: Open the expanded window enough so that a typical probe motion (*lift-off*) signal in the calibration standard on ***CH 4 400KHz C3*** is clearly visible in addition to the ***100% TWH*** signal.

STEP 15: Left- or right-click in the ***Span*** box of each Lissajous to adjust the signal size to fit within the Lissajous.

STEP 16: Shift+left-click once in the ***Rotation*** box for ***CH 4 400KHz C3***. The Lissajous displays 3 red vectors at 0°, 90°, and 40°.

STEP 17: Left-click+drag in the Lissajous display and set the ***100% TWH*** signal vertical (*going up from null*) while setting the lift-off horizontal (*flat*).

STEP 18: Repeat the applicable preceding steps for channels 4, 6, 8, and 10. The final result for all absolute channels should be *lift-off horizontal with flaws going up* as shown in Figure 1-11.

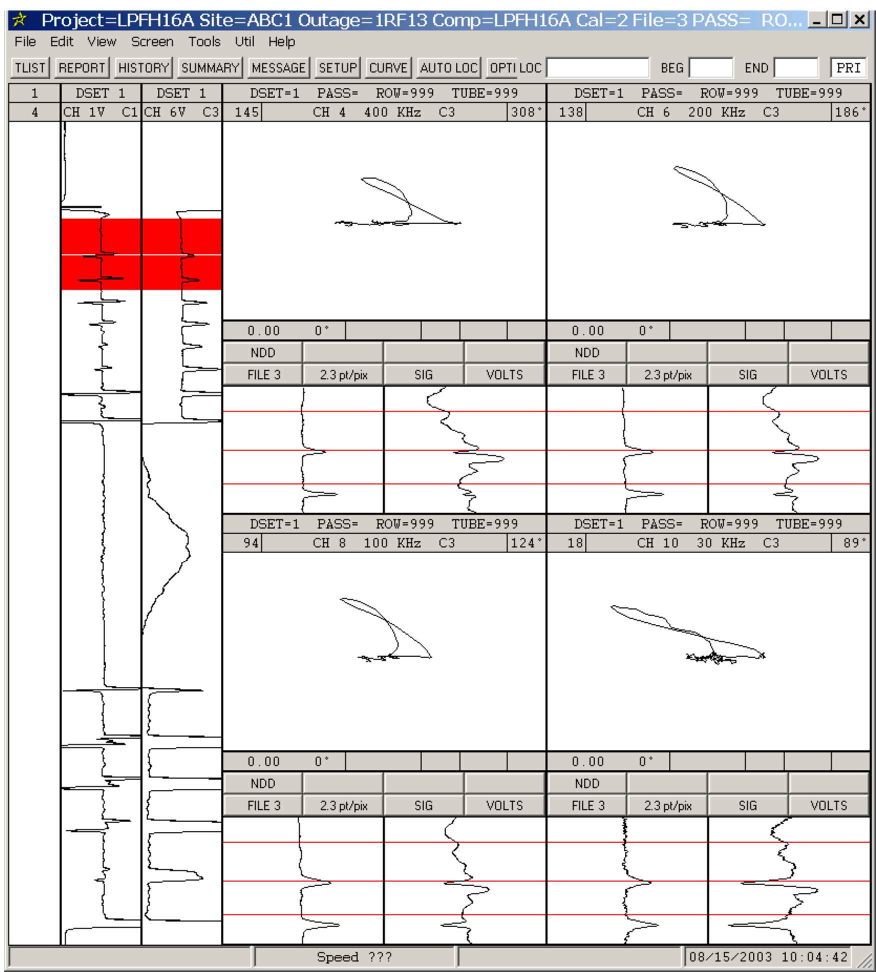


Figure 1-11. Analysis Screen - Absolute Channels - Rotation Setup

NOTE: Please disregard ***CH 2 600KHz C3*** for this eddy current data.

Before we set our voltage reference, we should create all necessary mixes and/or filters and set their rotations as well. This procedure requires a differential & absolute tube support mix (TSP) and a free-span noise mix for tube support & baffle enhancement used for the auto-locate feature (*the noise mix is not required, but strongly recommended-especially for badly pitted brass tubes*). A filter is not required for this tutorial; however, creating a filter is done in much the same way as mixes.

To create the two TSP mixes:

STEP 1: Since the cal run has no TSP, we'll need to mix on a good TSP in the feedwater. Click the ***File*** button once located under any Lissajous. This loads the next file in the ***TLIST***. The button should read ***File 4***.

STEP 2: Tube ***IN-16-1*** should be loaded. Locate, center, and trim down on the first TSP at the top of the long chart so that you screen appears to Figure 1-13. Balance as required.

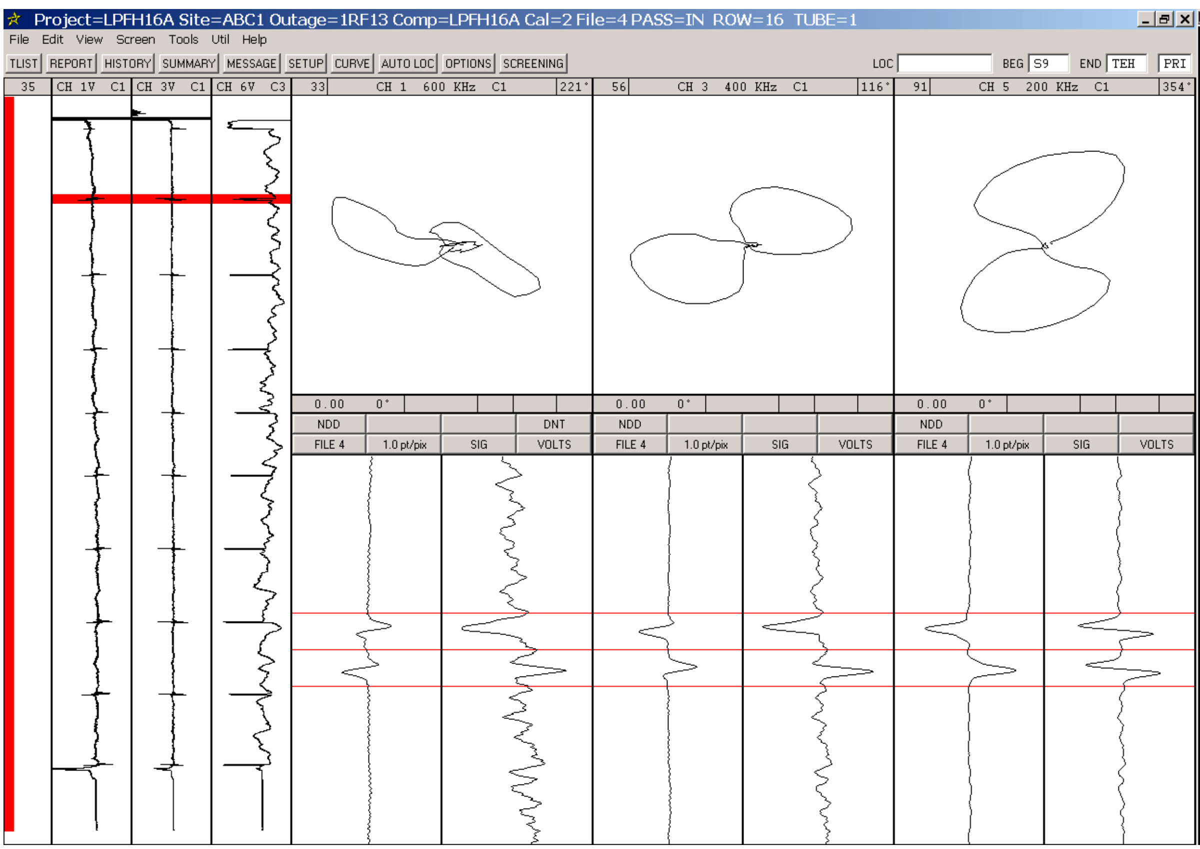


Figure 1-13. Tube Suppport Signal Prior to Mixing

STEP 3: Click the ***Setup*** button.

STEP 4: Click the ***Mix*** tab.

STEP 5: Click the ***Add Mix*** button 3 times.

STEP 6: Click or right-click (*if necessary*) in the ***Component 1*** field for M***ix # 1*** until ***1*** appears.

STEP 7: Click or right-click (*if necessary*) in the ***Component 2*** field for ***Mix # 1*** until ***3*** appears.

STEP 8: Click or right-click (*if necessary*) in the ***Component 1*** field for ***Mix # 2*** until ***4*** appears.

STEP 9: Click or right-click (*if necessary*) in the ***Component 2*** field for ***Mix # 2*** until ***6*** appears.

STEP 10: Click or right-click (*if necessary*) in the ***Component 1*** field for ***Mix # 3*** until ***9*** appears.

STEP 11: Click or right-click (*if necessary*) in the ***Component 2*** field for ***Mix # 3*** until ***1*** appears.

STEP 12: Click the ***Valid*** field (*red*) for ***Mix # 1***. The field turns white and displays ***Yes***. The TSP differential mix is now completed.

STEP 13: Click the ***Valid*** field (*red*) for ***Mix # 2***. The field turns white and displays ***Yes***. The TSP absolute mix is now completed. The ***Mix*** tab of the ***Setup*** dialog should now look like Figure 1-15.

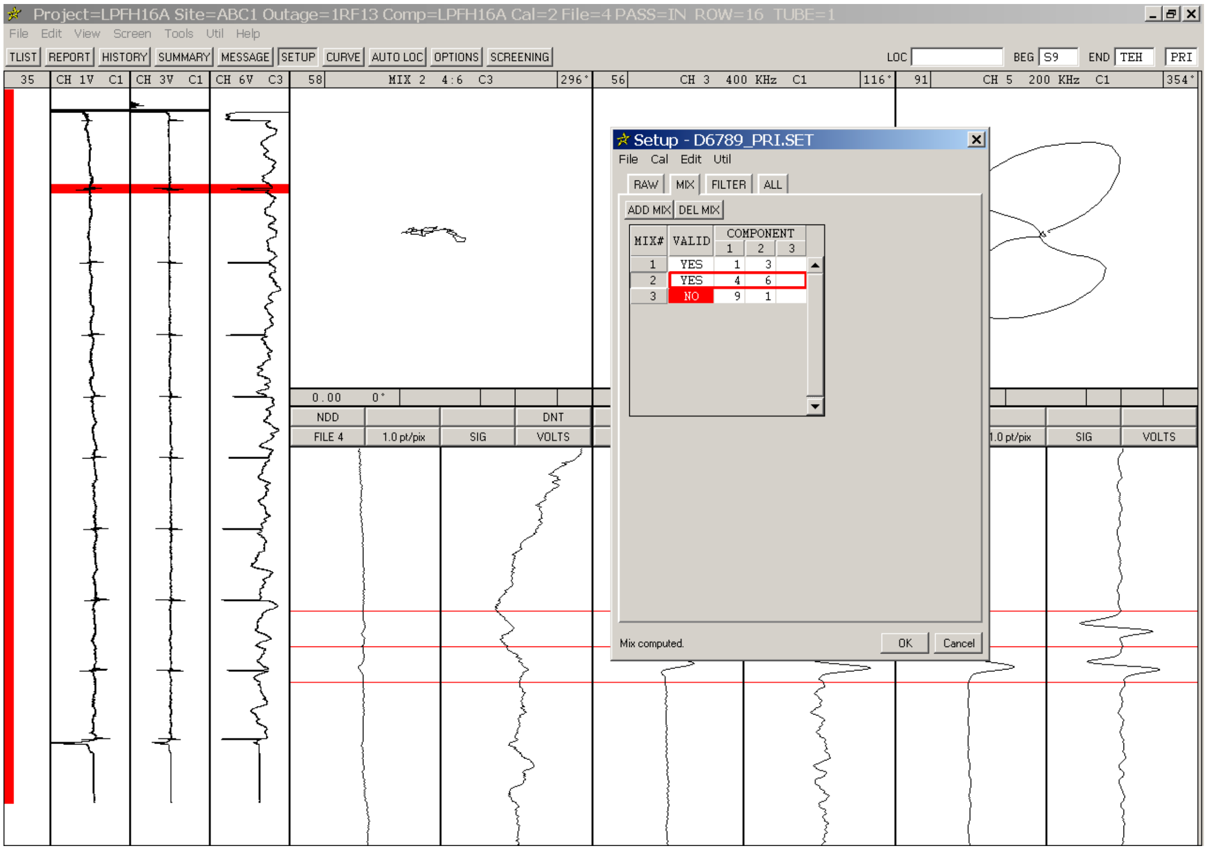


Figure 1-15. Setup Dialog - Mix Tab After Mixing Out TSP

STEP 14: Select ***CH 3 400KHz C1*** in the left Lissajous. Reposition the ***Setup*** dialog as necessary so that left Lissajous & expanded chart are clearly visible.

STEP 15: Click in the long chart to position the cursor mid-span (*between TSPs*).

STEP 16: Shift+click the ***pt/pix*** button under any Lissajous until it displays ***4.0 pt/pix***. This feature contracts and expands the data shown in the expanded chart.

STEP 17: Fully open the red-line cursors in the expanded chart.

STEP 18: Fine tune the position of the cursor in the long chart mid-span so that no part of a TSP is displayed in the Lissajous.

STEP 19: Click the ***Valid*** field (*red*) for ***Mix # 3***. The field turns white and displays Yes. The differential noise mix is now completed as shown in Figure 1-17.

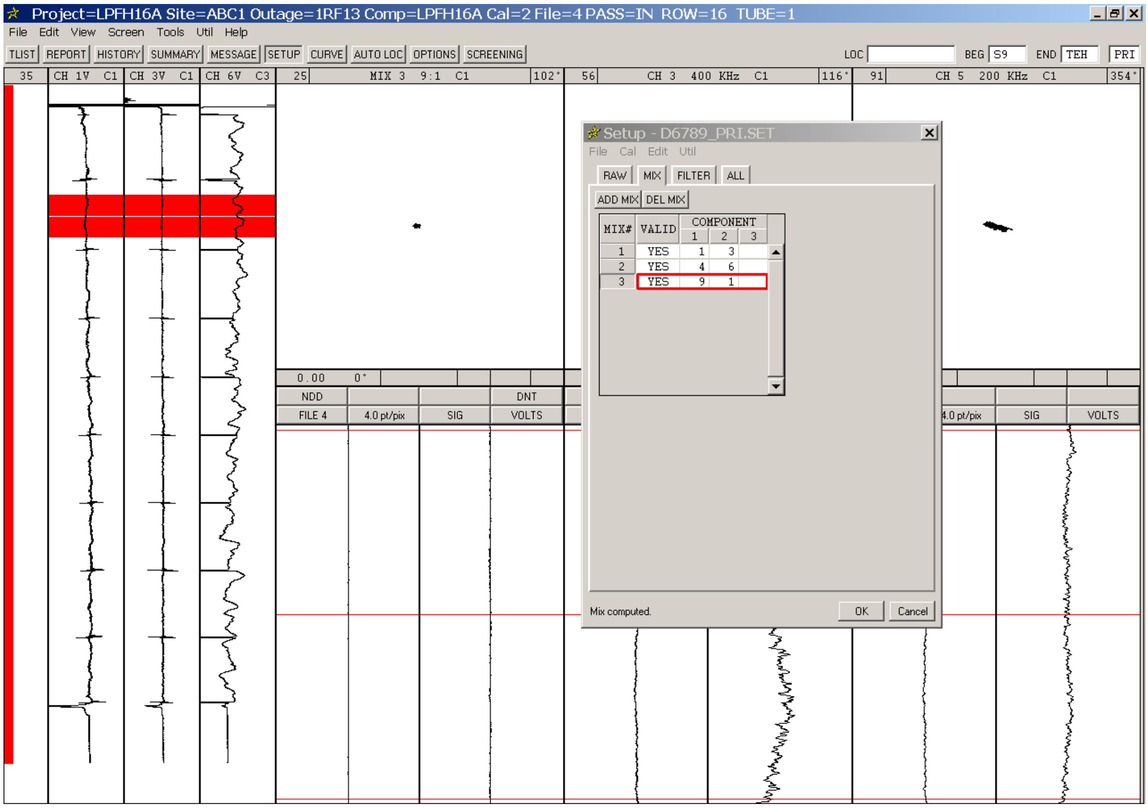


Figure 1-17. Setup Dialog - Mix Tab After Mixing Out Noise

STEP 20: Middle-click the ***pt/pix*** button under any Lissajous to reset to the default value of ***1.0 pt/pix***.

STEP 21: Click ***OK*** on the ***Setup*** dialog.

STEP 22: Repeat the appropriate previous steps in this topic for setting the phase rotation of differential and absolute signals. Set ***Mix #1*** to ***40°*** measured ***Vmr*** for the ***100% TWH*** hole, ***Mix #2*** to lift-off horizontal with flaws up, and ***Mix #3*** to ***270° Vpp*** for a typical TSP signal.

At this point, all the raw channels and mixes should be calibrated in terms of phase rotation. Next, we'll set a relative voltage reference.

STEP 1: Play in the 3rd ***ASME*** cal run again. Balance as required.

STEP 2: Ensure that ***CH 1 600KHz C1*** is displayed in the title bar of the left Lissajous. If not, click the title bar of the Lissajous until ***CH 1*** is displayed.

STEP 3: Locate the ***4-20% FBH*** (flat-bottom hole) signal in the long strip chart. In this cal run, the ***4-20% FBH*** is the 5th signal from the top of the long chart.

STEP 4: Center and trim down on the ***4-20% FBH*** signal in the expanded strip chart so that it's clearly visible in the ***CH 1*** Lissajous window.

STEP 5: Click once in the left Lissajous to measure the ***4-20% FBH*** signal ***Vpp***. Red measurement balls appear at either end of the signal.

STEP 6: Click the ***Volts*** button immediately below the ***CH 1*** Lissajous. The ***Set Volts*** dialog appears.

STEP 7: In the ***Propagate*** area, enable ***Same Ratio*** (*green*).

STEP 8: In the ***Which Chans*** area, enable ***All Channels*** (*green*).

STEP 9: In the ***Volts*** field, enter ***4***.

STEP 10: The ***Set Volts*** dialog should now look like Figure 1-18. Click ***OK***.

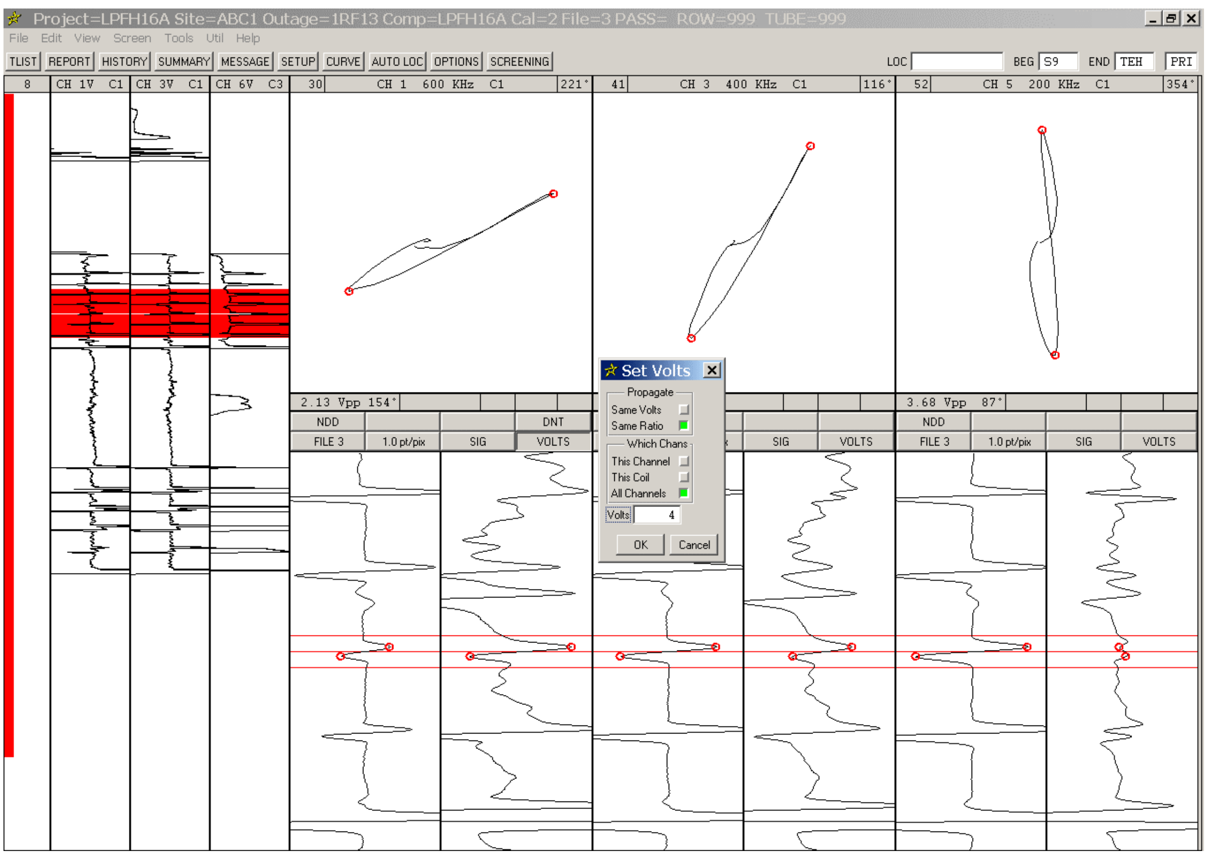


Figure 1-18. Analysis Screen - Setting Relative Voltage

STEP 11: Click in the left Lissajous to re-measure the ***4-20% FBH*** signal ***Vpp***. The voltage displayed for the ***CH 1*** Lissajous should read ***4.00 Vpp***. All other channels & mixes are now set *in reference to* the voltage level set for ***CH 1***.

> **TIP:** This is simply one way of setting a voltage reference. Depending on your analysis procedure, you may be required to set the voltage reference for each channel, mix, & filter independently or by coil. Experiment with the ***Set Volts*** dialog to see the different combinations you can use and their effects.

We're not finished with our setup yet, but we now have quite a *time investment* in setting up for data analysis, so it would be prudent of us to save our setup at this stage, and then continue. It's a good idea to store your setup often during the course of setting up.

To store the current setup:

STEP 1: Click the ***Setup*** button. The ***Setup*** dialog appears.

STEP 2: Select ***Cal | Store Cal Setup*** button. If you get a message box asking you if you want to overwrite the existing setup, click ***Yes***.

STEP 3: Click ***OK*** on the ***Setup*** dialog.

> **NOTE:** This action stores a setup file as follows:
>
> ***LPFH16A\ecdata\000002.CAL\D6789_PRI.SET***

Let's create our differential *phase-to-depth* calibration curves next. Afterwards, we'll create some *volts-to-depth* curves for a couple absolute channels.

STEP 1: Click the ***Curve*** button along the top of the analysis screen. The ***Curve*** dialog appears.

STEP 2: Click & drag the ***Curve*** dialog by the title bar and position it just to the right of the left Lissajous. The left Lissajous is synchronized with the ***Curve*** dialog channel selection button.

STEP 3: Click or right-click the channel selection button in the upper left corner of the ***Curve*** dialog until ***CH 1 600 KHz*** appears in the button and in the left Lissajous.

IMPORTANT:	**Always** use the channel selection button on the the ***Curve*** dialog for the left Lissajous when the ***Curve*** dialog is opened and calibration curves are being setup. **Do not** change the channel in the left Lissajous by clicking the title.

STEP 4: Click the ***DEG*** tab at the bottom of the ***Curve*** dialog.

STEP 5: Click the ***ADD*** button once.

STEP 6: Click the channel selection button until ***CH 3 400 KHz*** appears in the button and in the left Lissajous.

STEP 7: Click the ***DEG*** tab.

STEP 8: Click the ***ADD*** button once.

STEP 9: Repeat the preceding 3 steps for channels ***5***, *7*, ***9***, and ***Mix 1***.

STEP 10: Click or right-click the channel selection button on the ***Curve*** dialog until ***CH 1 600 KHz*** appears in the button and in the left Lissajous.

STEP 11: Center and *trim down* on the ***100% TWH*** signal in the expanded strip chart so that it's clearly visible in the left Lissajous window. Balance as required.

STEP 12: Left-click twice in the left Lissajous to measure the ***100% TWH*** signal ***Vmr***. Red (***Vpp***) & green (***Vmr***) measurement balls appear at either end of the signal.

STEP 13: Click once in the ***Phase*** field for the ***100%TW*** on the ***DEG*** table. As shown in Figure 1-19, the phase measurement (~40°) for ***CH 1*** is automatically entered in the field and the curve starts to change.

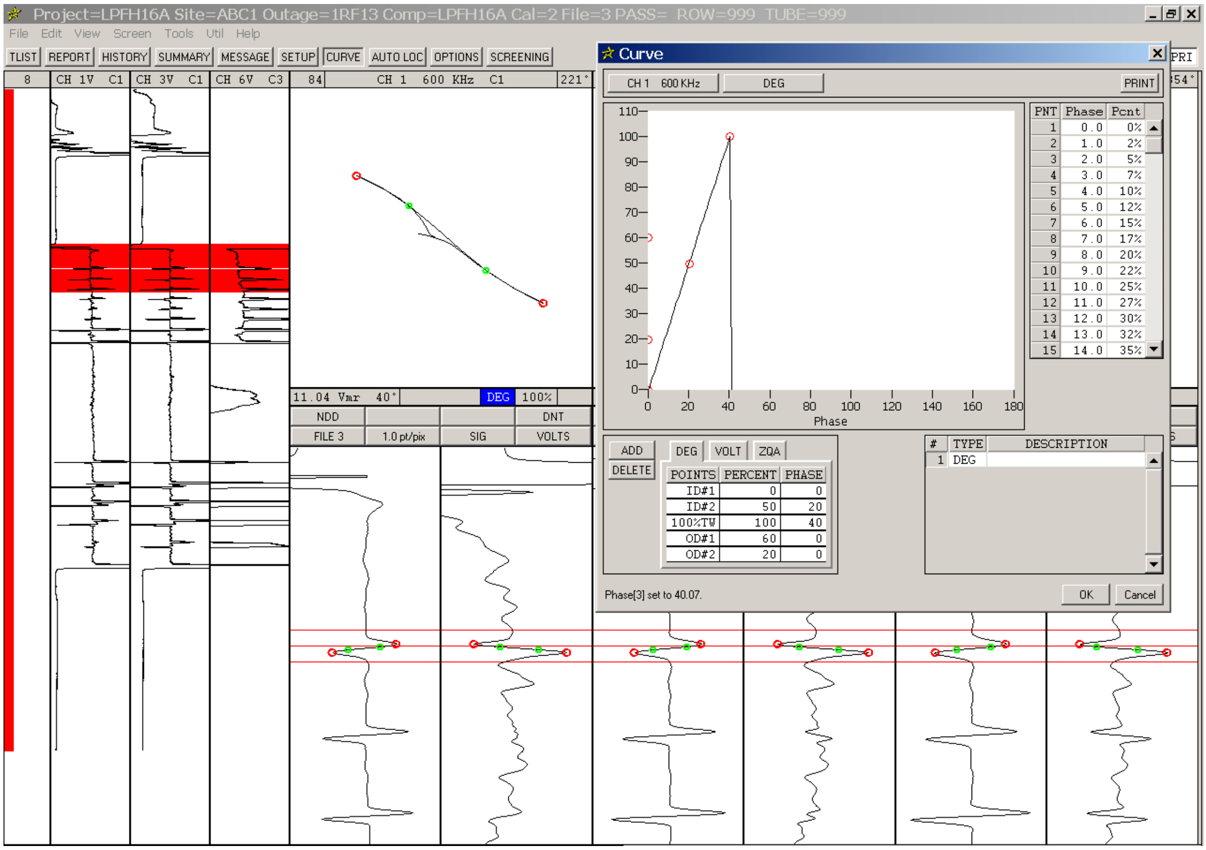


Figure 1-19. Analysis Screen - Curve Dialog - Entering the Phase of a Calibration Flaw

STEP 14: Click the channel selection button on the ***Curve*** dialog until ***CH 3 400 KHz*** appears in the left Lissajous.

STEP 15: Left-click twice in the left Lissajous to measure the ***100% TWH*** signal ***Vmr***.

STEP 16: Click in the ***Phase*** field for ***100%TW*** on the ***DEG*** table. The phase measurement (~40°) for ***CH 3*** is automatically entered in the field.

STEP 17: Repeat the preceding 3 steps for channels ***5***, ***7***, ***9***, and ***Mix 1***.

STEP 18: Click or right-click the channel selection button on the ***Curve*** dialog until ***CH 1 600 KHz*** appears in the button and in the left Lissajous.

STEP 19: Center and *trim down* on the ***58% FBH*** signal in the expanded strip chart so that it's clearly visible in the left Lissajous window. This signal is the 3rd from the top of the cal run in the long chart.

STEP 20: Click as required in the left Lissajous to measure the ***58% FBH*** signal ***Vmr***.

STEP 21: Click or right-click as required in the ***Percent*** field for ***OD#1*** on the ***DEG*** table and set the value to ***58***. This is the *as-built* depth for the nominal 60% FBH on this ***ASME*** standard.

STEP 22: Middle-click in the ***Percent*** field for ***OD#1*** on the ***DEG*** table. This sets this field to a value of ***58*** for the remaining differential curves, except for the mix channels.

STEP 23: Click in the ***Phase*** field for ***OD#1*** on the ***DEG*** table. The phase measurement (~106°) of the ***58% FBH*** signal for ***CH 1*** is automatically entered in the field as shown in Figure 1-20.

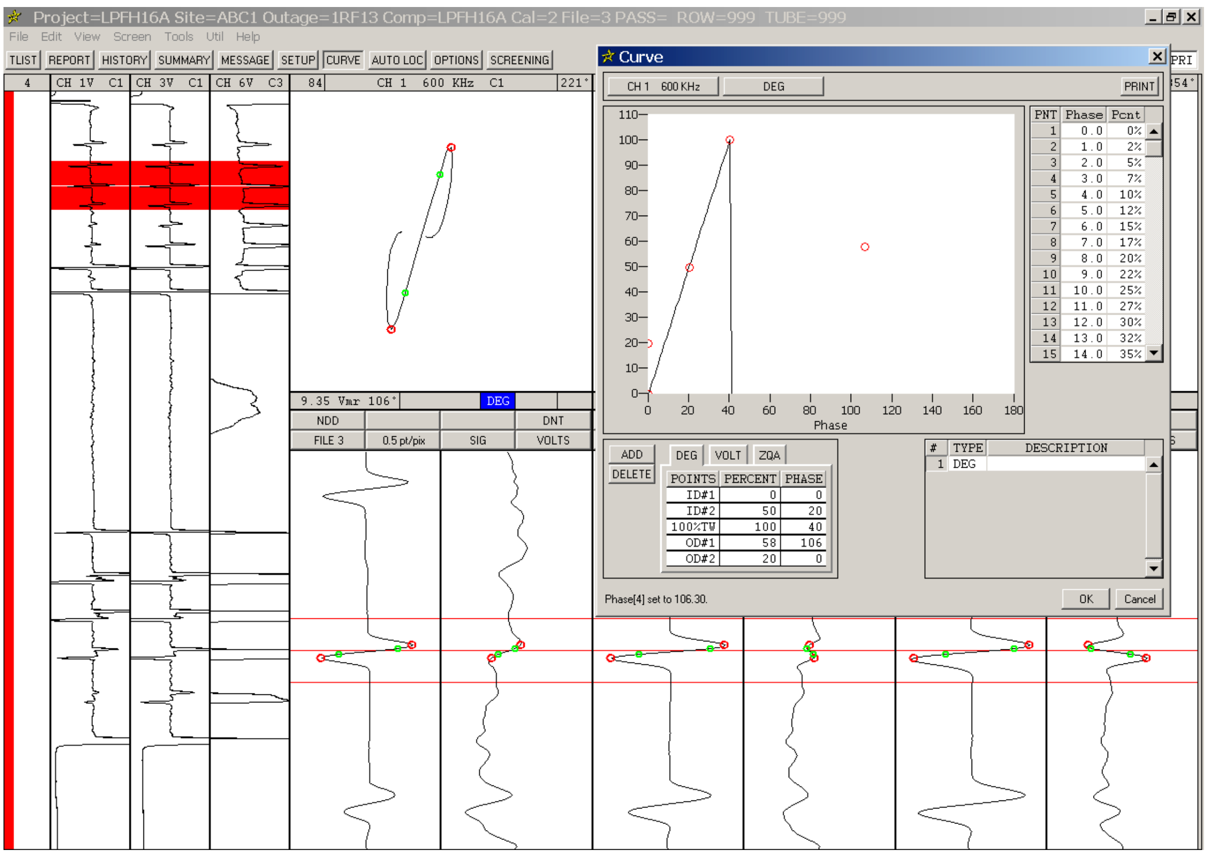


Figure 1-20. Analysis Screen - Curve Dialog - Entering the As-built Depth of a Calibration Flaw

STEP 24: Click the channel selection button until ***CH 3 400 KHz*** appears in the left Lissajous.

STEP 25: Click as required in the left Lissajous to measure the ***58% FBH*** signal ***Vmr***.

STEP 26: Click in the ***Phase*** field for ***OD#1*** on the ***DEG*** table. The phase measurement (~85°) for ***CH 3*** is automatically entered in the field.

STEP 27: Repeat the preceding 3 steps for channels ***5***, ***7***, ***9***, and ***Mix 1***. You will need to set the ***Percent*** field for ***OD#1*** on the ***DEG*** table for ***Mix 1*** to ***58***.

STEP 28: Click or right-click the channel selection button on the ***Curve*** dialog until ***CH 1 600 KHz*** appears in the left Lissajous.

STEP 29: Center and *trim down* on the ***4-20% FBH*** signal in the expanded strip chart so that it's clearly visible in the left Lissajous window. This signal is the 5th from the top of the cal run in the long chart.

STEP 30: Click as required in the left Lissajous to measure the ***4-20% FBH*** signal ***Vmr***.

STEP 31: Ensure that the ***Percent*** field for ***OD#2*** on the ***DEG*** table is set to ***20***.

STEP 32: Middle-click in the ***Percent*** field for ***OD#2*** on the ***DEG*** table. This ensures that this field is set to a value of ***20*** for the remaining differential curves, except for the mix channels.

STEP 33: Click in the ***Phase*** field for ***OD#2*** on the ***DEG*** table. The phase measurement (~153°) of the ***4-20% FBH*** signal for ***CH 1*** is automatically entered in the field as shown in Figure 1-21.

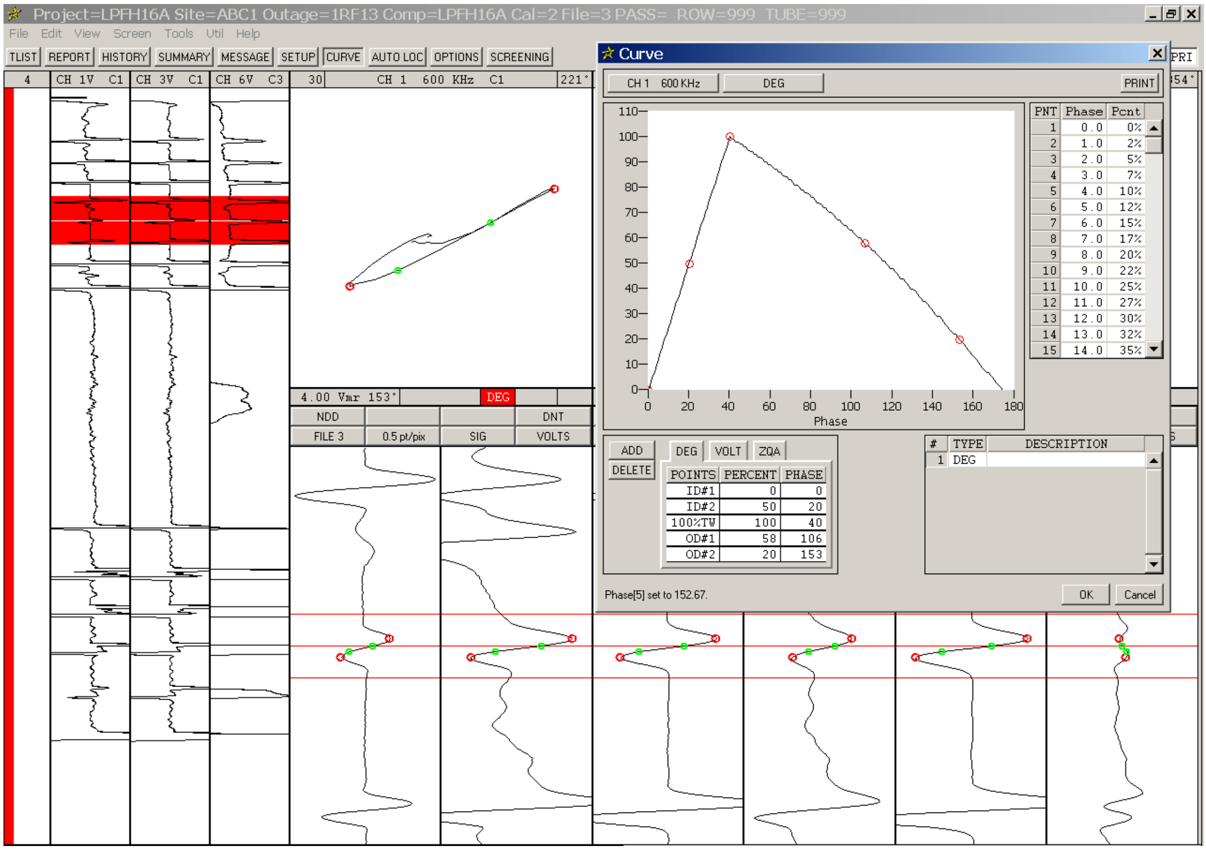


Figure 1-21. Analysis Screen - Curve Dialog – Completing a Phase Curve

STEP 34: Click the channel selection button until ***CH 3 400 KHz*** appears in the left Lissajous.

STEP 35: Click as required in the left Lissajous to measure the ***4-20% FBH*** signal ***Vmr***.

STEP 36: Click in the ***Phase*** field for ***OD#2*** on the ***DEG*** table. The phase measurement (~119°) for ***CH 3*** is automatically entered in the field.

STEP 37: Repeat the preceding 3 steps for channels ***5***, ***7***, ***9***, and ***Mix 1***. Ensure the *as-built* percent is correct for ***Mix 1***.

STEP 38: Click ***OK*** on the ***Curve*** dialog.

STEP 39: Store the setup.

Well, that takes care of all the differential channels and mixes. Let's move on and create the *volts-to-depth* calibration curves for the absolute channels. The calibration flaws of interest for this series of steps are the 180° TSP wear scars (*saddle wear*). They are the 4 signals in the cal run at the bottom of the long chart. They are best viewed in the long strip chart on ***CH 6***. Their *as-built* depths from bottom to top are 18, 32, 47, and 57%.

STEP 1: Click the ***Curve*** button along the top of the analysis screen. The ***Curve*** dialog appears.

STEP 2: Click & drag the ***Curve*** dialog by the title bar and position it just to the right of the left Lissajous.

STEP 3: Click or right-click the channel selection button in the upper left corner of the ***Curve*** dialog until ***CH 4 400 KHz*** appears in the left Lissajous.

STEP 4: Set the right long strip chart to ***CH 4***.

STEP 5: Adjust the ***Span*** of ***CH 4*** so that all 4 TSP wear scars are clearly visible in the long chart.

STEP 6: Click the ***VOLT*** tab at the bottom of the ***Curve*** dialog.

STEP 7: Click the ***ADD*** button once.

STEP 8: Click the channel selection button until ***CH 6 200 KHz*** appears in the left Lissajous.

STEP 9: Click the ***VOLT*** tab at the bottom of the ***Curve*** dialog.

STEP 10: Click the ***ADD*** button once.

STEP 11: Repeat the preceding 3 steps for channels ***8*** and ***Mix 2***.

> **NOTE:** To this point, you should have ***DEG*** curves assigned to channels ***1***, ***3***, ***5***, ***7***, ***9***, and ***Mix 1***, while ***VOLT*** curves are assigned to channels ***4***, ***6***, ***8***, and ***Mix 2***.

STEP 12: Click or right-click the channel selection button on the ***Curve*** dialog until ***CH 4 400 KHz*** appears in the left Lissajous.

STEP 13: Click or right-click the ***Percent*** field for ***PT#2*** in the ***VOLT*** table at the bottom of the ***Curve*** dialog until the *as-built* value of ***32*** is displayed.

STEP 14: Middle-click the ***Percent*** field for ***PT#2*** to set the *as-built* value of ***32*** for channels 6 and 8.

STEP 15: Click or right-click the ***Percent*** field for ***PT#3*** in the ***VOLT*** table at the bottom of the ***Curve*** dialog until the *as-built* value of ***47*** is displayed.

STEP 16: Middle-click the ***Percent*** field for ***PT#3*** to set the *as-built* value of ***47*** for channels 6 and 8.

STEP 17: Click or right-click the channel selection button on the ***Curve*** dialog until ***MIX 2*** appears in the left Lissajous.

STEP 18: Click or right-click the ***Percent*** field for ***PT#2*** in the ***VOLT*** table at the bottom of the ***Curve*** dialog until the *as-built* value of ***32*** is displayed.

STEP 19: Click or right-click the ***Percent*** field for ***PT#3*** in the ***VOLT*** table at the bottom of the ***Curve*** dialog until the *as-built* value of ***47*** is displayed.

STEP 20: Click or right-click the channel selection button on the ***Curve*** dialog until ***CH 4 400 KHz*** appears in the left Lissajous.

STEP 21: Position and *trim down on* the ***32% TSP Wear Scar*** in the expanded strip chart of the left Lissajous. This is the 2nd signal from the bottom of the cal run.

STEP 22: Adjust the ***Span*** and balance as required to clearly view the signal in the left Lissajous.

STEP 23: Click the left Lissajous twice to measure the signal using ***Vvm*** (*Volts Vertical Maximum or VertMax*). Yellow measurement balls will appear at the extreme top and bottom of the signal.

STEP 24: Click in the ***Volts*** field for ***PT#2*** on the ***VOLT*** table. The ***Vvm*** measurement (~23.74) for ***CH 4*** is automatically entered in the field as shown in Figure 1-22.

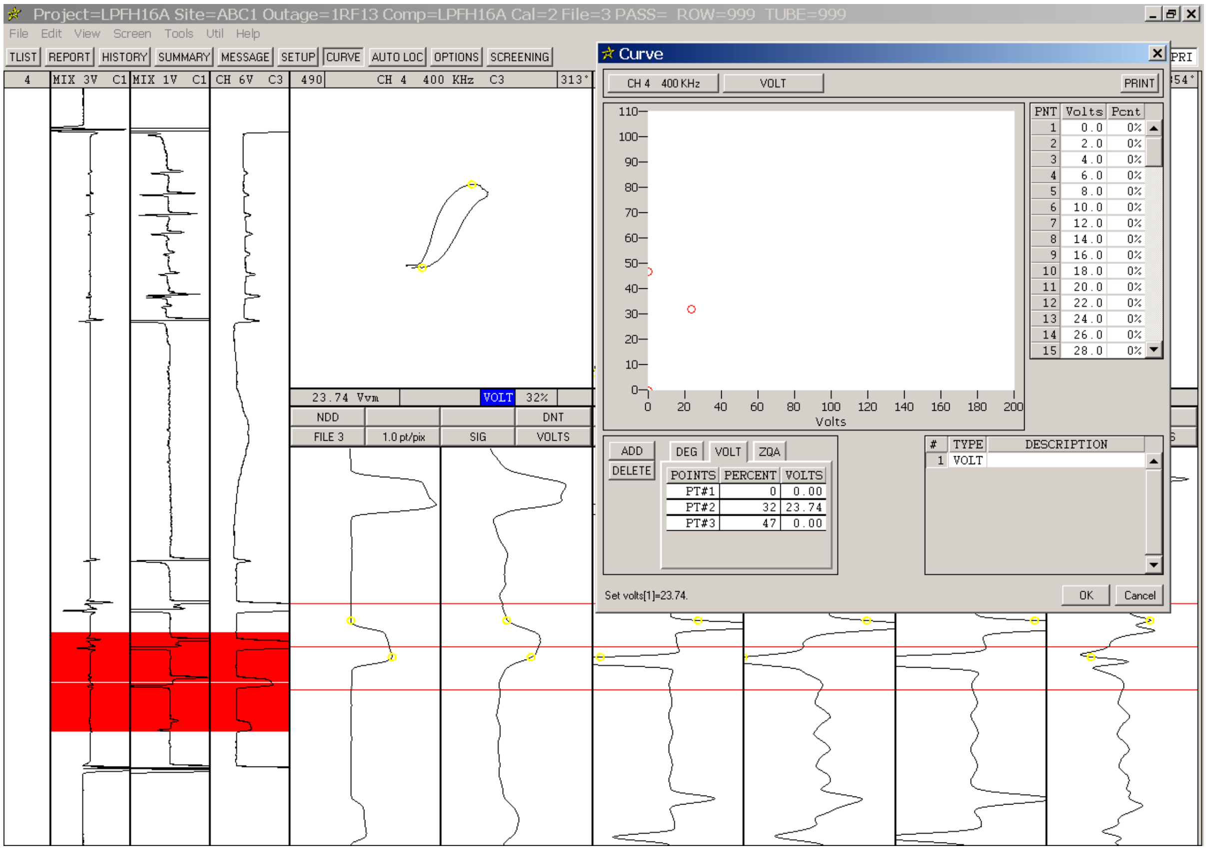


Figure 1-22. Analysis Screen - Curve Dialog - Entering the As-Built Depth and Volts Measurement of a Calibration Flaw

STEP 25: Click the channel selection button until ***CH 6 200 KHz*** appears in the left Lissajous.

STEP 26: Click the left Lissajous as required to measure the signal using ***Vvm***.

STEP 27: Click in the ***Volts*** field for ***PT#2*** on the ***VOLT*** table. The ***Vvm*** measurement (~39.54) for ***CH 6*** is automatically entered in the field.

STEP 28: Repeat the preceding 3 steps for channels ***8*** and ***Mix 2***.

STEP 29: Click or right-click the channel selection button on the ***Curve*** dialog until ***CH 4 400 KHz*** appears in the left Lissajous.

STEP 30: Position and trim down on the ***47% TSP Wear Scar*** in the expanded strip chart of the left Lissajous. This is the 3rd signal from the bottom of the cal run.

STEP 31: Adjust the ***Span*** and balance as required to clearly view the signal in the left Lissajous.

STEP 32: Click the left Lissajous twice to measure the signal using ***Vvm***.

STEP 33: Click in the ***Volts*** field for ***PT#3*** on the ***VOLT*** table. As shown in Figure 1-23, the ***Vvm*** measurement (~49.20) for ***CH 4*** is automatically entered in the field and the final curve for this channel is displayed.

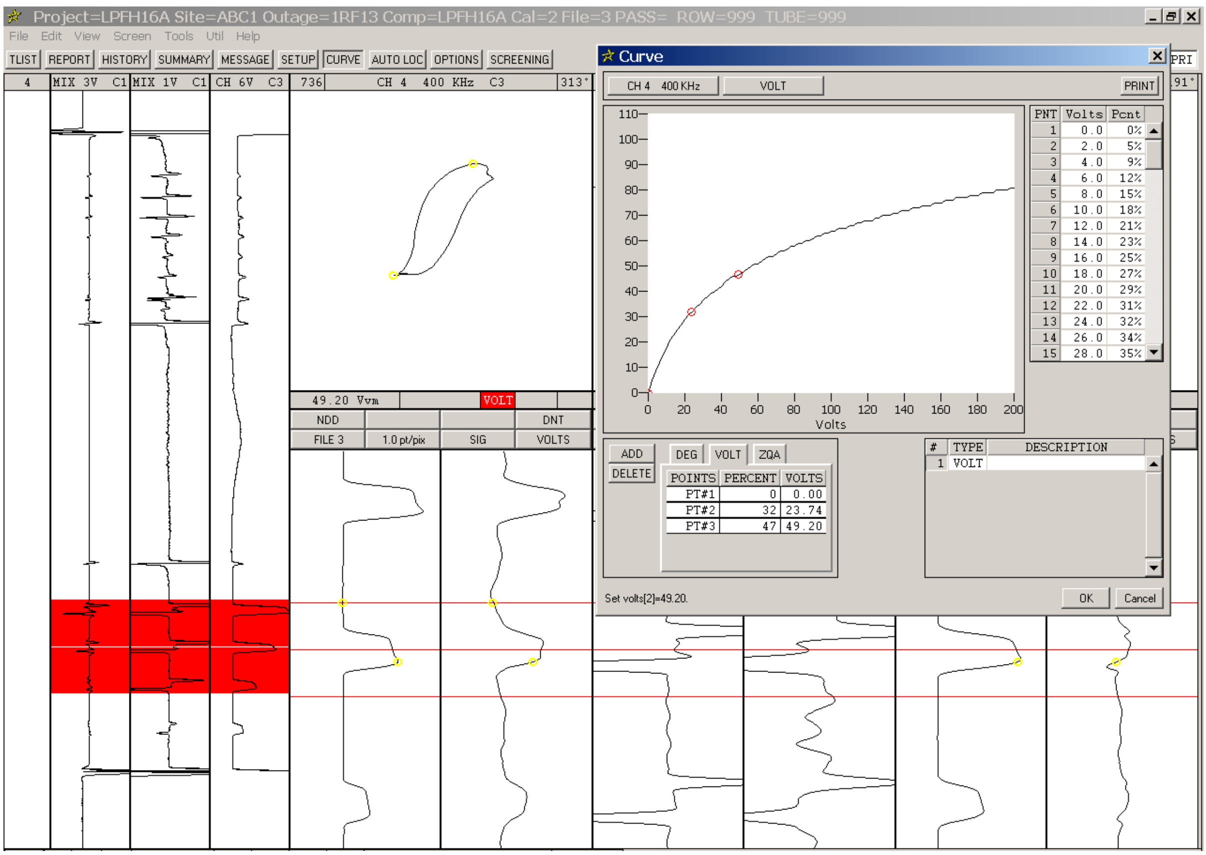


Figure 1-23. Analysis Screen - Curve Dialog – Completing a Volts Curve

STEP 34: Click the channel selection button until ***CH 6 200 KHz*** appears in the left Lissajous.

STEP 34: Click the left Lissajous as required to measure the signal using ***Vvm***.

STEP 35: Click in the ***Volts*** field for ***PT#3*** on the ***VOLT*** table. The ***Vvm*** measurement (~70.82) for ***CH 6*** is automatically entered in the field and the final curve for this channel is displayed.

STEP 36: Repeat the preceding 3 steps for channels ***8*** and ***Mix 2***.

STEP 37: Click ***OK*** on the ***Curve*** dialog.

STEP 38: Store the setup.

So now, we're all calibrated and nearly ready to start analyzing data. There are a few more details we need to address before we get started. First, let's embed a header in our eddy current report. It's a good idea to always insert an embedded header and/or footer while a cal run is loaded.

To insert a report header:

STEP 1: Click the ***Options*** button. The ***Options*** dialog appears.

STEP 2: Click the ***Report*** tab.

STEP 3: Ensure that ***CoreStar*** is selected in the ***Format*** list box.

STEP 4: As shown in Figure 1-24, ensure that ***Check NDD Code*** is enabled (*green*). This is a quality check that either warns the user when all calls in a tube are being replaced with a single ***NDD*** call <u>OR</u> when an ***NDD*** call is being replaced with another call(s).

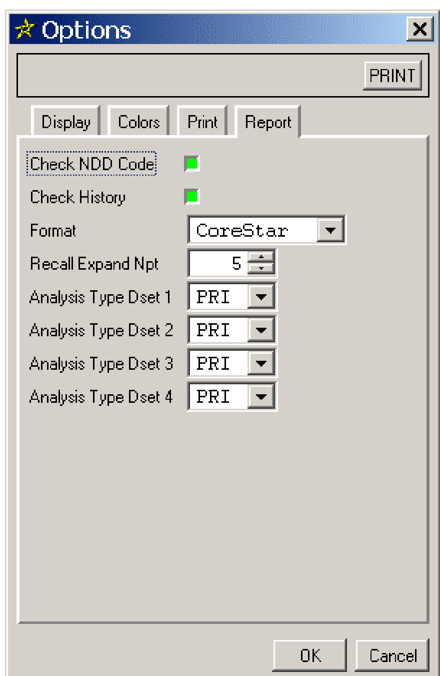


Figure 1-24. Options - Report Tab - Check NDD Code

STEP 5: Click ***OK***.

STEP 6: Click the ***Report*** button. The ***Report*** dialog appears.

STEP 7: To ensure the current report is clear, select ***Edit | Clear Report***, and then click ***Yes*** in the message box.

STEP 8: Click anywhere along the dark gray row in the ***Report*** dialog. A blue cursor appears.

STEP 9: Right-click on the blue cursor and select ***Start Cal*** from the popup menu. As shown in Figure 1-25, certain information is automatically entered. Embedded headers and footers will have a gray background.

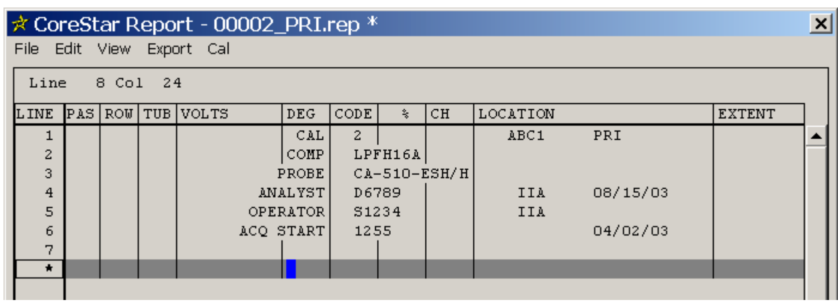


Figure 1-25. Report Dialog - Using an Embedded Header

STEP 10: Click ***OK***. The report with the newly embedded header now appears in the report at the bottom of the analysis screen.

TIPS:

- Embedded header and footer macros can be easily modified for the current project by selecting ***Edit | Report Macros*** from the main screen.
- To delete report entries, including headers and footers, simply ***click and drag*** in the report to highlight (*blue*) the desired entries, and then press the ***Delete*** key on the keyboard.
- In addition, the printed report can include a report title, report header, page headers and page footers as desired. Select ***Edit | Print Options*** in the ***Report*** dialog.
- Select ***Edit | Print Options*** in the ***Report*** dialog, and then click the ***Options*** tab to modify the margins for printed reports.

Let's setup the ***Screening*** feature so that everything time we load a tube from the ***TLIST***, initial screening levels and the desired channels are reset – kind of like *an analysis starting point* for each tube. This avoids having to reload the cal setup every time a Lissajous or long strip chart is changed or ***Span*** values are changed or even the distance between the moveable red cursors in the expanded window is changed.

To use ***Screening***:

STEP 1: With the cal run still loaded, locate, center, and trim down on the ***100% TWH*** signal.

STEP 2: Select ***Screen | ANA-3L/3S***.

STEP 3: Set the 3 long strip charts left-to-right as ***Mix 3***, ***Mix 1***, & ***CH 6V***.

STEP 4: Set the 3 Lissajous' left-to-right as ***Mix 1***, ***CH 3***, & ***CH 5***.

STEP 5: Middle-click any ***Span*** box in any Lissajous in order to auto-span the ***100% TWH*** signal.

STEP 6: Adjust the ***Span*** of ***Mix 3*** to ***~60***. This will reduce the size of the tube supports in the data to a reasonable size.

STEP 7: Locate, center, and trim down on the ***32% TSP Wear Scar*** signal. This is the 2nd signal from the bottom of the cal run.

STEP 8: Set a Lissajous so that ***CH 4*** is displayed.

STEP 9: Adjust the ***Span*** of ***CH 4*** so that the size of the ***32% TSP Wear Scar*** signal is about 4 divisions.

STEP 10: Repeat the 2 preceding steps for channels ***6***, ***8***, ***10***, and ***Mix 2***.

STEP 11: Open the expanded chart to about 50%.

STEP 12: Ensure that the 3 long strip charts and 3 Lissajous' are set per Steps 3 & 4 above.

STEP 13: Store the setup. This way you can always load your setup in case your ***Screening*** defaults are inadvertently overwritten.

STEP 14: Click the ***Screening*** button. The button appears depressed (*enabled*). A file named ***`screening.set`*** is stored to the project. This file is reloaded every time a data file is loaded.

NOTES:	• The current ***`screening.set`*** is overwritten anytime the ***Screening*** button is toggled. • The current ***`screening.set`*** can be reloaded at anytime by right-clicking the ***Screening*** button.

All we have left to do is load a tube, setup auto-locate, and start analyzing.

To setup or *train* auto-locate:

STEP 1: Click the ***File*** button in any Lissajous until ***File 4*** (*the first tube*) is displayed in the button. The header should show tube ***IN-16-1*** and the data file should be loaded.

STEP 2: In the long strip chart, position the cursor at the tube end signal. The tube end signal is the first large amplitude signal at the top of the long chart.

STEP 3: Set ***CH 9*** in the left Lissajous.

STEP 4: Adjust the ***Span*** and expanded window of ***CH 9*** so that the tube end signal is clearly visible in the Lissajous as shown in Figure 1-26.

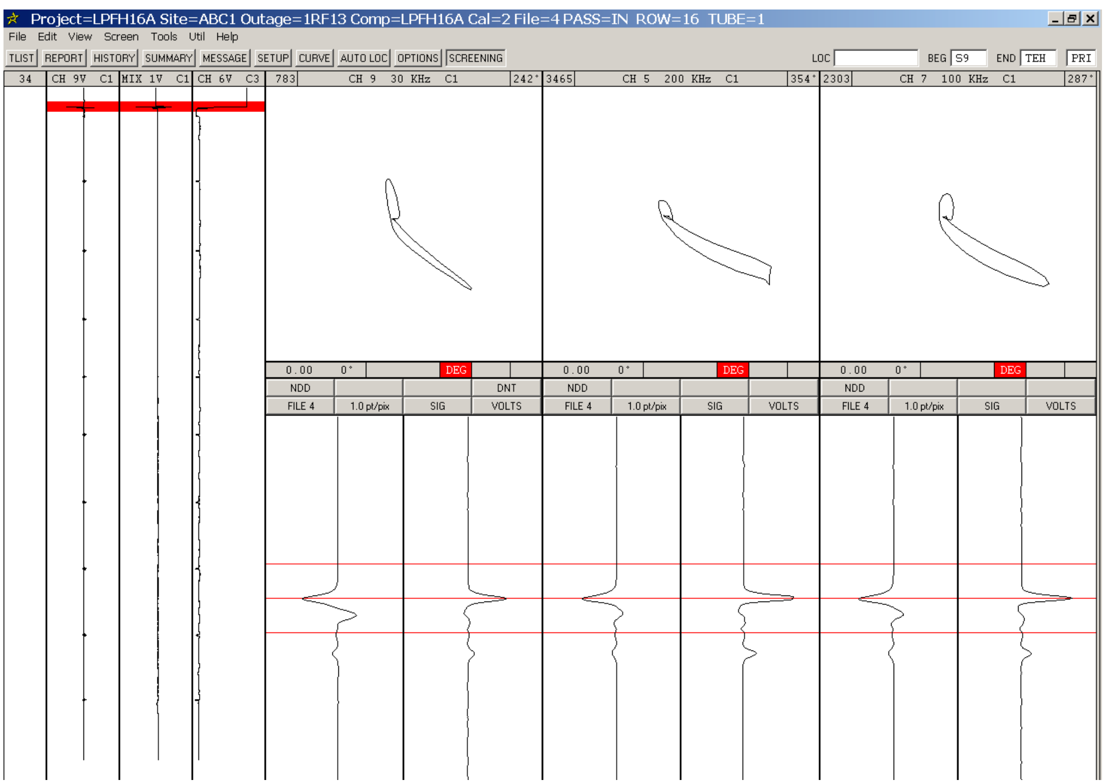


Figure 1-26. Auto-locate - Identifying the Tube End

STEP 5: Click the ***Auto Loc*** button along the top of the analysis screen. The ***Auto Loc*** dialog appears.

STEP 6: In the ***Nom PPI*** field, enter ***60***.

STEP 7: In the ***Chan*** field for ***TEH***, click until ***9*** is displayed.

STEP 8: In the ***Chan*** field for ***TSH***, click until ***9*** is displayed.

STEP 9: In the ***Chan*** field for ***SP***, click until ***M3*** is displayed.

STEP 10: Click once in the ***CNT*** (*count*) field for ***TEH***. The number ***1*** appears.

STEP 11: Click the ***FIND ENDS*** button. As shown in Figure 1-27, ***TEH*** should appear in the ***Landmark*** strip to the left of the long strip charts.

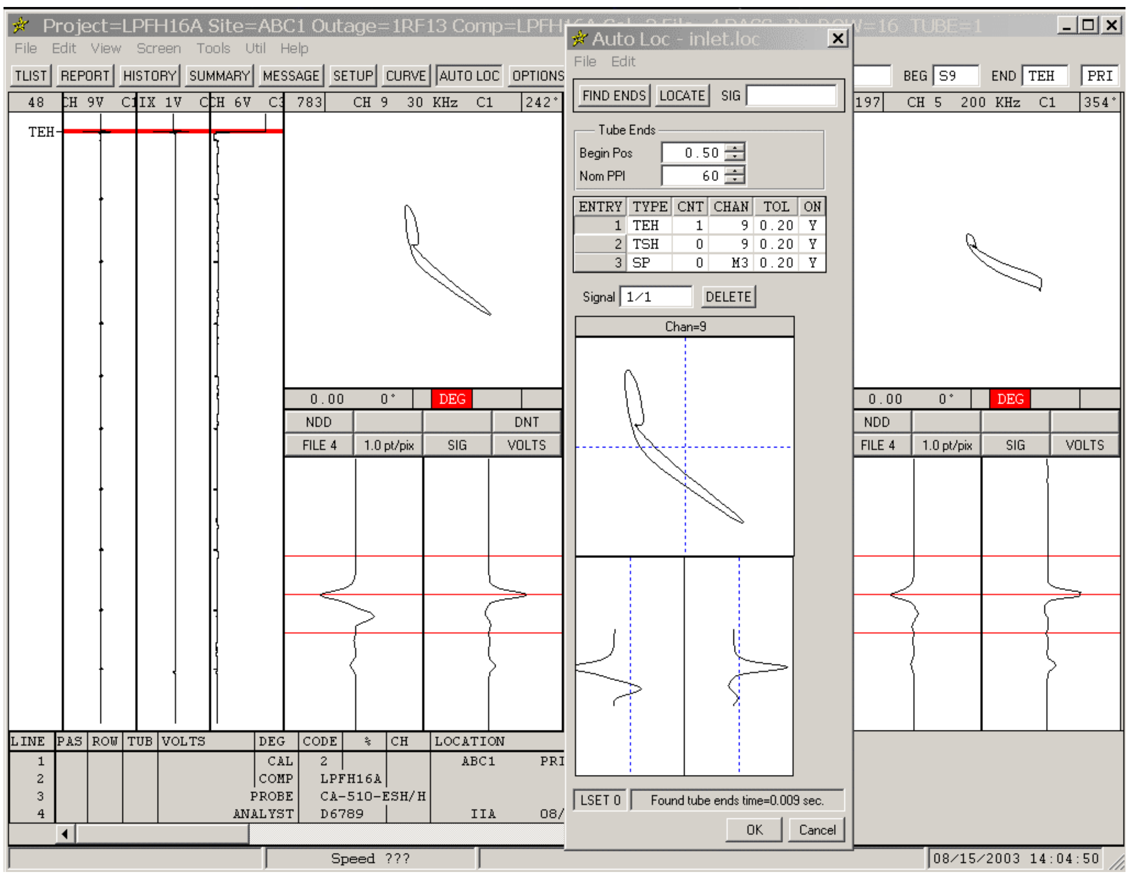


Figure 1-27. Auto-locate - Training the Tube End

HINT: If the tube end is not labeled by auto-locate in the step above, check the ***Summary*** to ensure that ***INLET*** is selected in the ***Leg*** field.

STEP 12: In the long strip chart, position the cursor at the top-of-tubesheet signal. The top-of-tubesheet signal is the second medium amplitude signal just below the tube end signal.

STEP 13: Adjust the ***Span*** and expanded window of ***CH 9*** so that the top-of-tubesheet signal is clearly visible in the Lissajous. It is very important to trim the signal down as much as possible.

STEP 14: Click once in the ***CNT*** (*count*) field for ***TSH***. The number ***1*** appears.

STEP 15: Click the ***LOCATE*** button. As shown in Figure 1-28, both ***TEH*** and ***TSH*** should appear in the ***Landmark*** strip at the correct locations in the data file.

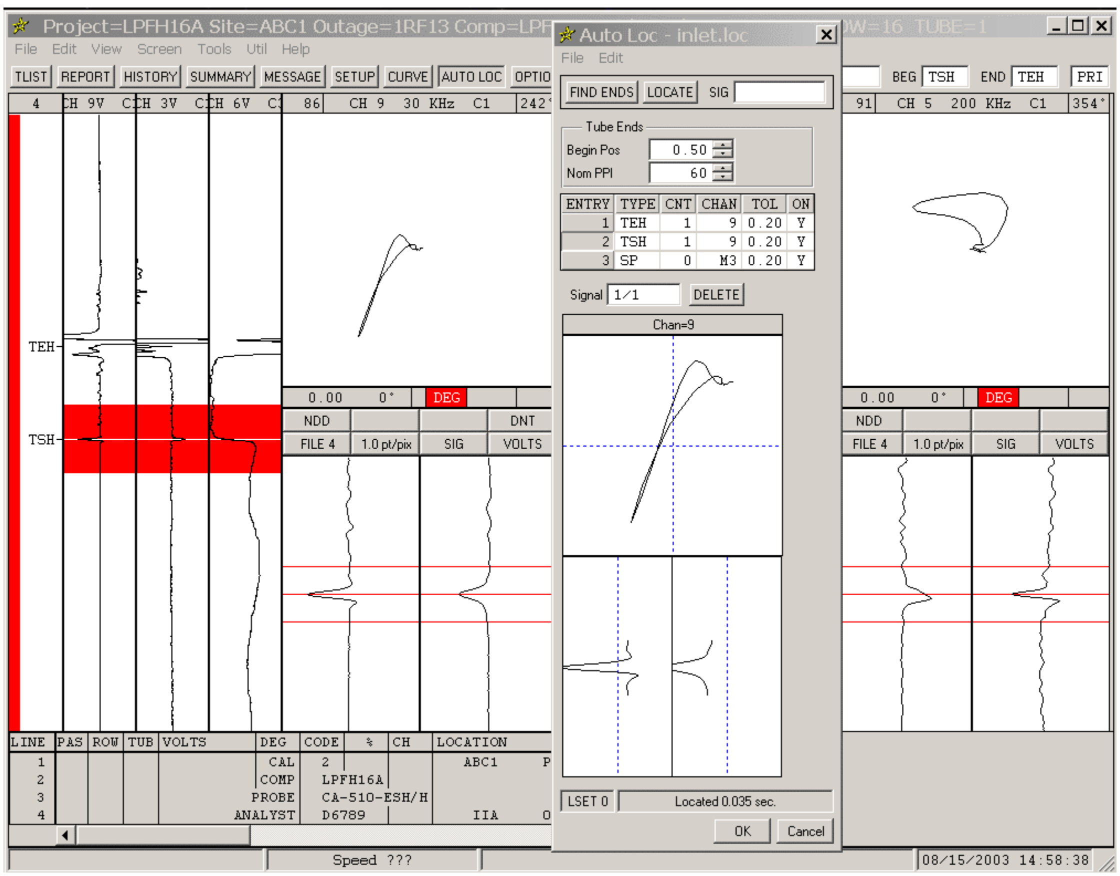


Figure 1-28. Auto-locate - Training the Tubesheet Interface

STEP 16: In the long strip chart, position the cursor at the first tube support signal from the top-of-tubesheet.

STEP 17: Set ***Mix 3*** in the left Lissajous.

STEP 18: Adjust the ***Span*** and expanded window of ***Mix 3*** so that the tube support signal is clearly visible in the Lissajous.

STEP 19: Click once in the ***CNT*** (*count*) field for ***SP***. The number ***1*** appears.

STEP 20: Click the ***LOCATE*** button. As shown in Figure 1-29, all, but possibly 2 of the tube supports, should appear in the ***Landmark*** strip at the correct locations in the data file.

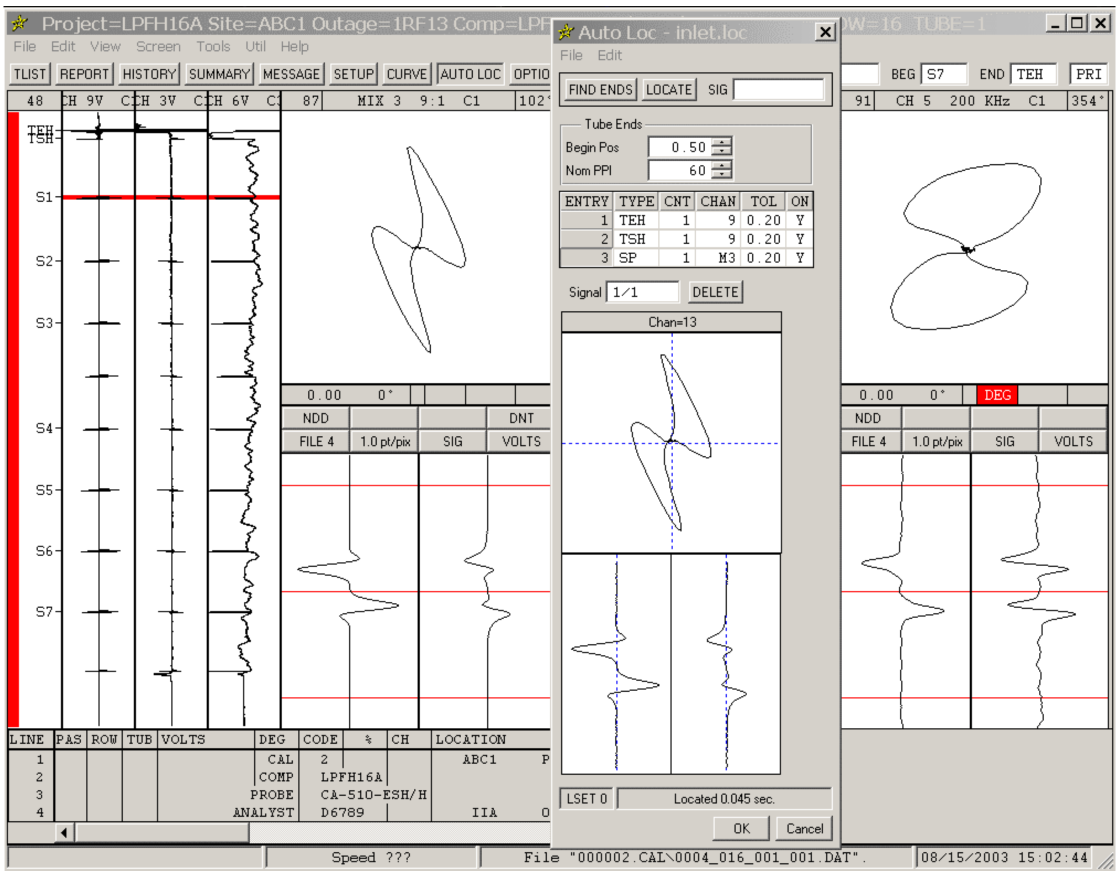


Figure 1-29. Auto-locate - Training Tube Support Structures

STEP 21: In the long strip chart, position the cursor at the first tube support signal that did not landmark.

STEP 22: Click once in the ***CNT*** (*count*) field for ***SP***. The number ***2*** appears.

STEP 23: Click the ***LOCATE*** button. The tube end, the top-of-tubesheet, and all tube supports should now appear in the ***Landmark*** strip at the correct locations in the data file as shown in Figure 1-30.

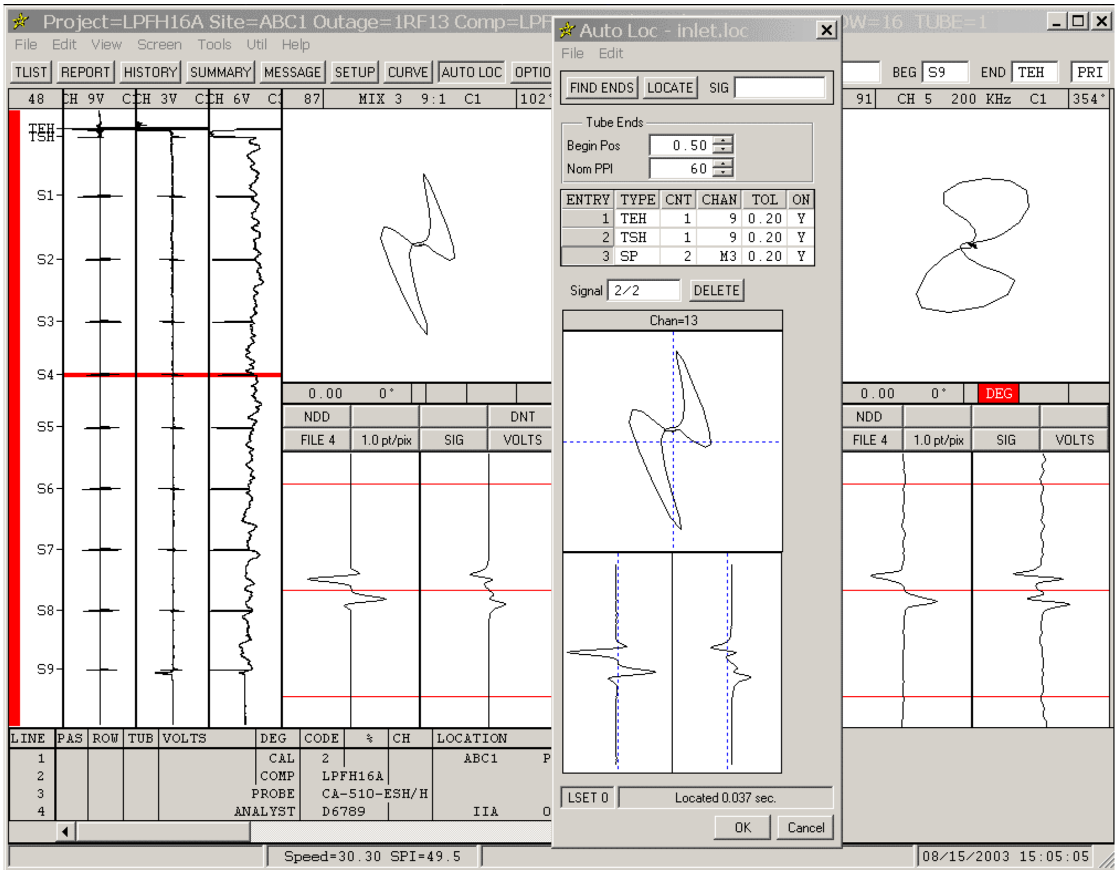


Figure 1-30. Auto-locate - Training More Tube Support Structures

STEP 24: Click the ***File*** button in any Lissajous. ***File 5*** is displayed in the button. The header should show tube ***IN-16-2*** and the data file should be loaded.

STEP 25: In the long strip chart, position the cursor at a tube support signal that did not landmark in this data file.

STEP 26: Click once in the ***CNT*** (*count*) field for ***SP***. The number ***3*** appears.

STEP 27: Click the ***LOCATE*** button.

STEP 28: Continue *training* auto-locate by repeating the preceding 4 steps until all landmarks are labeled correctly.

STEP 29: Click the ***File*** button to play in the next data file (***IN-16-3***).

STEP 30: Repeat the *training* steps above as necessary until you can click the ***File*** button and observe that all the landmarks are labeling correctly.

STEP 31: Once the auto-locate is working to your satisfaction, select ***File | Save As***.

STEP 32: Enter a filename of ***inlet*** and click ***Save***.

STEP 33: Click ***OK*** on the ***Auto Loc*** dialog.

NOTE: Please see ***Appendix D*** for more information about ***Auto-locate***.

Let's analyze a few tubes and make some report entries:

STEP 1: Click the ***TLIST*** button and load the first tube in the ***TLIST*** (*IN-16-1*).

STEP 2: Ensure that the landmarks *hit* correctly.

STEP 3: Click at the bottom of the long chart (below S9).

STEP 4: Under any Lissajous, begin clicking the ***SIG*** (*signal*) button. The cursor will jump to the next landmark on each click. This is a handy way of screening the landmarks for flaw indications.

HINT: Right-click the ***SIG*** button to move to the previous (*lower*) landmark.

STEP 5: Return to ***S8***. There's a small dent signal that we need to call.

STEP 6: Center and trim down on the dent signal so that it's clearly visible in the ***Mix 1*** Lissajous. The location box (*upper right corner*) should display ***S8-0.62*** or similar.

STEP 7: Shift+click or Shift+right-click on one of the blank macro buttons in the ***Mix 1*** Lissajous until ***DNT*** appears. These buttons are located to the right of the *fixed* ***NDD*** button.

STEP 8: Click in the ***Mix 1*** Lissajous and measure the dent signal ***Vpp***.

STEP 9: Click the newly assigned ***DNT*** macro button in the ***Mix 1*** Lissajous. As shown in Figure 1-31, the *call* is made and added to the ***Report*** at the bottom of the screen.

HINT: To use desired defect codes, they must exist in the ***Defect List***. Select ***Edit | Defect List*** to add or modify defect codes.

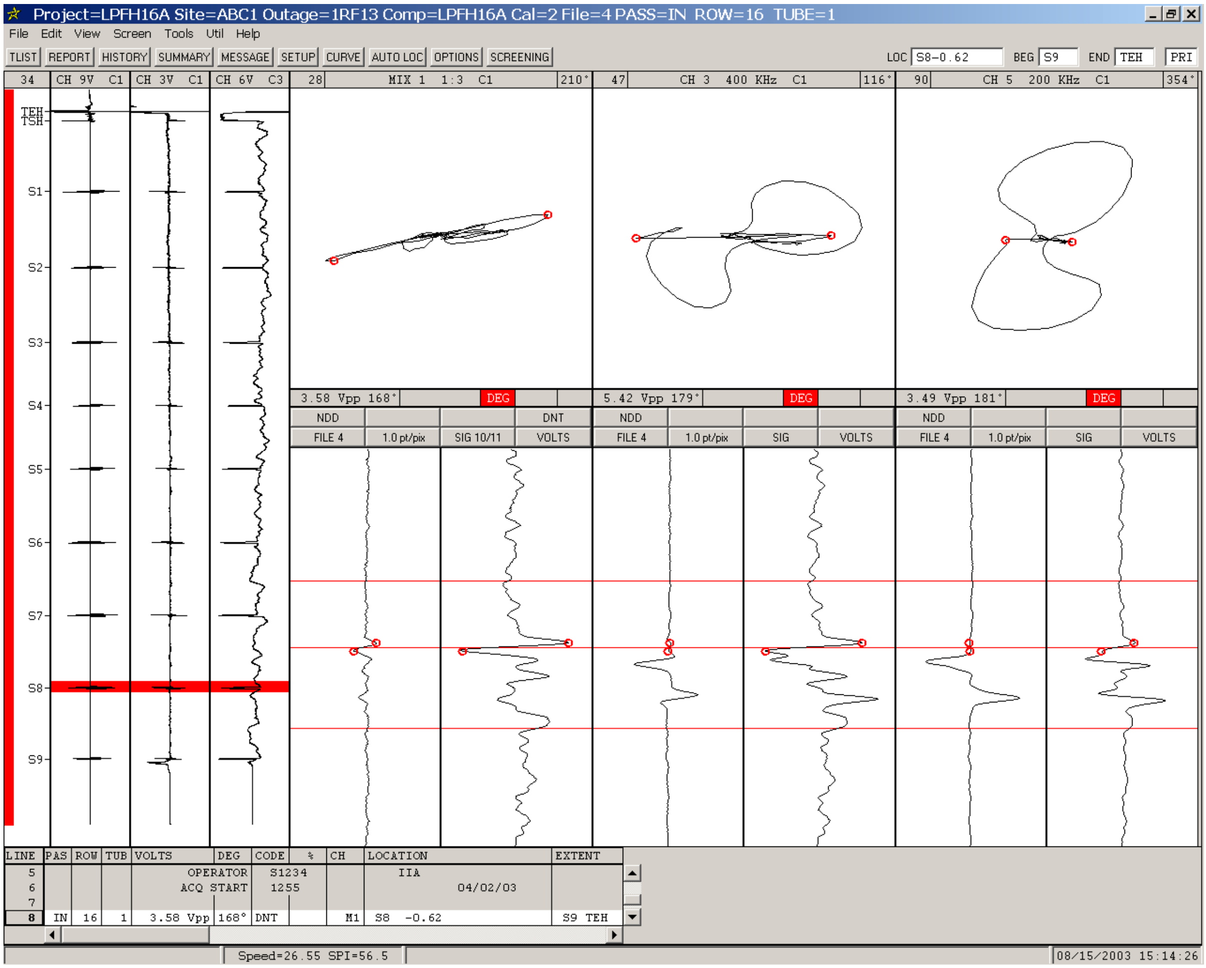


Figure 1-31. Analysis Screen - Making Calls Using Macro Buttons

STEP 10: To analyze the remainder of the tube, especially the free-span regions, you may:

- Click & drag the cursor along the long strip chart,
- press the ***Up*** or ***Down*** arrow keys on your keyboard, or
- zoom in, and then use the ***Page Up*** & ***Page Down*** keys on your keyboard.

See ***Util | Dispay Util*** for more ways to view or screen data files.

STEP 11: We're done with the first tube. Click any ***File*** button to load the next data file (*IN-16-2*).

STEP 12: Ensure that the landmarks *hit* correctly.

STEP 13: There are a few small dings in this tube as well, but let's simply make an ***NDD*** call. Click the ***NDD*** button below any Lissajous once. Note that an ***NDD*** call was added to the ***Report*** and the next data file was loaded.

STEP 14: Open the ***TLIST*** and load ***Entry 18*** (*IN-16-19*).

NOTES:	• Notice in the ***TLIST*** under the ***DUPL*** (*duplicate*) column that ***Entry 18*** is the 3rd recording of the 3rd attempt, i.e., ***3/3***. • The ***QA*** column shows ***1*** for the first two tubes. This means that each of these tubes have one call in the ***Report***. If this field is empty, it means that the data file has not yet been analyzed.

STEP 15: If the landmarks didn't hit, neither did mine. This is due to the partial tube end signal for this data file. Simply trim down on the tube end signal, open the ***Auto Loc*** dialog, add another ***CNT*** for ***TEH***, and click ***LOCATE***.

STEP 16: Click any ***File*** button until *77* appears. When this data file loaded, the ***History*** dialog should have appeared as shown in Figure 1-32. If the location is accurate in the ***History*** dialog and the landmarks *hit*, you can double-click the entry and the software will place the cursor at the historical location.

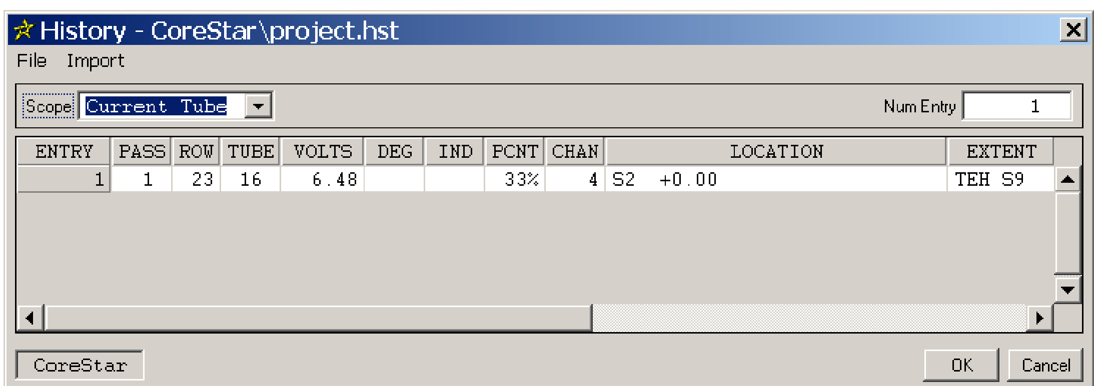


Figure 1-32. History Dialog

STEP 17: Center and trim down on the absolute signal (*tube-to-tube wear*) between ***S1*** and ***S2***. Use a Lissajous with ***CH 6*** selected.

STEP 18: Click the ***pt/pix*** button under the Lissajous in order to compress the signal so that it will clearly fit inside the expanded chart. (*Hint: 3.0 pt/pix*)

STEP 19: Measure the signal in a ***CH 6*** Lissajous using ***Vvm*** (~22%).

This time I'll show you a different way to make a call, but you could easily setup a macro button for this defect code as well.

STEP 20: Right-click in the box to the immediate right of the percent measurement in the ***CH 6*** Lissajous' measurement grid.

STEP 21: As shown in Figure 1-33, select ***ODW*** from the popup list.

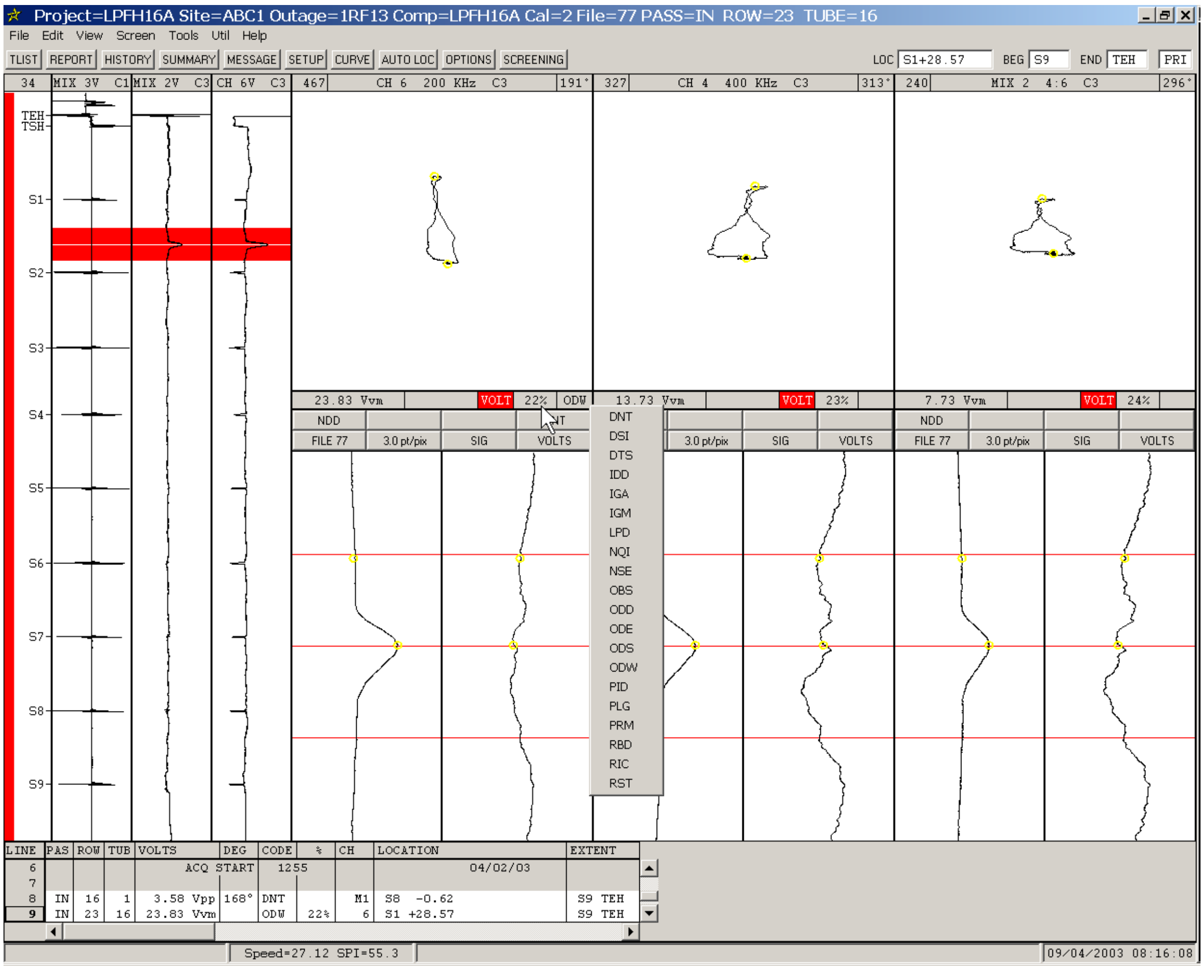


Figure 1-33. Selecting a Defect Code From the Popup List & Making a Call

STEP 22: Click once on the percent value in the ***CH 6*** Lissajous' measurement grid. The call is added to the ***Report***.

STEP 23: Middle-click the ***pt/pix*** button under any Lissajous in order to return to the default *non-compressed* setting of ***1.0***.

TIP:	Alternatively, middle-click the box to the right of the percent box and type in the defect code, and then click the percent box to add the call to the report. The defect code must exist in the ***Defect List***.

STEP 24: Load the ending cal run from the ***TLIST*** (*Entry 155*).

STEP 25: In the *on-screen* ***Report*** at the bottom of the analysis screen, click on the bottom dark gray row. A blue cursor appears.

STEP 26: Right-click the blue cursor and select ***End Cal*** from the popup menu. An embedded automatic footer is added to the bottom of the report.

STEP 27: Click the ***Report*** button. The ***Report*** dialog appears.

STEP 28: Click on the first report entry for ***IN-16-1***.

STEP 29: Scroll to the right slightly until the column labeled ***Analyst*** is visible.

STEP 30: Right-click on the ***Analyst*** column header. As shown in Figure 1-34, a ***Report Update*** dialog appears. This dialog is used to make global changes to non-text fields in the report. The ***Outage*** and ***Probe SN*** fields are the only ones that can be edited manually. All the other fields are controlled by ***Lookup*** tables or ***Summary*** entries.

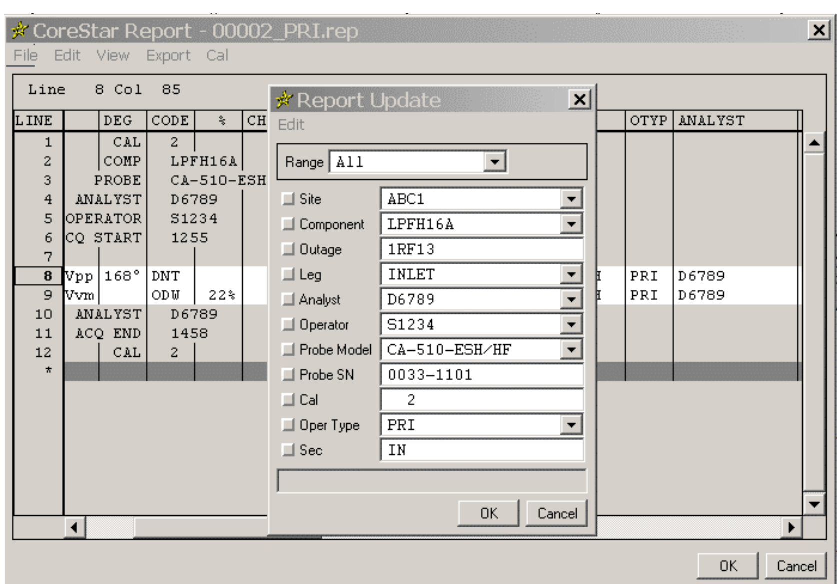


Figure 1-34. Modifying Report Entries Using the Report Update Dialog

STEP 31: Click ***Cancel*** on the ***Report Update*** dialog. No edits are required.

STEP 32: Right-click on the ***Extent*** column header. As shown in Figure 1-35, a ***Global*** edit dialog appears. This dialog is used to make global changes to text fields in the report.

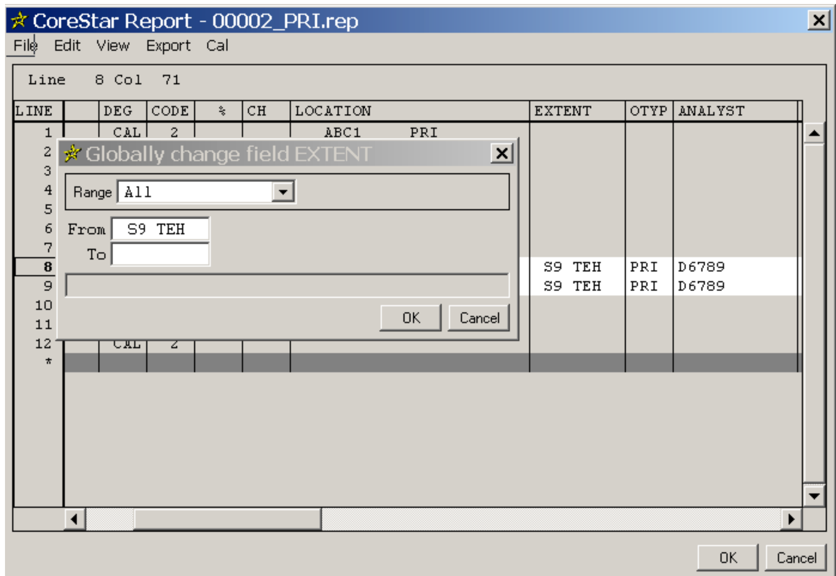


Figure 1-35. Globally Editing Text Values in a Report Using the Global Edit Dialog

STEP 33: Click ***Cancel*** on the ***Global*** edit dialog. No edits are required.

The preceding steps showed two different editing dialogs available to you. Try to avoid manual edits to the report as it only accepts certain data-types in certain fields. Using the editing dialogs above will ensure consistent report entries – especially for uploading to the database in the next topic.

STEP 34: After reviewing, editing, and ensuring our report is correct and ready for data management, select ***Cal | Store Report to Cal***. If you get a message box asking you if you want to overwriting the existing report, click ***Yes***. Our report is now saved in ***`\ecdata\000002.CAL\00002_PRI.rep`***.

STEP 35: Click ***OK*** on the ***Report*** dialog.

STEP 36: Open the ***TLIST***.

STEP 37: Select either ***File | Close Cal*** or ***Cal Closed*** in the ***CAL*** list box.

STEP 38: Click ***OK*** on the ***TLIST*** dialog.

NOTES:

- In the ***Report***, double-click the ***Line*** number to the left of any entry to recall the eddy current call.
- A *working* copy of all analysis reports exists on each work station were the analysis was performed. This is in addition to the report manually stored to the cal group.
- To load or *retrieve* a report from the currently open cal group, select ***Cal | Load Report from Cal*** on the ***Report*** dialog.
- To print a screen dump during analysis, simply select ***File | Print Screen*** or press ***Ctrl+P*** on the keyboard.
- Screen dumps are printed based on the settings in the ***Print*** (*tab*) dialog under the ***Options*** (*button*).

15. Uploading Reports to the Database

This topic will provide instructions for uploading analysis reports to the ***DBMS*** software. The first thing we need to do is start the ***DBMS*** (*database management software*) module and open our sample project as follows:

STEP 1: Double-click the ***DBMS*** icon on the desktop. The ***DBMS*** software will start.

STEP 2: The ***LPFH16A*** project should open. If not, select ***File | Open Project***, select ***c:\Projects\LPFH16A***, and click ***Open.***

During a typical inspection, the analysis reports will become available as they are completed by the analyst(s). Report files (***rep)*** are normally stored in each specific cal group directory where the eddy current data resides and was analyzed from. In our example, however, all the reports have been completed and are available for upload all at once from the reports directory of this project.

To upload the eddy current analysis report files:

STEP 1: Select ***Database | Upload Reports***. The ***Import Report*** dialog appears.

STEP 2: Ensure that ***CoreStar*** is selected in the ***Format*** list box.

STEP 3: If necessary, toggle the button to the immediate right of the ***Format*** list box so that it turns green and ***Binary Mode*** is displayed.

STEP 4: Select ***File | Open***.

STEP 5: In the ***Open*** dialog, click on the first report file, i.e., ***`00001_SEC.rep`***.

STEP 6: Shift+click on the last report file, i.e., ***`00010_SEC.rep`***. All 10 report files should now be highlighted.

STEP 7: Click the ***Open*** button. All the report files now appear in the upper section of the ***Import Report*** dialog.

STEP 8: Click on the first report, i.e., ***`00001_SEC.rep`***. The contents of the report are displayed in the lower section of the ***Import Report*** dialog.

STEP 9: Click ***Upload*** in the upper right corner of the ***Import Report*** dialog. As shown in Figure 15-1, the word ***Done*** appears in the ***Status*** field for this report in the upper section. In addition, a message is displayed in the status bar at the bottom of the ***Import Report*** dialog indicating that the upload was successful.

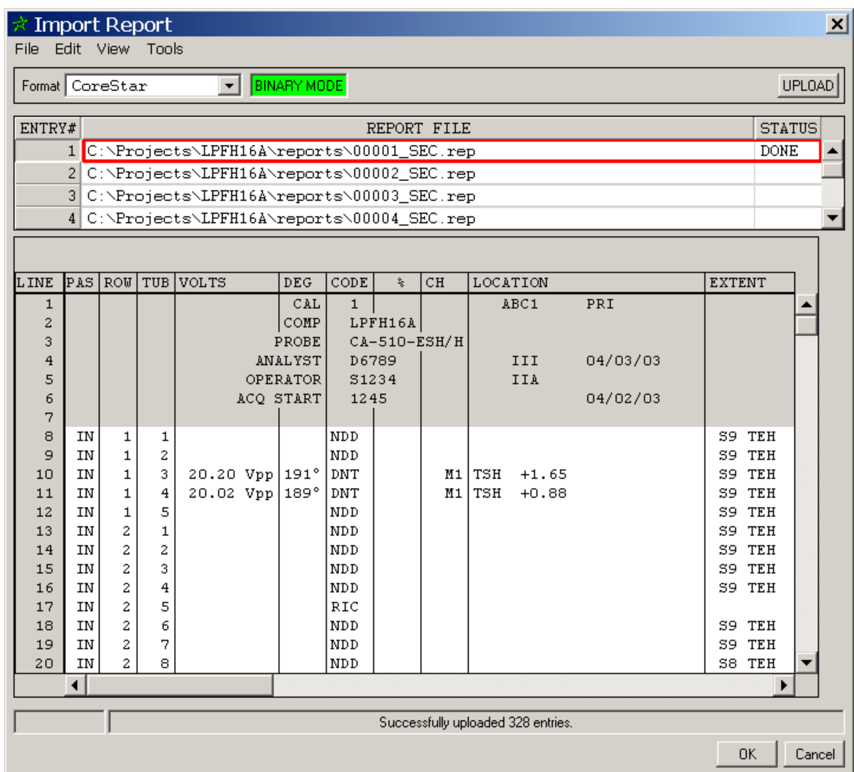


Figure 15-1. Import Report Dialog

STEP 10: Repeat the preceding step for the remaining 9 report files.

TIP:	Before uploading, reports should be carefully reviewed by the database administrator to ensure their accuracy and completeness. Inaccurate, incomplete, or questionable report entries should be brought to the attention of the responsible analyst for correction before uploading.

STEP 11: After all 10 reports have been uploaded, click ***OK*** on the ***Import Report*** dialog.

16. Plotting Results on a Tubesheet Map

This topic will provide instructions for querying the database for certain information that we want to plot on a tubesheet map. We will learn how to change symbols, colors, and descriptions of items plotted on the same map. Furthermore, we'll learn how to add and edit a map header, how to turn on and off available borders on the printed map, and how to adjust the margins of the printed map.

It's always good practice to plot the plugged tubes on all final maps. This provides quick reference points on the tubesheet map and also aids you and the customer in identifying problem areas in a tube bundle, i.e., new damage appearing near existing plugged tubes.

To query for and plot the plugged tubes in our sample project:

STEP 1: Double-click the ***DBMS*** icon on the desktop. The ***DBMS*** software will start.

STEP 2: The ***LPFH16A*** project should open. If not, select ***File | Open Project***, select ***`c:\Projects\LPFH16A`***, and click ***Open.***

STEP 3: Select ***File | New Map***. Click ***No*** if the '*Do you wish to save changes to the map?*' question box appears.

STEP 4: Click the ***Query*** button is the upper right of the ***DBMS*** window. The ***Query*** dialog appears.

STEP 5: Select ***File | Open***.

STEP 6: Open the file named: ***\queries\EXISTING PLUGS FROM REPAIRS TABLE.qry.*** If this query file is not available, simply type the following text into the query dialog exactly as it appears:

```
SELECT sec, row, col
FROM repairs
WHERE repair_type = 'PLUG'
ORDER BY sec, row, col;
```

STEP 7: Click the ***Execute*** button in the upper left of the ***Query*** dialog. As shown in Figure 16-1, a list of *206 entries* should appear in the lower section of the ***Query*** dialog.

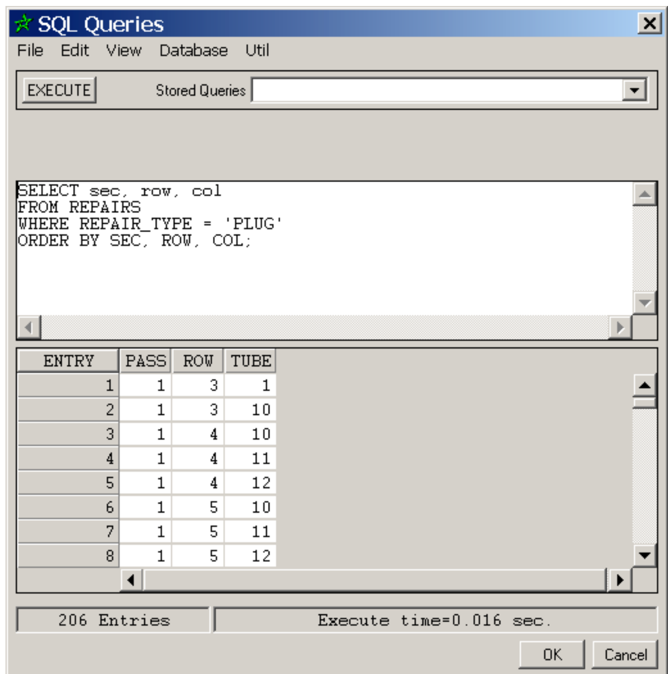


Figure 16-1. Executing a Query in the Query Dialog

STEP 8: Click ***OK***. As shown in Figure 16-2, an entry should appear in the legend in the upper right area of the ***DBMS*** window. The entry should show ***206*** in the ***Tubes*** column.

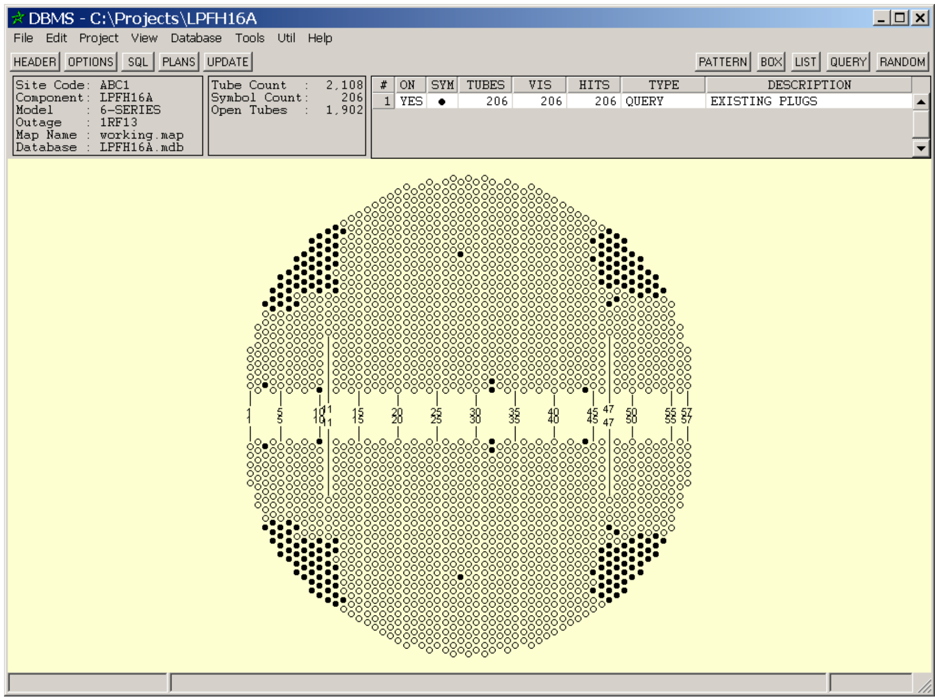


Figure 16-2. Results of a Query Plotted on a Tubesheet Map

OK, so now you've opened, ran (*executed*), and plotted your first query. As shown in Figure 16-2, let's change the symbol to a filled black circle as follows:

STEP 1: Right-click the symbol for the plugged tubes' entry under the ***SYM*** column in the legend. As shown in Figure 16-3, the ***Choose Symbol*** dialog appears.

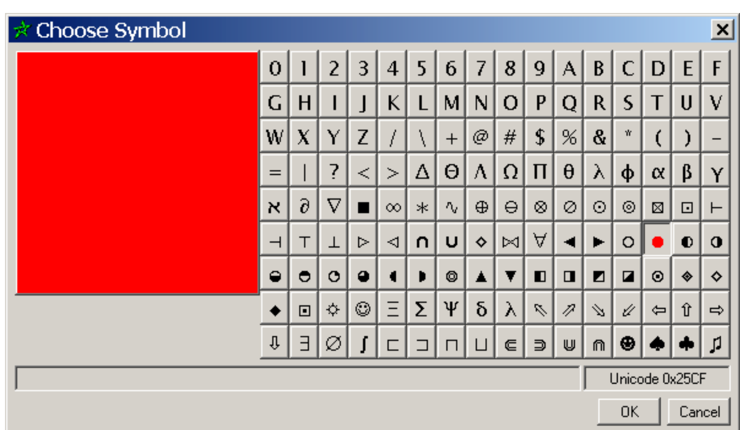


Figure 16-3. Choose Symbol Dialog

STEP 2: Click on the ● symbol.

STEP 3: Right-click on the large colored area on the left side of the ***Choose Symbol*** dialog. As shown in Figure 16-4, the ***Color*** dialog appears.

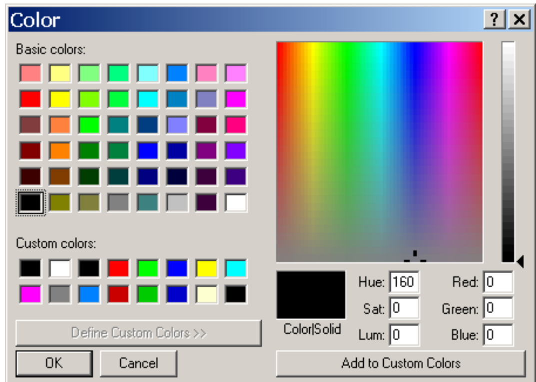


Figure 16-4. Symbol Color Dialog

STEP 4: Click the color ***black***, and then click ***OK***.

STEP 5: Click ***OK*** on the ***Choose Symbol*** dialog. The symbol for the plugged tubes' entry in the legend should now be a filled black circle (●).

To change a description for a legend entry:

STEP 1: Click in the ***Description*** field for the plugged tubes' entry. All the existing text (*if any*) is highlighted.

STEP 2: Simply type-in the desired description. For our example, enter ***PLUGGED PRIOR TO 1RF13*** as shown in Figure 16-5.

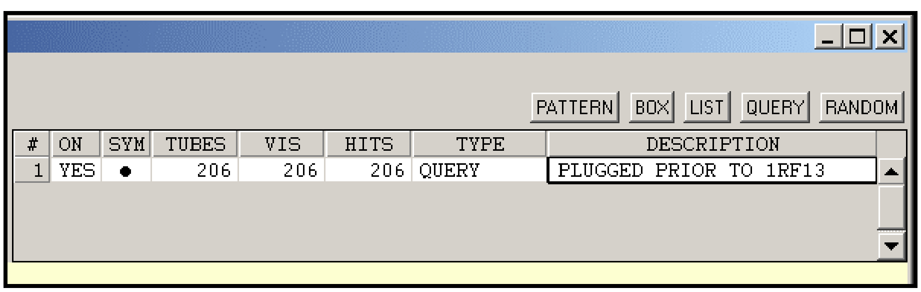


Figure 16-5. Changing the Description for an Entry in the Legend

Let's query for the tubes that had percent calls ≥50% during this outage and plot these on our map:

STEP 1: Click the ***Query*** button in the upper right of the ***DBMS*** window. The ***Query*** dialog appears.

STEP 2: Select ***File* | *Open***.

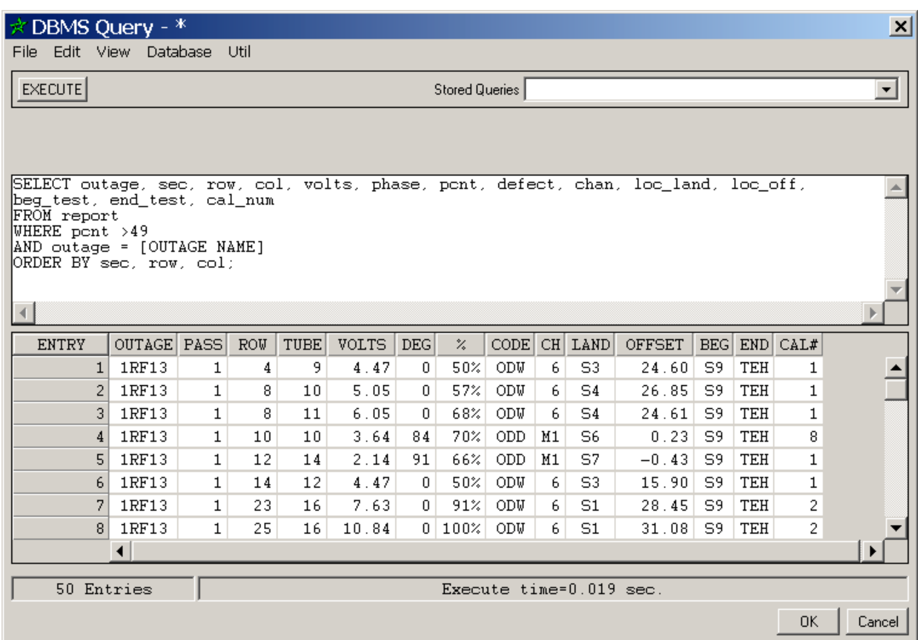
STEP 3: Open the file named: ***\queries\FINAL 50-100 CALLS.qry***. Ensure that the query is identical to the one shown below – especially for ***outage*** in the ***Where*** statement. If this query file is not available or is not identical, simply type-in or edit as follows:

```
SELECT outage, sec, row, col, volts, phase, pcnt, defect,
chan, loc_land, loc_off, beg_test, end_test, cal_num
FROM report
WHERE pcnt >49
AND outage = [OUTAGE NAME]
ORDER BY sec, row, col;
```

STEP 4: Click the ***Execute*** button in the upper left of the ***Query*** dialog.

STEP 5: Click ***OK*** on the ***Parameters*** dialog that appears.

STEP 6: As shown in Figure 16-6, a list of *50 entries* should appear in the lower section of the ***Query*** dialog.



ENTRY	OUTAGE	PASS	ROW	TUBE	VOLTS	DEG	%	CODE	CH	LAND	OFFSET	BEG	END	CAL#
1	1RF13	1	4	9	4.47	0	50%	ODW	6	S3	24.60	S9	TEH	1
2	1RF13	1	8	10	5.05	0	57%	ODW	6	S4	26.85	S9	TEH	1
3	1RF13	1	8	11	6.05	0	68%	ODW	6	S4	24.61	S9	TEH	1
4	1RF13	1	10	10	3.64	84	70%	ODD	M1	S6	0.23	S9	TEH	8
5	1RF13	1	12	14	2.14	91	66%	ODD	M1	S7	-0.43	S9	TEH	1
6	1RF13	1	14	12	4.47	0	50%	ODW	6	S3	15.90	S9	TEH	1
7	1RF13	1	23	16	7.63	0	91%	ODW	6	S1	28.45	S9	TEH	2
8	1RF13	1	25	16	10.84	0	100%	ODW	6	S1	31.08	S9	TEH	2

Figure 16-6. Executing a Query That Returns Specific Inspection Results

STEP 7: Click ***OK***. An entry should appear in the legend in the upper right area of the ***DBMS*** window. The entry should show ***40*** in the ***Tubes*** column and ***50*** in the ***Hits*** (*records*) column.

HINT:	Tubes can contain multiple calls.

STEP 8: Repeat the preceding steps for the plugged tubes' plot as required to change the symbol for this legend entry to a red filled square (■).

STEP 9: For this legend entry, enter a ***Description*** of ***50 TO 100% INDICATIONS***.

<u>The plotting priority of multiple symbols on a map is from bottom to top</u>. Let's move the ***PLUGGED PRIOR TO 1RF13*** entry to the bottom of the legend as follows:

STEP 1: Ctrl+click and drag the entry number in the # column of the ***PLUGGED PRIOR TO 1RF13*** entry (*should be entry #1*) to just below the ***50 TO 100% INDICATIONS*** entry, and then release. The ***PLUGGED PRIOR TO 1RF13*** entry should now be located below the ***50 TO 100% INDICATIONS*** entry as shown in Figure 16-7.

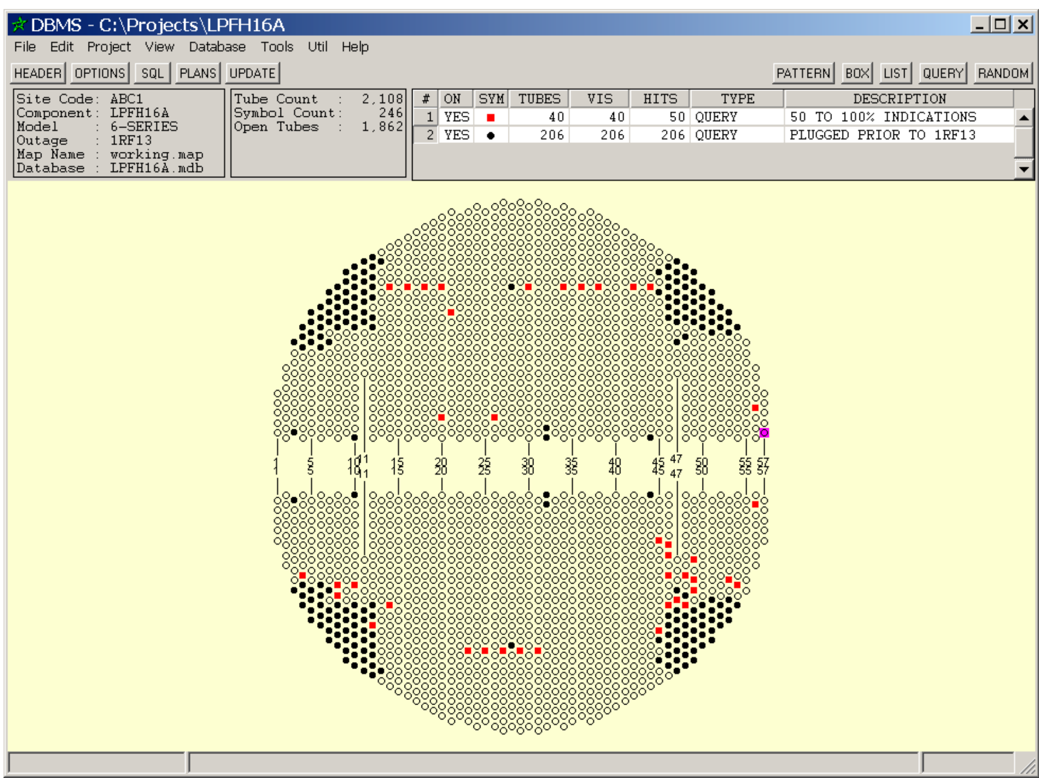


Figure 16-7. Moving and Rearranging Legend Entries

HINT:	You must *drop* the entry being moved just above or below the center of an entry at the new location. With a little trial and error, you'll see how it works.

Let's add a header and footer to our map:

STEP 1: Click the ***Header*** button in the upper left of the ***DBMS*** window. The ***Map Header*** dialog appears. The upper section is for the ***Header*** and the lower section is for the ***Footer***.

STEP 2: In the ***Font 1*** and ***Size*** list boxes, select ***Arial*** and ***16***, respectively.

STEP 3: In the ***Font 2*** and ***Size*** list boxes, select ***Times New Roman*** and ***12***, respectively.

STEP 4: Enter the following text in the ***Header*** area provided:

```
ABC NUCLEAR POWER STATION - UNIT #1 - 1RF13 (7-03)
50 TO 100% INDICATIONS
LOW PRESSURE FEEDWATER HEATER - 16A
MATERIAL: 304 STAINLESS STEEL
TUBE SIZE: 0.625-inch OD X 0.035-inch WALL
VIEW:  INLET-OUTLET TUBESHEET - FACING EAST
```

STEP 5: In the ***Footer Font*** and ***Size*** list boxes, select ***Arial*** and *7*, respectively.

STEP 6: Enter the following text in the ***Footer*** area provided:

```
CoreStar International Corp., DBMS Rev 5.0, [CUR DATETIME]
```

TIPS:

- Alternatively, you may select ***File | Open***, open an existing map header file, and edit it to match accordingly.
- **`[CUR DATETIME]`** shown in the ***Footer*** above is one of the many system macros available for your use in headers, footers, other macros, and queries. All macros are available for your selection from the ***Macros*** list box on the ***Map Header*** dialog or you can type them in; however, they <u>*must be*</u> all cap's and typed out exactly as required.
- Click and hold the ***Expand*** button to preview the value of all macros.
- Options for ***Save***, ***Save As***, ***Import***, and ***Export*** are available for ***Map Header*** files under the ***File*** menu as desired.

STEP 7: The ***Map Header*** dialog should look like Figure 16-8. Click ***OK***.

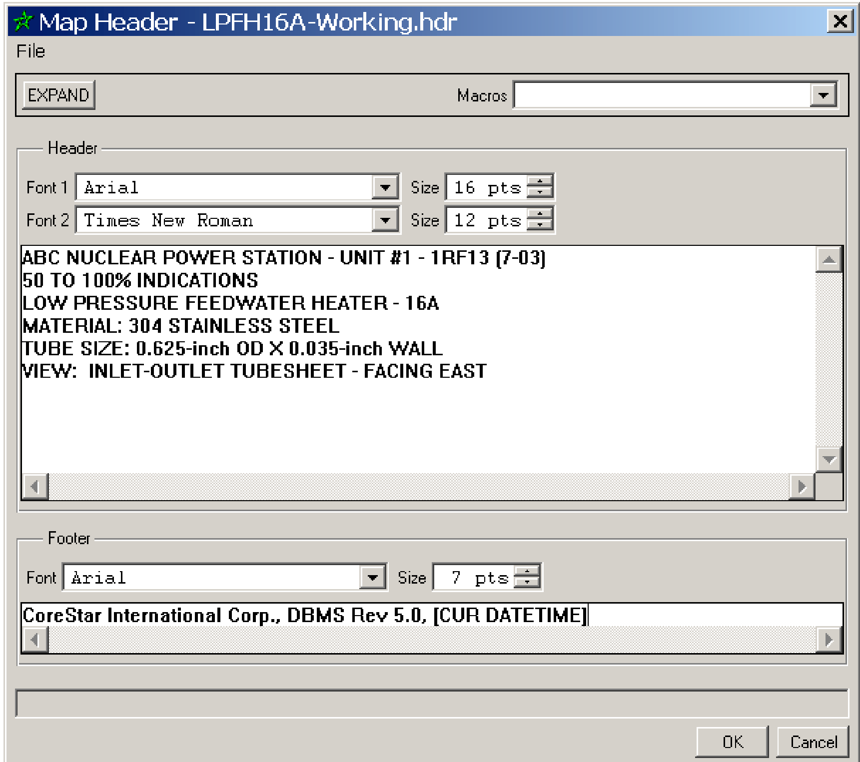


Figure 16-8. Map Header Dialog

We're close to printing a completed tubesheet plot, but let's check a few other options first:

STEP 1: Click the ***Options*** button in the upper left of the ***DBMS*** window. The ***Options*** dialog appears as shown partially in Figure 16-9.

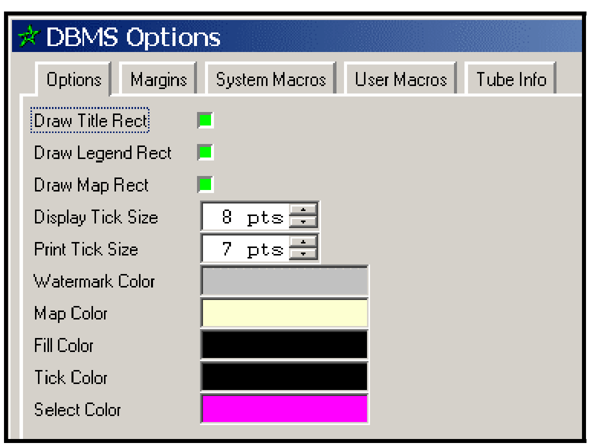


Figure 16-9. DBMS Options Dialog - Options Tab

STEP 2: Click the ***Options*** tab.

STEP 3: Enable ***Draw Title Rect*** (*border*), ***Draw Legend Rect***, and ***Draw Map Rect***.

STEP 4: Set ***Print Tick Size*** to *7*.

STEP 5: Right-click ***Watermark Color***, select a soft gray, and click ***OK***.

STEP 6: Click the ***Margins*** tab.

STEP 7: Enter ***0.50”*** in all 4 margin fields. You may enter the values from the keyboard or increment and decrement the values using click or right-click.

STEP 8: The ***Margins*** tab of the ***Options*** dialog should look like Figure 16-10. Click ***OK***.

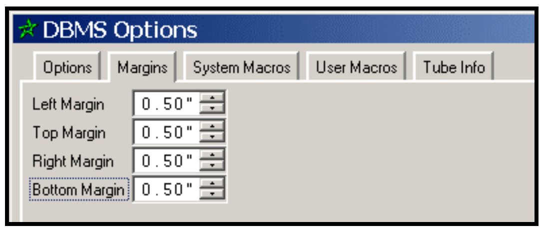


Figure 16-10. DBMS Options Dialog -Margins Tab

OK, we’ve spent some time creating a new tubesheet plot (*map*), let’s save the plot as a ***map*** file that we can open later, add more queries to, update the values of all the queries by clicking the ***Update*** button, print, etc. Think of ***map*** files as being the documents created using ***DBMS*** - much like Word is used to create ***doc*** files.

To save a plot as a ***map*** file:

STEP 1: Select ***File | Save Map As***.

STEP 2: In the ***Save As*** dialog, enter ***Working*** as the filename and click ***OK***. The current plot is now saved in the current project as:

c:\Projects\LPFH16A\maps\Working.map

TIPS:

- Map files are one of the single most powerful tools available in ***DBMS***. If you create your map files right, you can share them across all projects. The only edits you typically make is to the header.
- You may ***File | Import*** and/or ***File | Export*** map files from and to the current project if desired in order share map files with other projects.

Let's print our map. The following steps assume you have a printer connected to and appropriate print driver loaded on your computer.

First, to ensure our map is going to be what we expect and to save paper, let's preview it as follows:

STEP 1: Select ***File | Print Preview***. An approximated image of what the final printout will look like is displayed.

STEP 2: Click ***OK***.

> **NOTE:** If the ***Preview*** image doesn't display or the preview is not what you expect, select ***File | Print Setup***, select the desired printer, adjust the printer's properties as necessary, click ***OK*** – then try ***File | Print Preview*** again.

It's time to print our map as follows:

STEP 1: Select ***File | Print***. The typical Windows print dialog appears, which allows you make any final print property adjustments as desired.

STEP 2: Click ***Print***. The printer should print the map shortly.

If you upload additional eddy current analysis reports after making a map and desire to update the legend and re-execute all queries, simply:

STEP 1: Click the ***Update*** button along the upper left of the ***DBMS*** window.

There may be instances where you may desire to insert a copy of a map into a word processor such as a Word document for example. This function is really handy for sending a map to someone via email who doesn't have ***DBMS***.

To place a map image in a Word document:

STEP 1: Select ***File | Store Metafile***. A ***Save As*** dialog appears.

STEP 2: Enter a desired name for the file. In our example, enter **`test`**, and click ***OK***. The file is saved by default to the current project at:

`c:\Projects\LPFH16A\temp\test.emf`

STEP 3: Start a new document in Word.

STEP 4: In Word, select ***Insert* | *Picture* | *From File***.

STEP 5: In the ***Insert Picture*** dialog, select the file we just saved in ***DBMS***:

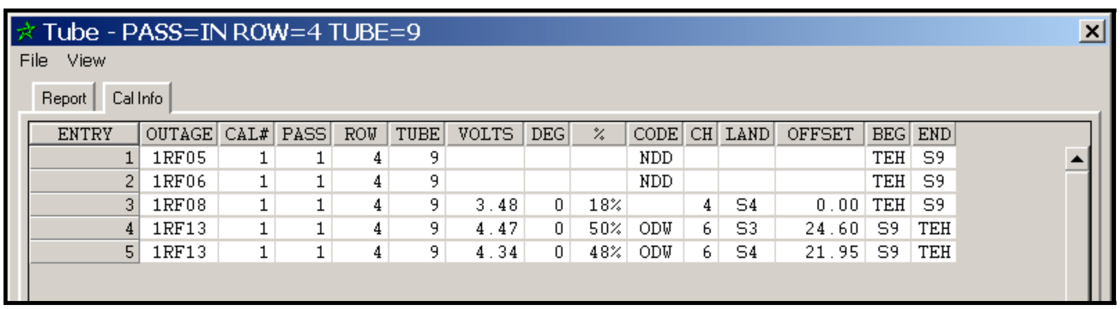
c:\Projects\LPFH16A\temp\test.emf

STEP 6: Click ***Insert***. An image of the plotted tubesheet map appears in the Word document.

TIP:	Alternatively, you may use ***File* | *Store Bitmap***; however, the quality of the ***bmp*** file is poor when compared to that of the ***emf*** (*enhanced metafile*) file.

Finally, to view the history for any tube:

STEP 1: Double-click the tube of interest. In our example, double-click tube number ***IN-4-9***. A ***Report*** dialog like the one partially shown in Figure 16-11 appears displaying all eddy current calls for this tube from the *report* table of the database.



ENTRY	OUTAGE	CAL#	PASS	ROW	TUBE	VOLTS	DEG	%	CODE	CH	LAND	OFFSET	BEG	END
1	1RF05	1	1	4	9				NDD				TEH	S9
2	1RF06	1	1	4	9				NDD				TEH	S9
3	1RF08	1	1	4	9	3.48	0	18%		4	S4	0.00	TEH	S9
4	1RF13	1	1	4	9	4.47	0	50%	ODW	6	S3	24.60	S9	TEH
5	1RF13	1	1	4	9	4.34	0	48%	ODW	6	S4	21.95	S9	TEH

Figure 16-11. Report Dialog Displayed By Double-clicking a Tube on a Tubesheet Map

NOTES:

- The default tab on the ***Report*** dialog is labeled ***Report***. The ***Cal Info*** tab was created by the author in order to view the history for a given tube in a user-defined order. You can add as many user-defined tabs to the ***Report*** dialog as desired. You can add new and edit existing tabs via the ***Tube Info*** tab of the ***Options*** dialog (*click the* ***Options*** *button to access*).
- You can delete records on any user-defined tab by clicking in any field of a record and pressing ***Shift+Delete*** on the keyboard. This is handy for 'cleaning up' the database at the end of an inspection.
- If a record is inadvertently deleted, click the ***Cancel*** button to undo.
- For additional querying tips, please see ***Appendix E***.

17. Printing Results

This topic will provide instructions for querying the database for certain information, defining a header & footer, and then printing a list of the results. Let's print a list of the percent calls ≥50% during this outage. We ran a query like this in the previous topic. You could click the ***Query*** button, open an existing query or type-in a new query; however, since the query we want has already been executed and plotted on our map, we can access it quicker right from the map legend.

To select and run an existing query from the map legend:

STEP 1: With ***DBMS*** running and displaying the map from the previous topic, right-click on the term ***Query*** in the ***Type*** column of the ***Legend*** for the ***50 TO 100% INDICATIONS*** entry. The ***Query*** dialog appears with the query already displayed that was used for percent calls ≥50% during this outage the on the map.

STEP 2: Click ***Execute***.

STEP 3: Click ***OK*** on the ***Parameters*** dialog if it appears. The retrieved records should now appear in the lower section of the ***Query*** dialog as shown previously in Figure 16-6.

Before we print the list, let's enter a page header and page footer that will print on each page of the list. Let's adjust the margins as well.

To enter a page header, a page footer, and adjust the margins:

STEP 1: On the ***Query*** dialog, select ***Edit | Print Options***. The ***Print Options*** dialog appears.

STEP 2: In the ***Page Header*** section, enter the following text using **Arial** font at **12** points and left justified:

CoreStar BOP Eddy Current Inspections
ABC Nuclear Station-Unit 1
1RF13-July 2003
Low-pressure Feedwater Heat Exchanger - 16A

TABLE II
Tubes Recording Indications 50 to 100%

Total Tubes: 40

STEP 3: Leave the ***Report Title*** and ***Report Header*** sections empty. The ***Headers*** tab should look like Figure 17-1.

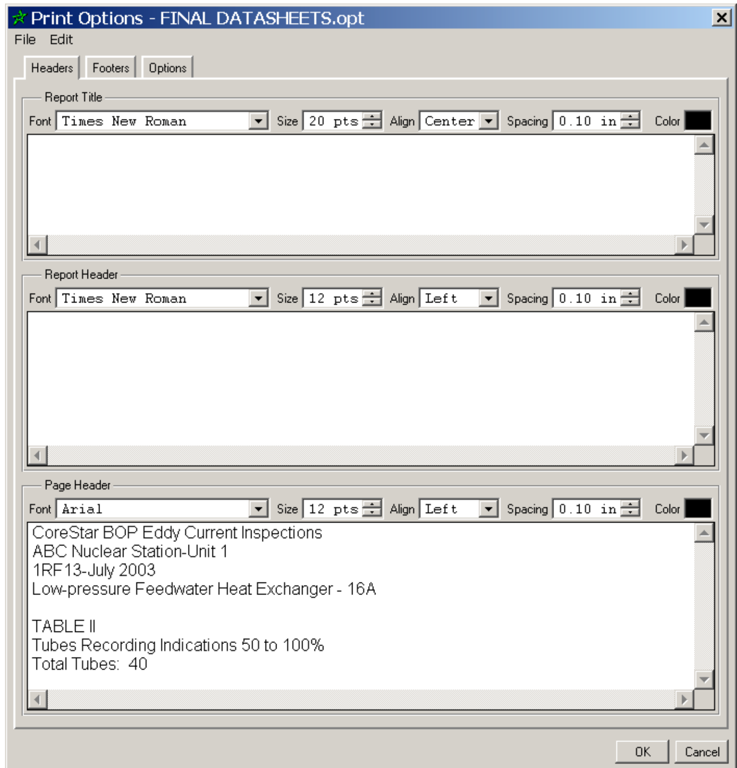


Figure 17-1. DBMS Print Options Dialog - Headers Tab

STEP 4: Referring to Figure 17-2, enter the following text using **Arial** font at **10** points and left justified into the ***Page Footer*** section:

PAGE [CUR_PAGE] OF [NUM_PAGE]

To allow a little space between the bottom of the list and the footer, you may want to enter a line feed (***Enter*** key) above the footer text.

STEP 5: Leave the ***Report Footer*** section empty.

STEP 6: Ensure that the ***Report Body*** section is set to **Courier New** font at **10** points and left justified.

STEP 7: In the ***Report Version*** section, enter the following text using **Times New Roman** font at **7** points and right justified:

CoreStar EddyVision 5.0

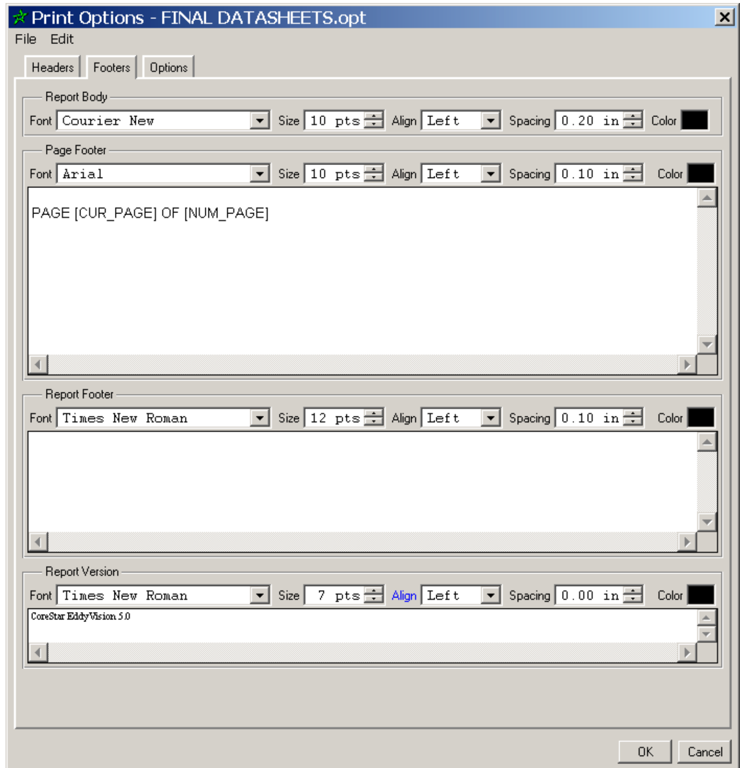


Figure 17-2. DBMS Print Options Dialog - Footers Tab

STEP 8: Click the ***Options*** tab and set the ***Margins*** to:

Left Margin: 0.75”
Top Margin: 0.50”
Right Margin: 0.50”
Bottom Margin: 0.25”

STEP 9: Enable the ***Draw Separators*** option. This will enable the borders and columns separators on our printout.

STEP 10: The ***Options*** tab of the ***Print Options*** dialog should look like Figure 17-3. Click ***OK***.

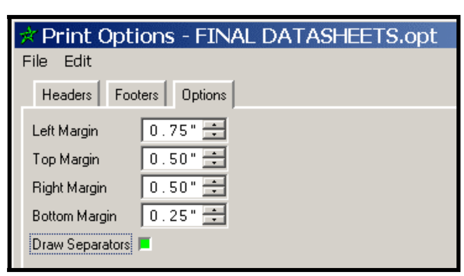


Figure 17-3. DBMS Print Options Dialog - Options Tab

We're now ready to print the list.

To print the results displayed in the ***Query*** dialog:

STEP 1: Select ***File | Print*** on the ***Query*** dialog. The typical Windows ***Print*** dialog appears.

STEP 2: Select the desired printer, adjust printer properties as desired, and click ***Print***. If the printed results are not as anticipated, re-adjust the ***Print Options*** as desired and print again.

NOTE:	Please see ***Appendix E: Understanding Queries*** for more information about SQL queries.

Appendix A: Understanding Pitch and Pitch Angles

The ***Tube Pitch*** required by the ***MakeComp*** software is the horizontal center-to-center tube pitch along any row. For square-pitch and equilateral tri-pitch, the tube pitch is the same horizontally or vertically as shown in Figure A-1.

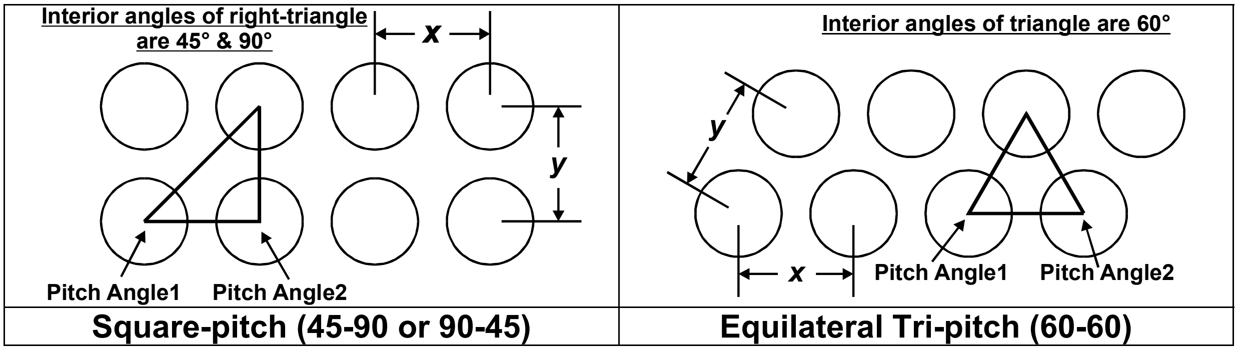
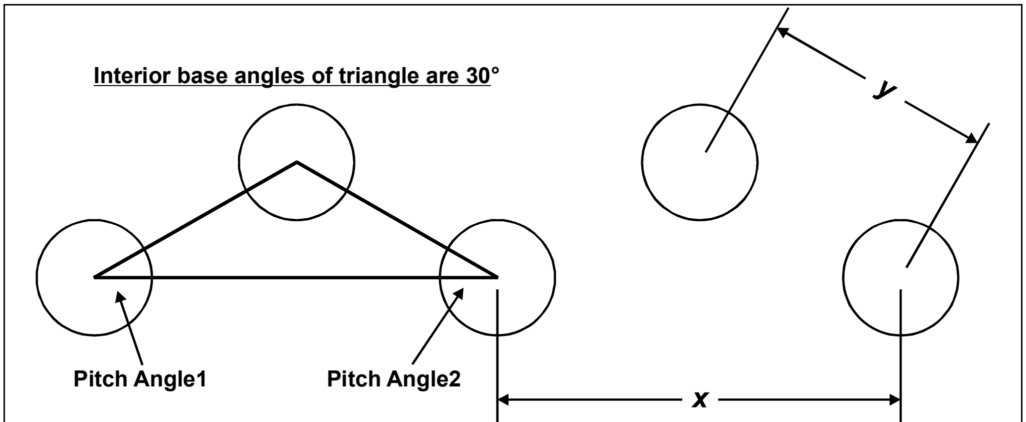


Figure A-1. Equilateral Tube Pitch Designs

The tube pitch provided on most specification drawings is not the horizontal tube pitch (x), but rather the tube pitch (y) along either side of a triangle created by selecting a tube in any row and drawing vectors down to the 2 adjacent tubes directly below it. In the first graphic of Figure A-1, this action creates a right-triangle with either leg being equal ($x = y$). In the second graphic, an equilateral triangle is created with all sides being equal ($x = y$). All the interior angles of the equilateral triangle are 60° while the interior angles of the right-triangle are 45°, 45°, and 90°. In both of these cases, the horizontal tube pitch (x) is the same as the vertical tube pitch (y) as shown.

However, a common tri-pitch design used in some heat exchangers is referred to as '*30-30*' tri-pitch. In this case, the resulting triangle is not equilateral and the interior angles are 30°, 30°, and 120° as shown in Figure A-2. The *y* dimension shown along one of the legs of the triangle is typically the tube pitch value given on manufacturer's specification drawings. MakeComp needs the horizontal tube pitch (*x*) to accurately calculate the position of the tubes on the *building grid*.



Tri-pitch (30-30)

Figure A-2. Non-equilateral Tube Pitch Design

To calculate the horizontal tube pitch for the 30-30 tri-pitch layout shown in Figure A-2 when only the *y* dimension is provided, use the following equation:

$$Cos30° \times y \times 2 = x$$

where,

x is the horizontal tube pitch;
y is the slanted or offset tube pitch given on the specification drawing; and,
Cos30° is the cosine of 30° and is a constant value of 0.866;

If *y* equals 0.938-inch (*15/16-inch – a common pitch for many heat exchangers*) for the example shown in Figure A-2, we get the following result:

$$0.866 \times 0.938 \times 2 = 1.625$$

Therefore, the horizontal tube pitch (*x*) would equal 1.625-inches and this value would be entered into the ***Tube Pitch*** field in the ***Group Parameters*** dialog.

Pitch Angle1 and ***Pitch Angle2*** are simply the left and right interior base angles (respectively) of the triangle created as shown in Figure A-1 or Figure A-2. For square-pitch and equilateral tri-pitch, the ***Pitch Angles*** are 45°/90° or 90°/45° for square-pitch depending how the triangle is drawn and 60° for equilateral tri-pitch

For asymmetrical tri-pitch (Figure A-2), enter the actual base angles. In the example shown in Figure A-2, values of 30° would be entered in these fields. By entering accurate values in these fields and the *Tube Pitch* field, any conceivable tubesheet map can be created since these 3 values are used to accurately calculate the position of the tubes and their relationship to each other.

TIPS:	• ***30-30*** pitch is simply ***60-60*** pitch rotated 90°. • When the pitch for a ***60-60*** tubesheet isn't readily available, simply add 0.188-inch (*3/16-inch – a common ligament value in many heat exchangers*) to one tube diameter and this sum as the ***Tube Pitch*** value in ***MakeComp***.

Appendix B: More About Acquisition and Analysis

Mouse and Keyboard Shortcuts

- ***Right-click*** in any Lissajous window or press the ***Space*** bar to balance at the current location of the cursor.

- ***Left-click+drag*** in the long strip chart to move the position of the cursor.

- ***Left-click*** above then ***right-click*** below the area of interest in the long strip chart in order to ***Zoom In*** on that area.

- ***Middle-click*** in the long strip chart in order to ***Auto-scale*** (*auto-fit*) the currently loaded data file.

- The red cursor in the long strip chart represents the data currently being displayed in the expanded strip chart.

- The center red-line cursor in the expanded strip chart is stationary.

- ***Left-click+drag*** in the expanded strip chart to adjust the position of the data.

- ***Right-click+drag*** in the expanded strip chart to symmetrically adjust the 2 outboard red-line cursors in order to *trim down* on a signal.

- ***Shift+right-click+drag*** in the expanded strip chart to independently adjust either outboard red-line cursor.

Printing Eddy Current Graphics

To print a *screen dump*, setup the desired printing parameters on the ***Print*** tab of the ***Options*** dialog. Afterwards, with the desired screen layout displayed showing the eddy current signals of interest, click ***File | Print*** or press ***Ctrl + P*** on the keyboard.

Changing the X-Y Ratio

To change the ***X-Y*** ratio for a given channel, ***Shift+click*** the ***Span*** box of the specific Lissajous, then ***right-click & drag*** in the Lissajous until the desired ***X-Y*** ratio is achieved. The ***X-Y*** ratio is reported in the status bar along the bottom of the ***Analysis*** screen and in the ***YFAC*** column on the ***Raw*** tab of the ***Setup*** dialog. This feature adjusts the Y component of the eddy current channel. To reset the ***X-Y*** ratio, ***Shift+click*** the ***Span*** box of the specific Lissajous, then click in the Lissajous.

Independent Balance

During some inspections, especially of copper-based tubing in raw water heat exchangers, general full-length ID erosion can occur. In order to calibrate for and depth size this type of damage correctly, it is necessary to have the appropriate ID erosion calibration standard, plus have the ability to measure the wall loss in the tube relative to the balance point of the external reference tube. v5.0 provides for this ability.

To use this new feature, open the ***Setup*** dialog, click the ***All*** tab, and click in the ***BAL*** field for the channel(s) that will be used to measure this type of damage. ***SAV*** (*Save*) will appear in the field. Click ***OK***.

Next, position the cursor in the cal standard at a good balance point, then select ***Util | Save Balance***. All channels where ***SAV*** appears in the ***BAL*** column of the ***All*** tab in the ***Setup*** dialog will be balanced and the balance values saved. Balancing normally, i.e., Spacebar, will balance all non-***SAV*** channels normally, while performing only a display balance for all ***SAV*** channels. When a ***Vvm*** measurement is made on a ***SAV*** channel(s), only one yellow measurement ball will appear because, for all intensive purposes, the other yellow measurement ball is located *on* the saved balance point in the external cal standard.

Manually Using System Landmarks

Sometimes it may be necessary to manually label a landmark. For example, if ***Auto Locate*** mislabels one or more landmarks, it may be necessary to remove the invalid label(s) and manually insert the correct labels for the landmarks affected.

To remove a landmark label from the landmark strip:

STEP 1: ***Middle-click*** the landmark label.

To insert a ***System*** landmark label on the landmark strip:

STEP 1: Locate and accurately center the landmark to be labeled within the expanded strip chart.

STEP 2: ***Right-click*** in the landmark strip to display the ***System*** landmarks popup.

STEP 3: ***Click*** the applicable landmark label from the ***System*** landmarks popup.

STEP 4: Repeat the above steps until all the desired landmarks are labeled as desired.

Setting-up and Using a Manual Scale

To setup and use a manual scale, select ***Edit | Landmarks*** and click the ***User Defined*** tab. Here, the user can enter a list of manual landmarks as needed. The ***POINT*** column changes automatically as a landmark is set in the landmark area of the main. Any number of user-defined landmarks may be added. To insert a landmark in an existing ***User Defined*** list, position the cursor in any field of the landmark where you want the row inserted and press ***Shift + Insert***. To add a new entry to the bottom of the list, press ***Enter*** after each ***NAME*** and ***POS*** (distance) entry and a new row will automatically appear, or simply click the gray box above the vertical scroll bar. When entering user-defined landmarks, remember that each landmark builds in distance from the previous landmark. For example, if the next landmark at the bottom of a list was ***REF2*** located ***12-inches*** from ***TE***, ***REF2*** would be entered as the ***NAME*** and the ***POS*** value would be ***24-inches*** – the total distance from ***REF*** to ***REF2*** as shown in Figure B-1. The ***OK*** and ***CANCEL*** buttons work as update and undo functions, respectively.

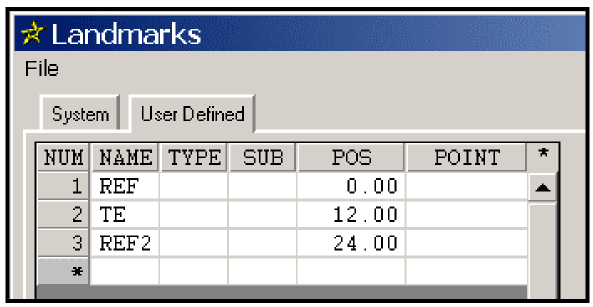


Figure B-1. Manual Landmarks

Once two or more *manual* landmarks are present in the ***User Defined*** landmarks table, follow the steps below to manually set and use these landmarks versus using the ***System*** landmarks from the component (*.cmp*) file:

STEP 1: Position the red cursor bar in the long chart at the first landmark to be labeled.

STEP 2: ***Right-click*** in the landmark strip to display the ***System*** landmarks dialog.

STEP 3: ***Click*** the title of the landmarks popup to switch to the ***User*** landmarks.

STEP 4: ***Click*** the applicable landmark from the ***User*** landmarks dialog. The selected landmark appears in the landmark strip at the cursor location.

STEP 5: Position the red cursor bar at the next landmark to be labeled.

STEP 6: ***Right-click*** in the landmark strip to display the ***User*** landmarks popup.

STEP 7: ***Click*** the applicable landmark from the ***User*** landmarks popup for this landmark.

From-To Measurements

Some eddy current indications are long and gradual - especially damage mechanisms such as steam erosion, tube-to-tube wear, etc. If it is desired to include the affected range of the damage indication, a **From-To** measurement may be performed.

To make a **From-To** measurement:

STEP 1: Position the red cursor bar in the long strip chart at the beginning of the indication. Refine the location in the expanded strip.

STEP 2: ***Shift+click*** in the ***LOC*** *(ation)* box in the upper right corner of the ***Analysis*** screen. The ***From*** portion of the ***From-To*** measurement will appear in the box & in the ***Status Bar*** below

STEP 3: Position the red cursor bar in the long strip chart at the end of the indication. Refine the location in the expanded strip

STEP 4: ***Shift+right-click*** in the ***LOC*** box. The ***To*** part of the ***From-To*** measurement will appear in the box. The complete ***From-To*** range now appears in the ***Status Bar*** below

STEP 5: Position the red cursor bar in the long strip chart to the center of the indication and refine the location in the expanded strip chart. If the length of the indication exceeds the expanded strip chart, click the ***Points/Pixel*** button in any Lissajous in order to *compress and fit* the total indication within the expanded chart. Measure the indication as desired, i.e., **Vpp**, **Vmr**, **Vvm**.

STEP 6: Make the call. Note that the location field for the new report entry now contains the ***From-To*** range.

Guess Angle

Sometimes you may need to measure a signal manually since none of the built-in measurement methods provide the voltage and/or phase angle you desire. This method is called ***Guess Angle (Vga)***. It's like stretching a rubber band along the desired axis of the eddy current signal.

To make a ***Guess Angle*** measurement, ***middle-click & drag*** in any Lissajous. A green vector appears in the Lissajous and the voltage, phase, and percent values vary while performing a ***Vga*** measurement.

Analyzing Data From a Central Server (PC)

It's very important that you open the project of interest locally on your workstation. If the data for the project is located over a Local Area Network (LAN) on a central server, all you need to do is to adjust your project parameters in ***EddyAdmin*** to point to where the data resides. A copy of the same project in its entirety must reside on the server as well as all other workstations that will be using the project.

WARNING: ***NEVER*** remotely open a project on a PC over a LAN from multiple workstations.

To modify the location of the eddy current data files (**`\ecdata`**) for a local project:

`STEP 1:` In ***Analysis***, select ***Tools | EddyAdmin***.

`STEP 2:` In the ***Data*** section of the ***EddyAdmin*** dialog, click the **...** button located to the right of the ***Directory*** field for ***DSET 1***. An ***Open*** dialog appears.

`STEP 3:` Select the **`\ecdata`** directory on the appropriate network drive and path for the desired project, and then click ***OK***.

`STEP 4:` Select ***File | Save***.

`STEP 5:` Select ***File | Exit***. You are returned to the ***Analysis*** screen.

Now when you open the ***TLIST*** in ***Analysis***, the cal groups located in the newly selected location will be available.

Appendix C: More about MakeComp

- To ***Zoom In*** on a map, ***Ctrl+Click***.
- To ***Zoom Out*** on a map, ***Ctrl+Right-click***.
- To ***Auto-zoom*** a map, ***Ctrl+Middle-click*** or ***ESC*** key.
- To ***Move*** or ***Pan*** a map, ***Middle-click+Drag***.
- To move a group of tubes to a new location, right-click the tubesheet map, select ***Move Group***. The cursor becomes a crosshair pointer. ***Click & drag*** any tube in any group to the new location and release.
- Select ***File | Import*** to copy an existing external component file (***`.cmp`***) into the current project. The copied file name becomes ***`project.cmp`***.
- Select ***File | Export*** to save a copy of the current component file to a location other than the current project. Any user-defined filename may be used during ***Export***.
- To manually select more than one tube at a time on the tubesheet map, click on the first tube to highlight it (*green*), and then ***Shift+click*** the remaining tubes.
- Alternatively, ***Shift+click*** a highlighted tube to deselect it.

Appendix D: More about Auto-locate

- Auto-locate is very speed dependent. If the probe speed varies significantly during acquisition, so will the auto-locating accuracy. The sample data provided in this tutorial shows evidence of this at the beginning & end of some of the data files. Note that the tube support signals at the beginning of some of the data files is very short – indicating the probe *snapped*. A similar condition exists at the end of some of the data files where the probe technician began to pull the probe back at the same time as the probe-pusher while anticipating the exist of the probe from the tube and, in effect, increasing the probe speed at that localized area of the data file.

- To remove a landmark label from the landmark strip, middle-click the landmark label.

- To manually add a landmark at the cursor location in the long strip chart, right-click in the landmark strip and select the desired label.

- System landmarks reside in the ***`project.cmp`*** file. Select ***Tools | MakeComp*** to edit the landmark tables.

- A red landmark label indicates a landmark that could not be found, but where the software estimates it should be. This is very useful for segmented baffle arrangements where baffles *come and go*.

- A white landmark label indicates a landmark has been identified & marked accordingly.

- Auto-locate files can be shared between users to save setup time.

- If you open ***Cal 6*** (*outlet data*) of the tutorial sample data, you will have to set the ***Leg*** field in the ***Summary*** to ***INLET*** even though it's outlet data. This is because we are treating each half of this U-tubed component as separate tube bundles. Regardless, the outlet landmark labels will be displayed in the landmark strip per the landmark names assigned in the specific landmark table. You will need to *train* auto-locate in the same way as you just did for the inlet data.

Appendix E: Understanding Queries

The purpose of this appendix is to give you a better idea of what makes up an ***SQL*** query, explain a few simple ***SQL*** commands, and give some examples of basic queries. It is not intended to teach advanced query logic or discuss ***SQL*** beyond practical application in ***DBMS***.

<u>What is SQL?</u>

SQL stands for *Standard Query Language*. It is a language like any other in that it has grammar, syntax and some vocabulary. ***SQL*** offers a logical interface with relational databases in order to access or manipulate data. Using ***SQL*** we can ask the database for specific data, change or delete records, and even create tables and fields.

<u>What is an SQL Query?</u>

The most common ***SQL*** query is the ***SELECT*** query. The ***SELECT*** query *asks* the database for specific records organized in a specific way.

An example of a ***SELECT*** query asking for all tubes in row 5 and less with a DNT indication listed by descending voltage may look like this:

```
SELECT row, col, sec, phase, volts, defect, loc_land,
loc_off, beg_test, end_test
FROM report
WHERE row BETWEEN 1 AND 5
AND defect = 'DNT'
ORDER BY volts DESC;
```

The words in upper case are ***SQL*** commands instructing the database to respond in a certain way. It isn't necessary to place commands in upper case, although, it does help organize the query at least until you become comfortable working with ***SQL***.

Working line by line through the query above, let's see what each line does.

- ***SELECT*****:** The defining line of the query, ***SELECT***, tells the database first that we are looking for data and second what fields we want to see in a response. Each field must be typed as it appears in the database and separated by a comma. It is not case sensitive. You may order the fields in any way. The space between the comma and field names isn't necessary but, again, it helps organize the query.

- ***FROM*****:** Tells the database in which table to look for the data. In our example, the desired data is located in the ***report*** table. Again, the field names in the ***SELECT*** line must match the field names in the ***report*** table.

- ***WHERE*****:** Defines what specifics we are looking for. In our example, we want all tubes ***WHERE*** the row is between 1 and 5. ***BETWEEN . . AND*** is a command that tells the database to look between two values - inclusive. Other operators are: **=, <, >, < =, > =, < > *(not equal to)***.

- ***AND/OR*****:** Extends the main ***WHERE*** statement and further defines what you are looking for. There can be as many ***AND/OR*** lines attached to a ***WHERE*** statement as logic permits. ***AND*** implies that both statements must be true. ***OR*** implies that either one or the other statement must be true. Both the ***WHERE*** and ***AND/OR*** lines follow the same syntax. First the command, then a field name, a designator to indicate the relationship to the value we are interested in, then finally the value of interest.

- ***ORDER BY*****:** Says to organize the data according to which field. In our example, we chose to ***ORDER BY*** (*or sort by*) volts. To be more specific you can add ***DESC*** or ***ASC***. ***DESC*** instructs the database to place the records in descending order, ***ASC*** in ascending. If either ***DESC*** or ***ASC*** is not specified, ascending order is selected by default.

If you note there is a semi-colon at the end of the last line. This tells the database that the query has ended. It is not necessary to place the semi-colon for a query to be valid.

<u>WHERE Statements</u>

In this section, we will go over some simple variations in the ***SELECT*** query. All of these focus on the ***WHERE*** line of the query. Above we saw the ***BETWEEN . . .AND*** command. This is used to set an upper and lower limit and request the values that fall between them.

- ***NOT***: By using the ***NOT*** command, we tell the database to return values *not* matching the ones specified. For example, if you want to see the records for all inspections except *RFO7*, then you might query:

  ```
  WHERE NOT outage = 'RFO7'
  ```

  Note the positioning. ***NOT*** comes before the field and operator it affects.

- ***IN***: Another useful command is ***IN. IN*** allows us to specify more than one value for a given field. In our example above, perhaps we did not want to see data for *RFO6 and RFO5* as well. We could type:

  `WHERE outage NOT IN ('RFO7','RFO6','RFO5')`

  After ***IN*** the values are grouped in parentheses and separated by commas. Of course, this command can be used with or without ***NOT***.

- ***Text vs. Numeric Values***: Our examples show that when we have a text value, it is placed inside single quotation marks, i.e., *'RFO7'.* When we use numeric values, they do not need to be placed in quotes. For example, *WHERE row IN (1,2,3,4).*

- **Query on a query**. The FROM clause can use a query as well as a database table. For example, if you find you constantly use a long list of fields, create a query in the database called say myreport = SELECT row,col, …. FROM report. You can then use SELECT * FROM myreport followed by the usual WHERE and ORDER BY clauses.

- ***IS NULL***: We can also query on a field when we expect it to have no value. The command for this is ***IS NULL***. If we believe there were some *DNT* indications loaded into the database without voltage readings, then we might use the following query to find them:

  ```
  WHERE defect = 'DNT'
  AND voltage IS NULL
  ```

  This would return all the records with *DNT* in the *defect* field and no value (*empty*) whatsoever in the *voltage* field. A blank character *is not* the same as ***Null***.

Things to Remember

- Be aware of the order within the ***WHERE*** statement. ***SQL*** is logical and as you work through the lines of the statement, the order does matter. What you ask for first, second and so on will limit the records differently. Think through what you want a query to return. Make sure that your entire logic matches what you want. Remember, just because a query runs and returns something doesn't mean it's correct. As long as the logic is good, the query will run; however, make sure it's valid logic. If you're unsure of the operator precedence, you can use parenthesis to help define the logical flow. For example, X=3 AND (Y=4 OR Y = 5) is different from (X=3 AND Y=4) OR Y = 5. It's not so much the order that matters as the operator precedence and grouping.

- You can query any table. The table name goes in the **FROM** statement. Most queries will be directed at the *report* table.

- A library of queries is available via the CoreStar public FTP site. Please conduct the main office for instructions and a password to logon to the FTP site and download this library of queries. These queries make great templates to build on. In most cases there will be a saved query to help you at least get started, if not simply allow you to fill in the blanks.

- It cannot be stressed enough to buy a good book on ***SQL*** or have some formal instruction. We have only taken a practical look at the bare bones of ***SQL***. ***SQL*** is an extremely powerful tool and the key to maintaining a relational database. Without ***SQL***, you lose an enormous amount of ***DBMS*** potential.

  SQL for Dummies is an excellent reference book, written in plain language. CoreStar's primary ***DBMS*** instructor owns this book and swears by it.

- You can always contact CoreStar Technical Support directly for basic questions regarding ***SQL***.

- The ***Query*** dialog allows you to *store* queries internally or *save* them externally. Externally saving query files (qry) offers the advantage of easily sharing the files project-to-project, user-to-user, via email, etc. Certain complex *joined* queries must reside internally to execute properly.

- ***File | Save*** or ***File | Save As*** on the ***Query*** dialog saves the current query to the queries directory of the current project. Conversely, ***File | Export*** saves the query to a user-defined location *outside* the current project.

- ***File | Open*** on the ***Query*** dialog points to the queries directory of the current project. Conversely, ***File | Import*** allows the user to import (*copy into the current project*) a query from a user-defined location *outside* the current project.

Appendix F: Project Tips

The project concept is a major advancement in the ***EddyVISION32*** software suite. To better prepare for future new projects, consider creating a ***Master*** project. This ***Master*** should contain, but not limited to, the following typical files:

- Your favorite queries copied to the `[MasterProject]\queries` directory.
- Your favorite print option files copied to the `[MasterProject]\print_options` directory. These files contain headers and footers used for printed query lists and eddy current reports.
- Your common map files copied to the `[MasterProject]\maps` directory. Remember, you can share these map files between projects with just a few edits. This is one of the biggest timesavers within DBMS.
- Your common defect list files copied to the `[MasterProject]\defect_lists` directory.
- Your common map header files copied to the `[MasterProject]\map_headers` directory.
- All common ***Project Lookup Table*** entries in ***EddyAdmin*** s, i.e., components, users, standards, testers, probes, etc.

In addition, you can open each software module and set user options as desired. For example, some options may include color choices for Analysis/Acquisition/DBMS, and screen layouts for Analysis/Acquisition.

Once your ***Master*** project is complete, you can easily and quickly use ***Project | Copy*** to create an identical duplicate of the ***Master*** project named as directed by you. All you would need to do is open the newly copied project in ***EddyAdmin***, import the component file if it exists, import the database if it exists, make the desired selections on the ***EddyAdmin*** dialog, and ***File | Save***.

You may elect to use ***Project | Replicate*** instead to create a new project based on the ***Master*** project. ***Replicate*** does not copy map files, data files, or report files to the replicated project; however, no data files or report files should exist in your ***Master*** project anyway.

These techniques will save you hours on end of preparation time and keep your entire project consistently the same.