

## Frequently Asked Questions

### URSA

#### Installation, configuration and setup

##### **Question**

*I just installed URSA and chose Winsock as my connection type. When I try to connect, I get the message "UNABLE TO OPEN SOCKET." What does this mean?*

##### **Resolution**

To connect to the A Series server using Winsock, you need to create a connection profile that specifies the A Series host name.

1. In URSA, choose the **Connection Profile** command (File menu).
2. In the **Connection Profile** dialog box, specify a name for the profile.
3. Select a connection type (Winsock in this case).
4. Choose the **Configure** button.
5. Specify your station name (optional) and your A Series host name.
6. Enter any logon information you need in the **Connection Profile** dialog box.
7. Choose **Save**.
8. Choose **OK**.

---

##### **Question**

*I just installed the latest 32-bit versions of URSA and the Micro Filter for Windows. When I try to connect to URSA, I get a "CONNECTION CLOSED" message. What could be wrong?*

##### **Resolution**

There may be a problem with the COMS window name. When you install URSA, you need to specify a valid COMS window name in the **Server Window** dialog box. In this case, the actual window name was URSA and URSADemo was entered in the **Server Window** dialog box. To fix the problem, reinstall URSA and specify the correct window name *or* modify the **Drgscript.txt** file that is in your URSA directory. The window name is listed three times in the **Connecting** section of the file. In this case, replace all instances of ?ON URSADemo with ?ON URSA.

---

### Question

*When I try to connect to URSA, I get a "MAX RETRIES EXCEEDED" message. What could be wrong?*

### Resolution

There may be a problem with the COMS window name. When you install URSA, you need to specify a valid COMS window name in the **Server Window** dialog box. In this case, the COMS window for DARGAL was specified rather than the one for URSA. In other cases, the name may be invalid. To fix the problem, modify the **Drgscript.txt** file that is in your URSA directory. The window name is listed three times in the **Connecting** section of the file. In this case, replace all instances of ?ON DARGAL with ?ON URSA.

---

### Question

*When I try to bring up URSA, I cannot make a connection or select any of the options in the product. I never get a logon prompt either. What could be wrong?*

### Resolution

There is probably incorrect information stored in the connection profile. Perhaps there's an invalid host name or ID specified in the profile. When the profile was set up, the option **Auto Connect on Startup** was checked in the **Connection Profile** dialog box and the default user ID and password were filled in. Since the connection is automatic and the ID and password are prefilled, you cannot see the information that is being passed to the host.

Change the setting for automatic connection in the **ursaconn.ini** file in your URSA folder. Edit the file so that **AutoConnect** is set to 0 rather than 1. If there is more than one profile, change the setting for each one. Then run URSA again. Since the connection is no longer automatic, you can check the information specified in the connection profile. Change the invalid information and then resave the connection profile.

---

### Question

*We are performing numerous tests on one of our machines. When I try to execute a spec in URSA, I get the message "waiting on port." Is there a problem with the spec or is this related to our testing?*

### Resolution

This message typically appears when a machine's resources are taxed. It implies that a port is not available at the time you made the request. The report should run fine once a port becomes available. It does not mean there is anything wrong with the spec.

---

### Question

*When I try to run an URSA report that sorts about 4 million records, I get the message "A DISK FILE AREA COULD NOT BE ALLOCATED." What is the problem?*

### Resolution

DARGAL uses a file named **SEQFILE** to sequence reports. The default maximum attributes for **SEQFILE** are 1000 areas with an areasize of 1000 (this number varies with record size). If you need to sequence a report that contains more than 1,000,000 records, file-equate **SEQFILE** with a larger areasize.

To determine the areasize needed, divide the number of records for the report by 1000 and round up to a convenient number. For example, if you think 4,000,000 records are being used by the report, specify an areasize of approximately 4000 with the URSA host. Use the WFL MODIFY command to set this option:

**WFL MODIFY (DARGAL)DARGAL/URSAHOST; FILE SEQFILE (AREASIZE=4000);**

*Note:* This is a powerful command that can produce undesirable results if used incorrectly.

---

### Question

*I have set up URSA on my machine the way I need it. Can I copy the "INI" files to other users PCs to save time? We use the same connection profiles, etc.*

### Resolution

You can copy the "INI" files to other PCs. The **Ursa.ini** file contains information about default connection profiles, settings, report options, preferences, etc. The **Ursaconn.ini** file contains detailed information about your connection profiles. You can even copy the **Dargal.ini** file which contains micro filter settings such as DDE extensions and path settings. If you need to make changes to any of the files after you copy them, use a text editor to modify them.

---

### Question

*Whenever I exit URSA, I get an application error. The message states that the instruction at "0x01208cd3" referenced memory at "0x01208cd3" and the memory could not be read. What does this mean?*

### **Resolution**

This error occurs if you have the original Micro Filter for Windows and the Micro Filter for TCP/IP loaded in the same directory on a PC that is running Windows NT or Windows 2000. Uninstall both Micro Filters and then reinstall just the Micro Filter for TCP/IP. You don't need both Micro Filters; use just the one for TCP/IP.

---

### **Question**

*I have more than one connection profile defined for URSA, yet I don't see any way to choose the one I want. Isn't there supposed to be a way to choose the one I want when I to connect to the host?*

### **Resolution**

As of URSA Version 3.00.0240, you can choose the connection profile you want to use when you connect to the host. When you choose the **Connect to Host** command (File menu), URSA displays a dialog box that contains a list of all your available connection profiles (if you have defined more than one). If you have defined only one connection profile, the dialog box does not appear. Also, if you have set **Select Auto Connect on Startup** to connect to the host automatically each time you run URSA, the dialog box does not appear. In this case, your last default connection profile is used.

---

### **Question**

*I downloaded URSA Version 3.00.0240 from your web site. When I try to connect to URSA, I receive the message "CONNECTION CLOSED." What is wrong? I'm sure I set everything up correctly, including the COMS window.*

### **Resolution**

Uninstall URSA and remove all files associated with your previous installation. Then install the new version of URSA. In this case, this procedure remedied the problem.

---

### **Question**

*I just installed new releases of Version 59 DARGAL and URSA. When I try to run URSA, I get the message "(DARGAL)OBJECT/DARGAL/SUPPORT NOT STARTED." What does this mean?*

### **Resolution**

This message can appear if you have not copied the DARGAL MONITOR file. In Version 59 and earlier, the file used to run the Monitor is named **\*OBJECT/DARGAL/MONITOR** and must be located on the halt/load pack. In Version 60, the file is named **OBJECT/DARGAL/MONITOR** and it must be on the same usercode and pack as your other DARGAL OBJECT files.

---

### **Question**

*We just installed a new security application. When I run URSA, I get the message "CONNECTION CLOSED" after I enter the usercode and password. What is wrong?*

### **Resolution**

The new security application and **Drgscrpt.txt** file are not in sync with each other. Modify the **Drgscrpt.txt** file that is in your URSA directory to reflect information on the custom screen. Replace #SESSION with PASSWORD and #VERB with EXIT to get it working properly.

---

### **Question**

*We are getting the message "UNABLE TO LOAD DLLs" when we try to open URSA. We use FASTerm as our emulator and just upgraded. This never used to be a problem. What could be wrong?*

### **Resolution**

At this site, they were using 16-bit URSA and 16-bit Micro Filter for Windows, but they upgraded to the 32-bit version of FASTerm. To be safe, you should use either all 32-bit versions or all 16-bit versions. If possible, use the 32-bit versions.

---

### **Question**

*When I try to open a spec in URSA, I get the message "SPEC VERSION NEWER THAN CURRENT VERSION." How could this be?*

### **Resolution**

You are using a version of URSA that is older than the version in which the spec you are trying to open was saved. At this particular site, they run different versions of URSA. The user had recently reinstalled URSA on his computer and inadvertently installed an older version. Installing the version in which the specs were created solved the problem.

---

### **Question**

*I normally send my URSA reports to a particular system printer. How do I specify that I want to use a different printer?*

### **Resolution**

If you have more than one system printer, the system sends output to the first available one. Use the **Destination** option in the **Output Device** dialog box (for system printer) to name a specific device. In the **Destination** text box, enter the name of the device at which you want the file to print. You can specify a line printer (LP <device #>), image printer (IP <device #>), station (STAtion=<station name>), group name, or virtual device name.

## Functionality-related

### **Question**

*Can you create crosstabulation reports in URSA the way you can in EZSPEC/DARGAL?*

### **Resolution**

You cannot create crosstabulation reports in URSA as you can in EZSPEC/DARGAL. However, you can create a spec that displays URSA output as a spreadsheet with row and column totals. Contact our Client Services department (704-845-1000 or cs@dsinc.com) if you want more information about this type of application.

---

### **Question**

*Is there a way to stop a report in URSA?*

### **Resolution**

1. Choose the **Status** button on the toolbar after URSA begins to execute the report.  
The **Report Status** dialog box appears. Report processing is suspended at this point.

A message reflecting the report's status appears near the top of the dialog box. The mix number of the host and the number of records read and processed up to this point appear.

2. Choose the **Stop** option button.

When you choose this option, the **Discard Output** check box becomes available. You can select this check box to discard the output received up to this point. If you do not select this check box, the report stops but URSA retains the output received before you stopped the report.

3. Choose the **OK** button.
- 

## Question

*I've generated a DARGAL report in URSA. Now how do I run it?*

## Resolution

Once you generate a DARGAL program from within URSA, you can run it in DARGAL by using the Run command. You cannot run it from within URSA, but you can list it in URSA by choosing Host Commands (File menu) and then entering LIST <program name> beside **Command** in the **Host Commands** dialog box.

---

## Question

*I'm creating a report that contains two break fields. Do I need to sort by these fields also? If not, when is sorting needed?*

## Resolution

When you create a report that contains break fields, URSA automatically uses these fields as sort fields. You do not need to specify any separate sort. If your report contains detail data only or detail data as well as summary data, you can sort the detail data in a particular order by specifying sort fields. Depending on the type of report you are creating, you can specify a sequence by break fields, sort fields, or both. Follow these steps to specify sort fields:

1. Use the **Sort** button on the toolbar.
2. Highlight the sort fields (the fields by which you want to sequence detail data) and choose the **Insert** button. Then choose the **Close** button.

The **Sort** dialog box appears.

If you have specified any break fields for your report, they appear in the **Groups** list.

You cannot change break fields in the **Sort** dialog box. (If you need to change them, use the **Groups** dialog box.) They are listed in the **Sort** dialog box to tell you that URSA sorts the report by the break fields first if you have specified both break and sort fields. The fields you chose for the detail sequence appear in the **Sort Fields** list. You can specify **Ascending** or **Descending** order for each sort field.

3. Choose the **OK** button.

---

### Question

*Is there a way to make an URSA define and a join available to all users in all sessions?*

### Resolution

The best way to provide mass access to a define is to add it to the FD using URSA Administrator, EZFD, or FDSYS. Then make sure that all users who need the FD can access it through their masters. To provide mass access to a join, make an entry for the join FD in the masters of all users who need to access it. Once again, use URSA Administrator, EZFD, or FDSYS to make the entry.

---

### Question

*Is there a way to make an URSA define and a join available to all users in all sessions?*

### Resolution

The best way to provide mass access to a define is to add it to the FD using URSA Administrator, EZFD, or FDSYS. Then make sure that all users who need the FD can access it through their masters. To provide mass access to a join, make an entry for the join FD in the masters of all users who need to access it. Once again, use URSA Administrator, EZFD, or FDSYS to make the entry.

---

### Question

*When I sequence my report, I get a message about records excluded on sequence errors. How can I print out these "bad" records?*

### Resolution

There is an option called **Sequence Error Action** on the URSA **Report Options** dialog box (**Report** button). The default setting for **Sequence Error Action** is **SKIP**. This means that URSA passes over any records containing bad or missing data in fields used for sequencing.

Other possible settings for this option appear in a drop-down list:

- **BEGINNING** causes records containing bad or missing data to be sorted to the beginning of the report.
- **END** causes such records to be sorted to the end of the report.
- **ERROR** terminates the report with an error if bad or missing data is encountered in an expression used for sequencing.

Choose **BEGINNING** or **END** and then print your report.

---

## Question

*Is there a way to show the name of the host you are using in URSA?*

## Resolution

The name of the host to which you are connected appears in the right side of the status bar in URSA. You need to be running URSA Version 3.00.0240 or higher and DARGAL Version 59.180 or higher to see this feature.

---

## Question

*I'm trying to use the cutoff year feature that was added to URSA Version 3.00.238. I can't seem to get to this option; where is it?*

## Resolution

First of all, you need DARGAL Version 59.170 or higher to use the cutoff year feature in URSA. To set a cutoff year in URSA, use the **Report Options** command (Options menu) to display the **Report Options** dialog box. The **Report Options** dialog box contains a section labeled **Cut Off Year**. The **Cut Off Year** section contains two options. The first option allows you to specify the cutoff year value. The second option is a check box labeled **Use Host Value**. It allows you to determine whether URSA uses a cutoff year value from your URSA setupfile. If you need more specific information, see the online help in URSA.

---

## Question

*I have created some custom help files specific to the data we use at our site. How do you suggest I make them available to URSA users?*

### **Resolution**

You can set up an icon (on each user's desktop) that points to the custom help files. They can open the help files at the same time they are using URSA.

---

### **Question**

*I frequently run test reports before I run a whole report. Ten records is not enough for me to get a feel for the report. Is there a way for me to increase the number of records displayed in a test report?*

### **Resolution**

The default number of records URSA reads for a test report is 10. To specify a different number, enter the number in the box beside **Test report size** in the **Report Options** dialog box. Follow these steps:

1. Choose the **Report Options** command (Options menu).
  2. Choose the **Report** option button in the **Options** section of the **Report Options** dialog box.
  3. Enter a new number (for example, 50) in the box beside **Test report size**.
  4. Choose **OK**.
- 

### **Question**

*Is there a way for me to copy an object from one section to another in the Layout window? I've tried and am not having success.*

### **Resolution**

Yes, copying objects is a simple process. Just press and hold the CTRL key on your keyboard while you drag and drop the object into the new section.

---

### **Question**

*What do the various 'EXCLUDED ON' errors mean at the end of my URSA reports?*

## Resolution

There are several types of "EXCLUDED ON" errors that can occur in your report: selection ("SELECT" or "WHERE") errors, sequencing ("ORDER BY" or SEQUENCE") errors, and until errors. These errors occur when a field that you are using in a selection condition, sequencing logic, or until condition contains missing or invalid data. By default, URSA excludes these records from your report. If you want to include these records, reset the report options that determine what URSA does when it encounters one of these fields. You can make these changes from the **Report Options** dialog box. To specify the action URSA takes when it encounters an error, follow these steps:

1. Choose the **Report Options** command (Options menu).
2. Choose the **Report** option button in the **Options** section of the **Report Options** dialog box.

Click here to see the [Report Options](#) dialog box when **Report** is chosen.

The options **Until Error Action**, **Select Error Action**, and **Sequence Error Action** control how URSA handles an error when it encounters missing or invalid data in one of the fields.

Use **Until Error Action** to tell URSA how to handle UNTIL conditions that contain bad or missing data. The default value for this option is **CONTINUE**. The report continues as if no missing data had been encountered.

Other possible settings for this option appear in a drop-down list:

- **STOP**  
The report stops when bad or missing data is encountered, as if the UNTIL condition had been met.
- **ERROR**  
The report terminates with an error.

Use **Select Error Action** to tell URSA how to handle SELECT or WHERE conditions that contain bad or missing data. The default value for this option is **EXCLUDE**. If an error occurs in the evaluation of selection logic, the record is discarded.

Other possible settings for this option appear in a drop-down list:

- **INCLUDE**  
If you choose this setting, URSA selects records with bad or missing data.
- **ERROR**  
The report terminates when URSA encounters an error in selection logic.

Use **Sequence Error Action** to tell URSA how to handle ORDER BY or SEQUENCE conditions that contain bad or missing data. The default setting for this option is **SKIP**. URSA passes over any records containing bad or missing data in fields used for sequencing.

Other possible settings for this option appear in a drop-down list:

- **BEGINNING**  
If you choose this setting, URSA sorts records containing bad or missing data to the beginning of the report.
  - **END**  
If you choose this setting, URSA sorts such records to the end of the report.
  - **ERROR**  
If you choose this setting, URSA terminates the report with an error if it finds bad or missing data is encountered in an expression used for sequencing.
3. Specify your settings for the action options.
4. Choose the **OK** button in the **Report Options** dialog box.

### Microfilter-related

#### **Question**

*When I try to run a report in URSA, I get the message DARGAL MICRO CODE NOT FOUND. I know I installed it, so why is this happening?*

#### **Resolution**

You should check to see where you installed the Micro Filter. If you installed the Micro Filter in a location other than the default one, URSA may not be able to find the Micro Filter. Either move the files to the default location or start the Micro Filter manually.

---

#### **Question**

*How do I force URSA reports to go to my LAN printer rather than my local printer?*

#### **Resolution**

Specify the printer you want to use for report output in the **Micro Filter** window.

1. Bring up the **Micro Filter** window by clicking on the DARGAL icon (three nested probability curves).
  2. Choose the **Printer Setup** command (File menu).
  3. In the **Print Setup** dialog box, select your LAN printer from the **Name** drop-down list.
  4. Choose **OK**.
-

## Question

*When I send a report to micro disk, the message about the number of records read and processed doesn't appear in the file. Is there a way to make it display?*

## Resolution

The message about the number of records and processed appears when you send URSA output to the **Browse** window. It does not appear in output to micro disk by design, since you typically want to use the micro output file in another application, etc.

If you want the information to appear in the micro disk file, follow these steps to add it to the report footer:

1. Double-click in the **Report Footer** section in the **Layout** window.
2. The **Select Fields - Report Footer** dialog box appears.
3. Select the **Include** option **Strings**.
4. In the box above the fields list, enter "**RECORDS READ: "** and choose **Add**.
5. Enter **(READCOUNT)** and choose **Add**.
6. Enter "**RECORDS PROCESSED: "** and choose **Add**.
7. Enter **(SELECTCOUNT)** and choose **Add**.
8. Double-click on "**RECORDS READ: ."**
9. Highlight **(READCOUNT)** and choose the **Value** button.
10. Double-click on **RECORDS PROCESSED: ."**
11. Highlight **(SELECTCOUNT)** and choose the **Value** button.
12. Choose **Close** in the **Select Fields - Report Footer** dialog box.
13. Choose **OK** in the **Report Footer** dialog box.

Use the **Layout** window to arrange the information any way you want.

---

## Question

*How do I increase the record size for micro disk? My record size is greater than 2000 and I don't want the data to wrap.*

## Resolution

In the **Output Device** dialog box for Micro disk, you can specify micro file attributes (in parentheses) after the file name. For example, the following sets the record size to 300:

**D:\TEMP\ALIST.TXT (RECSIZE=300)**

The maximum record size you can specify for output to micro disk is 1900, however. If you need a larger size, as in this case, send your output to system disk. In the **Output Device**

dialog box for System disk, specify a file name and then enter **MAXRECSIZE=2000** in the **I/O Subsystem Options** section. You can then use the Copy utility to transfer the file from the mainframe to the micro.

---

### Question

*I'm trying to bring micro output into Word 2000. There seems to be a conflict. Is there a special option I need to set?*

### Resolution

If you are using Microsoft Word 2000, you need to change the security setting for macros to **Medium** in Word. The default setting is **High**, which does not allow you to enable DDE macros automatically. In Word 2000, choose **Tools** menu > **Macro** command > **Security** option > **Security Level** tab > **Medium** setting to reset the security level so you can use DDE.

---

### Question

*I get the message "UNEXPECTED TEXT" when I try to send an existing spec to a micro disk file. What's wrong?*

### Resolution

When you receive this message it typically means that there is an invalid character in the micro disk file name — possibly a space. Rename the file so the invalid character is no longer there.

---

### Question

*I'm importing URSA output into Access97. The data in one of my large numeric fields (that contains a code) is shown as missing in Access. What could be wrong?*

### Resolution

First, check the contents of the micro file from within URSA. Choose **Host Commands** (File menu) and then enter a command similar to the following beside **Command**. (Use the name of your file in the syntax.)

**LIST MICRO \TEST\MDOUTPUT.TXT**

*Note: The word MICRO indicates that you are listing the contents of a micro file.*

If the contents of the file appear okay, Access may not be able to handle the field properly. In some versions of Access, there is a long integer limit of two billion. If this is the case, create a define in URSA that converts the code field from a numeric item to a string item. You can use the \$ (string) function to convert the field. Then create a report that includes the define. Try to import the data again.

---

### **Question**

*I'm trying to send an URSA report to micro disk and am getting the message "DISK FULL." I know that it's not full, so what could the problem be?*

### **Resolution**

You may receive this message if you are running low on swap disk space. In this case, the user had numerous software packages open. Once the user closed the other packages, the report ran fine and sent the output to the microdisk.

---

### **Question**

*I run URSA in conjunction with the Micro Filter for TCP/IP and I've experienced problems when trying to send batch output to a micro disk file. What do I need to do to get this working?*

### **Resolution**

In URSA, you can send batch report output to a micro disk file on your PC or a micro printer. For example, you can send an URSA report in batchmode directly to a micro disk in Excel (.xls) format. To use this feature, the following must be true:

- You must be using the Micro Filter for TCP/IP on your PC.
- The Micro Filter for TCP/IP must be running on your PC.
- You must be using URSA Version 3.01.0027 or higher.
- You must be using DARGAL Version 60.340 or higher.
- The PC to which URSA is sending the output must be turned on and the Micro Filter for TCP/IP must be up and running at the time the report is run on the mainframe.
- When you are using the Micro Filter for TCP/IP, the host program on the A Series uses the host microcomputer's name or IP address to communicate with the Micro Filter. The Micro Filter for TCP/IP options in the **URSA Preferences** dialog box allow you to

send your machine's IP address to the URSA host so that it knows where to send the report output. You can also send the IP address or name of another computer that runs the Micro Filter for TCP/IP (to send reports to another microcomputer). One of these options needs to be set before you try to send output to micro disk or micro printer. You can access this dialog box by using the **Preferences** command (Options menu) in URSA.

*Note: The micro host — the microcomputer that communicates with the host through a direct TCP/IP connection and receives output — can be named in a user profile or setupfile. If a micro host is set in a user profile or setupfile and you send down a different IP address or name in URSA, the setting in URSA takes precedence.*

- When you run a report to micro disk or micro printer using the default settings, the **Micro Filter for TCP/IP** dialog box appears asking if it is okay to use the PC as the host. A user response is required in order to continue. If you do not want this notification dialog box to appear before the batch report begins to run, make sure that the access rights on the micro host PC are properly set. To prevent the dialog box from appearing, take these steps on the micro host:
  - Bring up the Micro Filter for TCP/IP.
  - Choose the **Access Rights** command (Options menu) in the **Micro Filter for TCP/IP** window.
  - Remove the check beside **Notify** in the **Access Rights** dialog box and choose **OK**.

(Access is allowed or denied based on the settings you made for Write and/or Print.)

*Note: If you need assistance setting up any of the options described here, please contact Client Services at 704-845-1000. There are two variations of Decision Support's Micro Filter for Windows: the original Micro Filter for Windows and the Micro Filter for TCP/IP. The feature described in this article does not work with the original Micro Filter for Windows (which includes those PCs that run terminal emulators such as HFS, INFOConnect, and FASTterm).*

## Formatting, headlines, output

### **Question**

*How do I double-space detail lines in URSA?*

### **Resolution**

1. In the **Format Detail Items** dialog box, highlight the first item in the **Detail Items** list.

2. Enter **1** in the box beside **Lines Before**.
  3. Choose **OK**.
- 

### Question

*I always want five spaces between my output fields. Is there a place where I can set an option like this?*

### Resolution

Yes, you can set this option in the **Report Options** dialog box. Follow these steps:

1. Choose the **Report Options** command (Options menu).
2. Enter a **5** in the box beside **Default Spaces Between Fields**.
3. Choose **OK**.

All options specified in the **Report Options** dialog box are saved in the **URSA.INI** file when you save your spec. They are retrieved when you open your spec.

---

### Question

*How do I display a 320-character string field in URSA? I've changed the display width to 320, but it's not working.*

### Resolution

In the **Format Detail Items** dialog box, change the width from 320 characters to 80 characters. Select the **Fold** format. It allows you to fold long text fields on the width specified with the **Characters** option so that long text fields print over multiple lines. Once you select the **Fold** format, the options **Wordwrap** and **Line Suppress** become available. **Wordwrap** prevents splitting words at the ends of lines. **Line Suppress** prevents printing blank lines at the end of a long text field. Check both of these options.

---

### Question

*How do I place the date and time on the beginning of my report?*

### Resolution

1. Double-click in the **Report Header** section in the **Layout** window.  
The **Select Fields - Report Header** dialog box appears.
  2. Choose the **System** button in the **Fields** section of the **Select Fields - Report Header** dialog box.
  3. Highlight **Date** and **Time** and choose the **Insert** button.
  4. Choose the **Close** button.  
The **Report Header** dialog box appears.
  5. If you are *not* changing the formatting of a title or modifying the items that appear in the **Titles** list, choose the **OK** button.  
If you are making changes, make them before you choose the **OK** button.
- 

## Question

*Can I extract the values entered for runtime input and place them in the page header?*

## Resolution

1. Double-click in the **Page Header** section in the **Layout** window.  
The **Select Fields - Page Header** dialog box appears.
2. Choose the **System** button in the **Fields** section of the **Select Fields - Page Header** dialog box.
3. Highlight **Selection logic** and choose the **Insert** button.
4. Choose the **Close** button.  
The **Page Header** dialog box appears.
5. If you are *not* changing the formatting of a title or modifying the items that appear in the **Titles** list, choose the **OK** button.  
If you are making changes, make them before you choose the **OK** button.

The value(s) that you enter for the input when you run the report will appear in the selection logic statement in the page header.

---

## Question

*One of my fields is showing up as a group header and group footer, and is being used in sort logic. I'm sure I didn't do this! How could this have happened?*

## Resolution

You probably inadvertently clicked on the field in the **Detail** section of the **Layout** window and dragged it into the **Group Header** section. When you set up a group header, URSA automatically sets up a corresponding group footer for the item. It also sets up sort logic based on the item. Just go into the **Groups** dialog box and delete the item from the **Groups** list.

---

## Question

*Sometimes we run reports where no records meet the selection logic. For example, every month we produce billing reports from the report log file. If a user did not execute any reports within the month, the report displays no output except the end of report message. We need to be able to show that there was no output for this user. Is there anything we can place in the report (in addition to the end of report message) to show that the report was run but no records met the criteria?*

## Resolution

You can display the selection logic in the report footer. This information along with the end of report message allows you to show that no records met the selection logic. Follow these steps to add the selection logic to the report footer:

1. Double-click in the **Report Footer** section in the **Layout** window.
  2. The **Select Fields - Report Footer** dialog box appears.
  3. Choose the **System** button in the **Fields** section of the **Select Fields - Report Footer** dialog box.
  4. Select the **Include** option **Strings**.
  5. Double-click on **Selection logic**.
  6. Choose the **Close** button.
  7. Choose **OK** in the **Report Footer** dialog box.
-

## Question

*I'm printing a report that contains detail data and one group footer. I want to skip a page after the footer, but the option in the Page Footer dialog box isn't working. What should I do?*

## Resolution

Since the report contains detail data and a group footer, it sounds as if you really want to skip a page after the group footer rather than the page footer. Select the option **Skip Page After Group** in the **Group Footer** dialog box to print each group on a separate sheet of paper. Page footers typically contain literal text, variable data (expressions), and options such as page number or date printed at the bottom of a page.

---

## Question

*I want several blank lines between my report header and page header. How do I set this up?*

## Resolution

In the **Report Header** dialog box, specify a blank by entering " " in the **Titles** list and choosing the **Add** button. Then enter a **4** (or however many lines you want) in the box beside **Lines Before**. You will then have several blank lines between your report header and page header.

---

## Question

*How do I set system options when sending output to a system disk file in URSA?*

## Resolution

Follow these steps to specify system options with system disk:

1. Choose the **Output** button on the toolbar.
2. Highlight **System disk** in the **Device** drop-down list box in the **Output Device** dialog box.
3. To specify an option, enter the option, an equal sign (=), and an option setting in the text box below **I/O Subsystem Options**. Separate multiple options with a comma.

For example, the following options automatically remove a file by the same name if it already exists and set security to private:

**REMOVE=TRUE, SECURITY=PRIVATE**

4. Choose **OK**.
- 

## Question

*How can I force column headings in output to system disk?*

## Resolution

Use the **Pagesize** option in the **Output Device** dialog box if you want column headings to appear in output sent to system disk files. The number you specify refers to the number of lines. Column headings appear after URSA prints the number of lines you specify. If you set **Pagesize** to 60, for example, the column headings will appear after every 60 lines of text. To display the headings only once at the beginning of the file, set **Pagesize** to a very large number — a number larger than the number of records in the output file. The maximum value you can specify is the maximum value allowed for an integer (about 549,000,000,000). Follow these steps to set the **Pagesize**:

1. Choose the **Output** button on the toolbar.
  2. Highlight **System disk** in the **Device** drop-down list box in the **Output Device** dialog box.
  3. Enter **PAGESIZE=60** in the text box below **I/O Subsystem Options**.
  4. Choose **OK**.
- 

## Question

*I need to know where to set the paragraph report option and I need more information about how it works. Where do I find this information?*

## Resolution

The Paragraph Report displays detail data for a particular record in a style similar to a paragraph. Each object in a paragraph consists of the name of a selected field, an equal sign (=), and the data in the field associated with that particular record. Each paragraph corresponds to an output record. URSA returns a paragraph for each record you select. This report type provides a way to display a large number of fields for each record. When you create a Paragraph Report, you apply the paragraph style to the Detail section of the report. You can also specify paragraph style for other sections of the report. You can specify paragraph style for some sections of a report and column style for other sections.

After you specify detail fields in the **Select Fields - Detail** dialog box, the **Format Detail Items** dialog box appears. Select **Paragraph** from the **Style** drop-down list.

You can specify paragraph style for a section of a report, even if other sections are column style. You can specify paragraph style in the **Group Header** dialog box or the **Group Footer**

dialog box. Select **Paragraph** from the **Style** drop-down list in either one of these dialog boxes.

See chapter 5 in the *URSA User's Guide* for more detailed information about paragraph reports and sections.

---

### Question

*I am not seeing any headings on some of the columns in my report. What might cause something like this?*

### Resolution

A heading does not appear if you enter an expression directly into a text box in a dialog box. URSA does not create a default heading for this type of expression. If you want a heading for the column, enter a heading beside **Title** in the dialog box where the expression is specified.

---

### Question

*When I send output from URSA to Microsoft Excel, the leading zeros are suppressed in my zip code field. How do I fix this?*

### Resolution

In Excel, select the cells you want to format. On the **Format** menu, choose **Cells**, and then click the **Number** tab. In the **Category** list, click **Special**, and then select the format you want to use. You can choose either **Zip code** or **Zip code + 4**.

## Extract Reports

### Question

*I've forgotten how to create and use extract files in URSA. What do I need to do?*

### Resolution

To create an extract file in URSA, create your report spec and set **EXTRACT = True** in the **Output Device** dialog box for **System disk**. Set this option in the **I/O Subsystem Options** text box and name the file in the **File Name** text box (**URSA/EXTRACT/DEPTS**, for

example). The extract file (**URSA/EXTRACT/DEPTS** in this example) is created when you run the report.

You can open an extract file in the **Open File** dialog box. URSA adds a \* to the end of the **Available Files** list in the **Open File** dialog box. When you choose \*, an **Extract File Attributes** dialog box appears. Beside **Attributes**, you can enter the name of the extract file after **NAME=**. (You can specify other attributes, too.) For example, if you were using our sample file, you would enter **NAME=URSA/EXTRACT/DEPTS** beside **Attributes**.

When you choose the file (press **OK** in the **Open File** dialog box), URSA assigns the file and all the fields you included in the extract file are available. You can then report from the file.

You can save and retrieve URSA report specs that report from extract files. The file name \* (which points to your extract file) is saved in the spec. You can join other files to the extract file — the extract file must be the primary file, though.

---

## Question

*Can I use an extract file in a join?*

## Resolution

You can join other files to the extract file. The extract file must be the primary file. Make sure the extract file includes all the fields you need to match to the secondary file's index.

You can open an extract file in the **Open File** dialog box. URSA adds a \* to the end of the **Available Files** list in the **Open File** dialog box. When you choose \*, an **Extract File Attributes** dialog box appears. Beside **Attributes**, you can enter the name of the extract file after **NAME=**. (You can specify other attributes, too.) For example, you would enter **NAME=URSA/EXTRACT/DEPTS** beside **Attributes** to open an extract file with that name. Then, open the file to which you want to join the extract file. Choose **Join** and proceed as usual.

## Batch reporting

### Question

*Does my PC need to be left on after I initiate a batch report in URSA?*

### Resolution

The answer depends on which device you specified for output.

When you run a batch report in URSA, DARGAL creates a program based on the URSA spec file. It sends off the job in a batch template file that includes all the commands necessary to run the spec in batch mode. If your output is going to a system device, you do not need to be concerned.

If you are sending output to a micro disk or microprinter and you use the original Micro Filter for Windows, we recommend that you use permanent directories. See your Unisys System Software manual for more information.

If you are sending output to a micro disk or microprinter and you use the Micro Filter for TCP/IP, the Micro Filter for TCP/IP must be running on your PC. Also, the PC to which URSA is sending the output must be turned on and the Micro Filter for TCP/IP must be up and running at the time the report is run on the mainframe.

See [Sending URSA batch report output to micro disk or micro printer](#) for more complete information about using the Micro Filter for TCP/IP to send batch report output to a micro disk or microprinter.

---

## Question

*We can run reports in URSA interactively without a problem. When we try to run them in batch, we get "MAINTENANCE FEE MORE THAN 30 DAYS OVERDUE." What's wrong?*

## Resolution

Check the batch template file you are using. Make sure that the statement that runs the host explicitly names the usercode and pack you are using.

---

## Question

*I run URSA in conjunction with the Micro Filter for TCP/IP and I've experienced problems when trying to send batch output to a micro disk file. What do I need to do to get this working?*

## Resolution

In URSA, you can send batch report output to a micro disk file on your PC or a micro printer. For example, you can send an URSA report in batchmode directly to a micro disk in Excel (.xls) format. To use this feature, the following must be true:

- You must be using the Micro Filter for TCP/IP on your PC.
- The Micro Filter for TCP/IP must be running on your PC.

- You must be using URSA Version 3.01.0027 or higher.
- You must be using DARGAL Version 60.340 or higher.
- The PC to which URSA is sending the output must be turned on and the Micro Filter for TCP/IP must be up and running at the time the report is run on the mainframe.
- When you are using the Micro Filter for TCP/IP, the host program on the A Series uses the host microcomputer's name or IP address to communicate with the Micro Filter. The Micro Filter for TCP/IP options in the **URSA Preferences** dialog box allow you to send your machine's IP address to the URSA host so that it knows where to send the report output. You can also send the IP address or name of another computer that runs the Micro Filter for TCP/IP (to send reports to another microcomputer). One of these options needs to be set before you try to send output to micro disk or micro printer. You can access this dialog box by using the **Preferences** command (Options menu) in URSA.

*Note: The micro host — the microcomputer that communicates with the host through a direct TCP/IP connection and receives output — can be named in a user profile or setupfile. If a micro host is set in a user profile or setupfile and you send down a different IP address or name in URSA, the setting in URSA takes precedence.*

- When you run a report to micro disk or micro printer using the default settings, the **Micro Filter for TCP/IP** dialog box appears asking if it is okay to use the PC as the host. A user response is required in order to continue. If you do not want this notification dialog box to appear before the batch report begins to run, make sure that the access rights on the micro host PC are properly set. To prevent the dialog box from appearing, take these steps on the micro host:
  - Bring up the Micro Filter for TCP/IP.
  - Choose the **Access Rights** command (Options menu) in the **Micro Filter for TCP/IP** window.
  - Remove the check beside **Notify** in the **Access Rights** dialog box and choose **OK**.

(Access is allowed or denied based on the settings you made for Write and/or Print.)

*Note: If you need assistance setting up any of the options described here, please contact Client Services at 704-845-1000. There are two variations of Decision Support's Micro Filter for Windows: the original Micro Filter for Windows and the Micro Filter for TCP/IP. The feature described in this article does not work with the original Micro Filter for Windows (which includes those PCs that run terminal emulators such as HFS, INFOConnect, and FASTterm).*

## Dates

### Question

*How do I add the date to the report header?*

### Resolution

1. Double-click in the **Report Header** section in the **Layout** window.

The **Select Fields - Report Header** dialog box appears.

2. Choose the **System** button in the **Fields** section of the **Select Fields - Report Header** dialog box.

Instead of containing a list of available fields, the dialog box now contains options available for display in the page and report headers and footers.

3. Highlight **Date** (if it's not already highlighted) and choose **Insert**.
4. Choose **Close**.
5. Choose **OK** in the **Report Header** dialog box.

You can specify the date in other sections of the report. Just double-click in the appropriate section of the **Layout** window and choose the **System** button in the **Select Fields** dialog box.

---

### Question

*Some of the dates in my URSA report are blank. An asterisk sometimes appears in output for a different date field. What does this mean?*

### Resolution

The substitution character for missing data in a date field is a blank (" "). Date-typed fields that have date formats appear blank in output if they contain missing data. Even though your other field contains date information, missing data appears as an asterisk (\*) because the field has not been date-typed — it is simply defined as a numeric field. If you want missing data in this field to appear blank (as with the other field), you need to flag the field as a date.

If you would rather use a different substitution character for missing dates, change the option in the **Report Options** dialog box (Report). You can specify either a one- or two-character string. If you enter a two-character string, both characters must be the same and URSA fills the output field with the character.

## Question

*I need to select records based on a date range. However, my month and year data are stored in separate fields. The month data is stored as a two-digit numeric field and the year data is stored as a four-digit numeric field. Is there any way to combine these fields and base a selection condition on them?*

## Resolution

Yes, create a numeric virtual field (define). Enter an expression similar to the following for the define text:

**(1#L-YEAR \* 100) + 1#L-MONTH**

Set the width to six digits. This expression creates a six-digit field that is a combination of the year portion of the date and the month portion.

Next, specify a selection condition based on the define you just created. To create selection logic that specifies a date range, choose the **BETWEEN** operator and then enter the beginning and ending dates as comparison values. For example, to select records for the dates March 01, 2001 through April 30, 2001, enter 200103 and 200104 in the **Comparison Values** box. When you run your report, all the records for those two months appear.

---

## Question

*I have a date field that represents when a vehicle is acquired (ACQ-DT) and another date field that represents when a vehicle is assigned (INV-CHG-DT). How can I determine the difference in number of days between two date fields?*

## Resolution

Apply the CENTURYDAY function to your date fields to obtain a numeric day count. Then, subtract the earlier date from the more recent date to return the difference in a number of days. Establish the calculation as a defined field in your report specification.

Create a define using the **Define Virtual Field** command (Data menu). Enter define text similar to the following:

```
CENTURYDAY(1#INV-CHG-DT) - CENTURYDAY(1#ACQ-DT)
```

The CENTURYDAY function converts the specified date to its centuryday form. The centuryday form of a date is the number of days elapsed since December 31, 1899 (the base date). Thus, if the INV-CHG-DT (date the vehicle was assigned) is May 6, 1998, the

centuryday value is 35920. If ACQ-DT (date the vehicle was acquired) is April 25, 1998, the centuryday value is 35909. In this case, the define returns a value of **11** for that record.

The CENTURYDAY function and all time/date functions require that the specified date field be in one of the following forms:

- It may be flagged as a date in the file description
- It can include a special character and a colon to indicate the form in which the value is stored.

For example, CENTURYDAY(J:1#INV-CHG-DT) indicates the date is stored in Julian form.

The **J**: preceding the field in the expression is mandatory when the field is not flagged as a date. See chap. 6 in the *DARGAL Reference Manual* for further information about time/date functions and use of special characters for non-flagged fields.

## Joins

### **Question**

*I want to create a join in URSA but I don't need to match all the key items. Is there a special option I need to set?*

### **Resolution**

After you match the key items you do need, choose the **Options** button in the **Joins** dialog box.

Select the **One-to-many (Select)** option in the **Multiplicity** section of the **Joininfo** dialog box and choose **OK**.

When you create a one-to-many join, you match some of the major key items (but not all of them). Since you do not match all the key items, you can expect to read more than one record from the secondary file.

When URSA reads the primary file in this type of join relationship, it builds a partial key for the secondary file. The key is then used to read all the records from the secondary file that match it. The primary record and the secondary record are returned as a single logical record to the reporting cycle. The same primary record is returned for each secondary record associated with it.

---

### **Question**

*Can I use an extract file in a join?*

### **Resolution**

You can join other files to the extract file. The extract file must be the primary file. Make sure the extract file includes all the fields you need to match to the secondary file's index.

You can open an extract file in the **Open File** dialog box. URSA adds a \* to the end of the **Available Files** list in the **Open File** dialog box. When you choose \*, an **Extract File Attributes** dialog box appears. Beside **Attributes**, you can enter the name of the extract file after **NAME=**. (You can specify other attributes, too.) For example, you would enter **NAME=URSA/EXTRACT/DEPTS** beside **Attributes** to open an extract file with that name. Then, open the file to which you want to join the extract file. Choose **Join** and proceed as usual.

---

### **Question**

*I'm trying to create a "join on the fly" in URSA, but I'm getting the message "JOINS NOT PERMITTED." I do this all the time. What might be wrong?*

### **Resolution**

Check to make sure that you are using the correct usercode or username. In this particular case, another user had logged in before the user who got the message. The previous user is not allowed to create "joins on the fly." The user who got the message logged in again with his own username and everything worked as expected.

## Aggregates, totaling

### **Question**

*Where do I access the aggregate functions in URSA?*

### **Resolution**

The five main aggregate functions are TOTAL, MEAN, MIN, MAX, and STDEV. The summary buttons in the **Select Fields - Group Footer** and **Select Fields - Report Footer** dialog boxes represent these functions as Total, Avg, Min, Max, and Std dev. Each aggregate function returns one value that summarizes a set of values. TOTAL returns the sum of a set of values. MEAN returns the average. MAX returns the largest value in the set. MIN returns the smallest value. STDEV returns the standard deviation. You can also access functions from the **Functions** button that is available in the **Define** and **Define: Selection Logic Formula** dialog boxes. (The aggregate functions are also called summary functions.)

### Question

*Is there a quick way to place a grand total on a report?*

### Resolution

Press and hold the CTRL key while you drag and drop the object from the **Detail** section into the **Report Footer** section. URSA copies the object. Since you're creating a grand total, you'll want to increase the object's width. You can click and drag an object's handle (small square in the lower right corner) to make the object's width larger.

---

### Question

*When totaling a field from my file, I get the message "**RECORDS NOT ACCUMULATED DUE TO ACCUMULATION ERRORS**" at the end of my report. What does this mean?*

### Resolution

You'll get the message "**RECORDS NOT ACCUMULATED DUE TO ACCUMULATION ERRORS**" when you total a field that contains missing or invalid data. This is very common if you are totaling a field from a secondary file in an outer (optional) join. When you use this type of join, the record from the primary file is returned and the secondary record is set to missing when there is not a match. URSA does not include records that have been set to missing in totals.

---

### Question

*I'm creating a report where I group data based on a code field. For some reason, the code field is showing up in the **Group Footer** section, too, and is being totaled? Why is this happening?*

### Resolution

You probably double-clicked on the code field in the **Select Fields - Group Footer** dialog box. If you do not choose one of the summary buttons and just double-click, URSA sets up a total for the field. To show the value of the code field instead, highlight the field in the **Select Fields - Group Footer** dialog box and choose the **Value** button.

---

## Question

*What is the significance of the Value button in the Select Fields - Report Footer dialog box?*

## Resolution

Although the **Value** button appears below the summary buttons, it is not one of them. If you choose the **Value** button, the current value of the field highlighted in the fields list will be used as a label. You can also use the **Value** button to specify literal text as a label, such as the words "Subtotal," "Dept. Total," or "Grand Total."

*Note: If you need more specific information, see "Labeling summary data" in the **URSA User's Guide**.*

---

## Question

*I need to get a total of all occurrences in my occurring field. What do I need to do? What do I do if I want to total only the first five occurrences?*

## Resolution

When you want to total items in an occurring field, use the TOTAL function. To total all the occurrences of a field named **ITEM-AMT**, for example, use an expression like the following:

**TOTAL(1#ITEM-AMT)**

To total just the first five occurrences of the field **ITEM-AMT**, enter an expression like the following:

**TOTAL(1#ITEM-AMT,5)**

## Managing specs

## Question

*Can I list the contents of an URSA spec to see the fields, formats, etc. that were specified?*

## Resolution

You can generate a DARGAL program and then list its contents to see the fields, formats, etc.

1. Choose the **Generate DARGAL Program** command (File menu).

2. Beside **File Name** in the **Generate DARGAL Program** dialog box, specify a name for the program.
3. Choose **OK**.

Then, to list the program from within URSA, follow these steps:

1. Choose **Host Commands** (File menu).
2. Specify **LIST <program name>** beside **Command** in the **Host Commands** dialog box.

For <program name>, use the name you specified in the **Generate DARGAL Program** dialog box.

3. Choose **OK**.

The program appears in a **Browse** window. You can print the listing by choosing **Print** (File menu) in the **Browse** window.

---

## Question

*I know that I can import EZSPEC report specs into URSA, but I don't know how to do it. What are the steps?*

## Resolution

1. Choose the **Import EZSPEC Specifications** command (File menu).

The **Import EZSPEC Specifications** dialog box appears.

2. Enter the name of a host directory in the **Host Directory** text box and choose the **Retrieve** button.

URSA displays a list of the EZSPEC report specs that are in the directory you named.

3. Select a spec from the list and choose the **OK** button.

URSA attempts to import the spec. If it is successful, the spec becomes your current URSA spec.

4. Save the spec.

See the *URSA User's Guide* for more information about importing EZSPEC report specs into URSA, including information about specs that must be modified, etc.

---

## Question

*I imported an EZSPEC report spec into URSA, but the record is wrapping at column 132. Are my EZSPEC settings not carried over?*

## Resolution

When you import an EZSPEC report spec into URSA, URSA's default fold point of 132 is used. It is not overwritten by any settings you may have specified in your EZSPEC report spec. You can change the fold point for any output device by specifying a value with the **Column Fold Point** option in the **Output Device** dialog box. There is a different **Output Device** dialog box available for each device.

---

## Question

*Can I set a default working directory for URSA specs?*

## Resolution

Beginning with Version 3.00.236, you can name a default directory for saving report specs. The steps follow:

1. Choose the **Preferences** command (Options menu).
2. Use the **Default Spec Directory** text box to name the directory where you want to store saved report specs. Use the **Browse** button to search for a directory.
3. Choose the **OK** button.

The default spec directory as well as all options specified in the **Preferences** dialog box are saved in the **URSA.INI** file.

---

## Question

*Is there a way to export an URSA report spec to EZSPEC format?*

## Resolution

Rather than exporting the spec to EZSPEC, you can generate a DARGAL program for the spec. You can modify it and run it like you do any other DARGAL program. Choose the **Generate DARGAL Program** command (File menu) to name the file that will contain the DARGAL code.

---

## Question

*How do I convert my old 16-bit URSA specs into 32-bit URSA specs?*

## Resolution

To use your old 16-bit specs with the latest 32-bit version of URSA, you must do the following:

1. Load the latest 16-bit version of URSA.
2. Run the latest 16-bit URSA.

*Note: We recommend that you update specs offline. Disconnect from the host before beginning the update.*

3. Choose the **Update Specs** command (File menu).
4. In the **Open** dialog box, specify the specs to update and choose **OK**.
5. Load the latest 32-bit version of URSA.

Once you update a spec in the latest 16-bit version of URSA, you can use it in either the latest 16-bit version or the 32-bit version. We recommend the 32-bit version. You can download the latest versions of URSA from our ftp site, <ftp://ftp.dsinc.com>.

---

## Question

*When I try to import an EZSPEC report spec into URSA, I see the message "BEGINNING IMPORT," but then nothing happens. What might the problem be?*

## Resolution

Before you try to import a spec, make sure that you are using the correct master for the spec. When you see the "BEGINNING IMPORT" message and then nothing happens, it usually means that something is wrong with the master.

## Calculations, defines, expressions

## Question

*How do I display a record number for each record in the file from which I am reporting?*

## Resolution

Follow these steps to add a counter:

1. Double-click in the **Detail** section in the **Layout** window.  
The **Format Detail Items** dialog box appears.
2. Enter **(READCOUNT)** in the **Detail Items** text box and choose **Add**.  
URSA adds **(READCOUNT)** to the **Detail Items** list.
3. To add a column heading for the expression, highlight **(READCOUNT)** in the list, enter **Count** (for example) beside **Title**, and choose **OK**.

When you run the report, the record number (count) appears for each record URSA returns for the report. For a running count of only the records selected for the report, use **(SELECTCOUNT)**.

---

### Question

*I'm trying to use the SQLCOUNT function in URSA, but it does not appear to be a supported function. Is there a way to use it in URSA?*

### Resolution

The SQLCOUNT function allows you to return the number of unique values found in the valid occurrences of a field. You can use the SQLCOUNT function in URSA at end of group (Group Footer) or at end of report (Report Footer). See [Getting a count of unique values](#) for an example that uses the SQLCOUNT function.

---

### Question

*I need to get a count of the unique account numbers in my report. I want to disregard multiple instances of the same number. How do I do this?*

### Resolution

Use the SQLCOUNT function with the DISTINCT option to return the number of unique values found in the valid occurrences of a field. Follow these steps:

1. Double-click in the **Report Footer** section in the **Layout** window.  
The **Select Fields - Report Footer** dialog box appears.
2. Enter **(SQLCOUNT (DISTINCT 1#PAT-ACCT-NBR))** in the text box above the fields list and choose **Add**.

3. Select **(SQLCOUNT (DISTINCT 1#PAT-ACCT-NBR))** and choose the **Value** button.
4. Choose the **Close** button.

The **Report Footer** dialog box appears.

You may want to add a literal expression to label the count.

5. If you are *not* changing the formatting or modifying the items that appear in the **Summary Items** list, choose the **OK** button.

If you are making changes, make them before you choose the **OK** button.

---

## Question

*I have a 16-character string field. How do I extract the first two characters and then the next ten separately?*

## Resolution

You need to define two separate string virtual fields. For the first one, enter define text similar to the following to extract the first two characters:

**1#CODE.(1,2)**

Enter define text similar to the following to extract the next ten characters:

**1#CODE.(3 THRU 12)**

---

## Question

*How can I select only records that contain any one of three particular string expressions? For example, I want to select records that contain a comma, an exclamation point, or an asterisk in a particular field.*

## Resolution

You need to create a selection logic formula to select only those records from the file. Follow these steps to enter the formula:

1. Choose the **Select** button on the toolbar.
2. Choose **Close** in the **Select Fields - Selection Logic** dialog box.
3. Choose **Formula** in the **Selection Logic** dialog box.

4. Enter text similar to the following in the **Define - Selection Logic Formula** dialog box.

**"," IN 1#COMMENT OR "!" IN 1#COMMENT OR "\*" IN 1#COMMENT**

5. Choose **OK** in the **Selection Logic** dialog box.
- 

### Question

*How do I create a report with weighted averages in URSA? I need to calculate a weighted average on a loan interest rate by account type.*

### Resolution

Create a summary report and then multiply the total principal balance by the interest rate and divide the result by the total principal balance. Specify **ACCT-TYPE** as your **Group #1 Header**. Then enter the following expression in the text box in the **Group #1 Footer** dialog box:

**(TOTAL(#PR-BAL \* #INTR-RATE) / TOTAL(#PR-BAL))**

Choose the **Add** button. URSA adds the expression to the **Summary Items** list. Add a label like the following by entering it in the text box:

**" Weighted average interest rate for account type:"**

To display the current value for **ACCT-TYPE**, choose the **Fields** button. In the **Select Fields - Group Footer** dialog box, highlight the field **ACCT-TYPE** and choose the **Value** button.

---

### Question

*I'm trying to enter an IF . . . THEN statement in a text box above an item list. It is not accepting my syntax because I'm not specifying an ELSE. I know I need to specify an ENDIF with the command, but I didn't think I needed an ELSE. What's wrong?*

### Resolution

If you need to enter an IF . . . THEN expression in a text box, you need to use the IF function rather than the DARGAL IF command. The syntax for the IF function requires that you enter an ELSE parameter. ENDIF is not part of the syntax at all.

---

### Question

*I need to add a condition to the middle of my selection logic list. URSA keeps putting it at the bottom. Why?*

### Resolution

The conditions that you specified previously are probably grouped. To add a new condition that comes between these, ungroup the existing conditions. Select all the conditions in the group and then choose the **Ungroup** button.

---

### Question

*I get an "Invalid character entered" message when I try to enter multiple values for my AMONG selection condition. Why can't I do this?*

### Resolution

When you need to enter multiple comparison values for a selection condition (as with AMONG or BETWEEN conditions), enter each value separately. Enter your first value, press the Enter key, and then enter your next value. Continue this process until you are finished.

---

### Question

*In URSA, how do I produce a report of customers who have water service (2#SERVICE = "SV") but do not have refuse service (NOT #SERVICE="RF")? I am working with a join between the customer file and the services file. It's a one-to-many join since there is a separate record for each service in the services file.*

### Resolution

You need to set up a SKIPIF expression in a **Group Footer**. The SKIPIF expression allows you to pass over output based on a logical expression. You also want to report the information in the **Group Footer**, rather than in the **Detail** section.

First, create a group defined by a unique identifying field such as customer ID. Display the information you want for each customer in the **Group Footer**. Also in the **Group Footer**, add the following expression as the first expression:

```
(!SKIPIF TOTAL(2#SERVICE = "RF" : 2) AND NOT TOTAL(2#SERVICE = "WA" : 2))
```

Enter this text in the box over the list of expressions in the **Group Footer** dialog box. The exclamation point passes the text that follows it directly as DARGAL syntax, rather than as a literal or expression. You may need to turn off the **Align with detail** option to make sure this expression appears first in the **Group Footer**.

The SKIPIF expression includes **TOTAL** functions that accumulate the occurrences of the values "WA" and "RF" in the field 2#SERVICE. (The ": 2" syntax assures that the accumulations are for file 2 only.) Because they are used in the Group Footer, the **TOTAL** functions will evaluate all the service records for each customer.

If *any* service record contains the given value, the **TOTAL** function will return a positive value, which is equivalent to a logical value of **True**. The **NOT** operator that is applied to the second **TOTAL** function reverses the logical value.

The first **TOTAL** function skips customers who have refuse service. The second skips customers who do not have water service. This means the report will *include* customers who have water service but do not have refuse service.

---

## Question

*Can I randomly sample data in URSA?*

## Resolution

URSA allows you to include records in (or exclude records from) a report by using the following types of sampling logic:

- every nth record sampling (for example, to include (or exclude) every 10th record)
- random sampling (for example, to include (or exclude) 20% of the records)

You can also terminate a report early based on the following:

- the number of records selected
- the number of records read

Before you begin to specify sampling conditions, open your data file and specify report contents.

Then follow these steps:

1. Choose the **Selection Logic** command (Data menu).

If you are defining selection logic for the first time, URSA displays the **Select Fields - Selection Logic** dialog box.

2. If you are specifying selection conditions on fields (in addition to sampling conditions), select the fields you want and choose **Close**. Otherwise, just choose **Close**.

URSA displays the **Selection Logic** dialog box.

3. To specify sampling conditions, choose the **Sample** button.

The **Sampling Conditions** dialog box appears.



4. Specify any sampling conditions you need.
5. Choose **OK** when you finish specifying conditions.

The Include options allow you to include every  $n$ th record or a random percentage of records in your report. (If you want to exclude records, choose the **Exclude** option button.)

To specify **every  $n$ th record**, follow these steps:

1. Select the check box beside **Every**.
2. Enter a number to indicate how often URSA should sample the file.

For example, enter **10** to select every 10th record.

(When you specify this type of condition, URSA uses the READCOUNT function with the MOD operator to select the records. In this example, it uses  $\text{READCOUNT MOD } 10 = 0$ .)

To specify a **random sample**, follow these steps:

1. Select the check box beside **Random**.
2. Enter a number to indicate the percentage of records. For example, enter **20** to sample 20% of the records.

(URSA uses the RANDOM function to generate the sample. In this example, it uses

RANDOM  $\leq$  0.2. The percentage of records selected is approximate. The larger the number of records, the more accurate the sample.)

*Note: You cannot specify every nth record sampling and random sampling at the same time.*

The **Report Until** options allow you to terminate a report when the host selects or reads a specified number of records.

To **terminate a report when a number of records have been selected**, follow these steps:

1. Select the check box beside **Records Selected**.
2. Enter a number. For example, enter 25 to stop the report after URSA selects 25 records.

(URSA uses the UNTIL condition with the SELECTCOUNT function to stop the report. In this example, it uses UNTIL SELECTCOUNT  $\geq$  25.)

To **terminate a report when a number of records have been read**, follow these steps:

1. Select the check box beside **Records Read**.
2. Enter a number. For example, enter **50** to stop the report after URSA reads 50 records.

(URSA uses the UNTIL condition with the READCOUNT function to stop the report. In this example, it uses UNTIL READCOUNT  $\geq$  50.)

*Note: You can specify both **Report Until** conditions (Records Selected and Records Read) at the same time. You can also use the **Include** (or **Exclude**) options and the **Report Until** options at the same time. For example, you can get a 20% sample of the first 25 records selected or the first 50 records read.*