TIBCO Foresight® EDISIM®

Self-Paced Tutorial: Introduction to EDISIM® EDIFACT D99A Orders

Version 6.20.0

May 2021

Contents

Overview of TIBCO Foresight EDISIM®

EDISIM® is a guideline and business rule management tool that allows organizations involved in EDI transaction processing to build, publish and test companion guidelines.

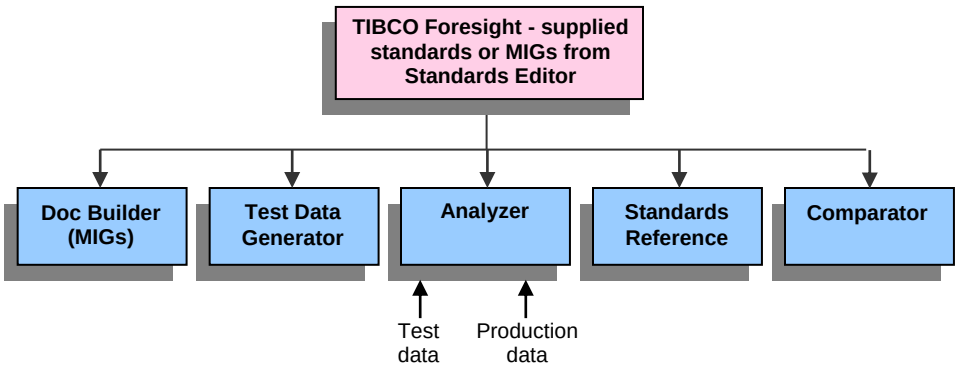
Standards Editor Coordinates Other Tasks

Standards Editor lets you:

- Edit existing EDI standards to create new MIGs (Message Implementation Guides) that you and your trading partners will use.
- Create whole new base standards for a group, industry, or committee.

After creating a MIG, you can use:

- EDISIM Doc Builder to print formatted documentation from it.
- EDISIM Test Data Generator to generate test data from it.
- EDISIM Standards Reference to browse through it.
- EDISIM Analyzer to check EDI data for adherence to it.
- EDISIM Comparator to compare it to other messages, or to migrate user changes between it and another MIG.



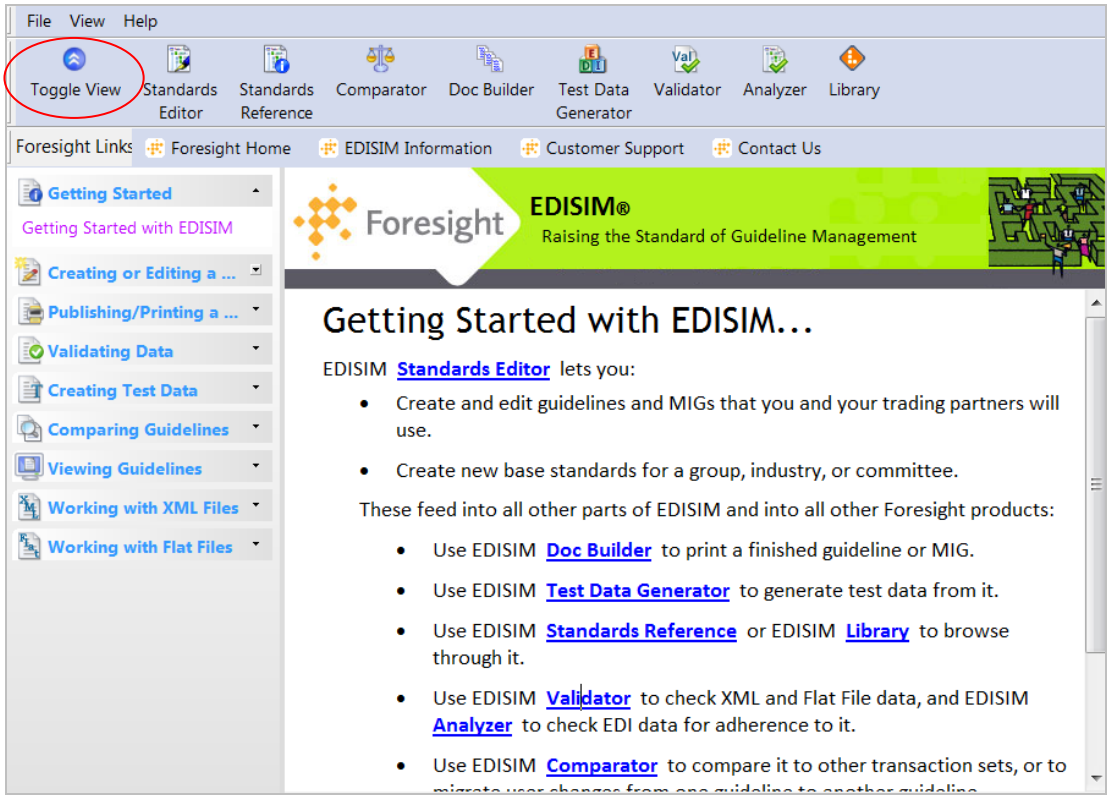
EDSIM Application Manager

The EDISIM Application Manager is a comprehensive user interface with access to all EDISIM programs plus lists of tasks and information about how to accomplish them.

START

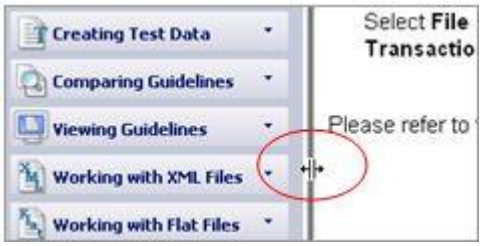
1. From your desktop, click this desktop shortcut:

2. The Application Manager appears. Click the Toggle View icon in the Application Bar and note how the task area can be toggled between a full and collapsed view. Click Toggle View again to return the screen to the full view.

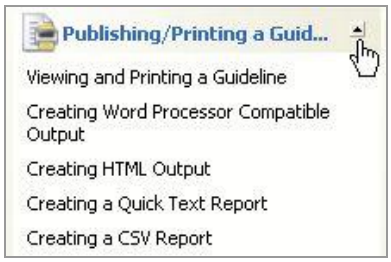


Also notice the other items in the Application Bar. Clicking one of these icons launches the associated EDISIM component.

3. The Task Pane appears on the left side. Adjust the width of the pane by dragging the splitter bar.



4. In the **Publishing/Printing a Guideline** task category, click the navigation arrow to see the full view of associated tasks.



5. Click the task **Creating Word Processor Compatible Output**.

   The right pane displays an overview of that task, including links to start the required EDISIM application and links to display the related documentation.

   In this case, Doc Builder is the right application for the task.

STOP

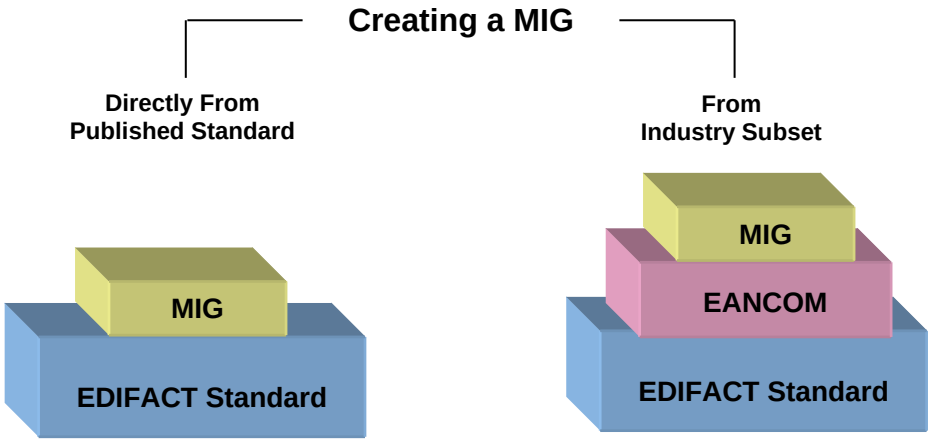
Comments

- You are not required to use the Application Manager; you can start any component directly from the start menu (**Start | All Programs | *<TIBCO_HOME>* | EDISIM**).

Standards Editor

Purpose

The purpose of this exercise is to customize a D99A Purchase Order Message (ORDERS) so that it matches the exact specification that trading partners will use. As you customize the ORDERS message, you will select the segments and elements to be used, and will define which specific codes will be acceptable in these elements. Once you have customized this standard, you can use it to generate MIGs and test data, and as a basis for analyzing actual or test EDI data for conformance.



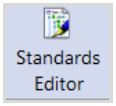
Comments

- Before starting exercises:
  - The student guide is yours to keep. You can write on it.
  - Exercises are broken into small units. Wait for instructor at each STOP.
  - Pictures are representative but may not always match exactly.
  - EDISIM uses standard mouse functions: left button click for selection and double-click for processing. Click+drag for multiple selection.
  - Exercises don't always stop on important parts. Don't just click blindly to finish quickly.
- Flow and purpose of course exercises:
  - Customize an ORDERS message for specific use (Standards Editor).
  - Print a customized MIG for this ORDERS message (Doc Builder).
  - Compare two standards or MIGs (Comparator).
  - Analyze data (Analyzer).
  - Generate ORDERS test data (Test Data Generator).
  - Use on-line reference (Standards Reference).

Selecting a Base Standard and Transaction Set

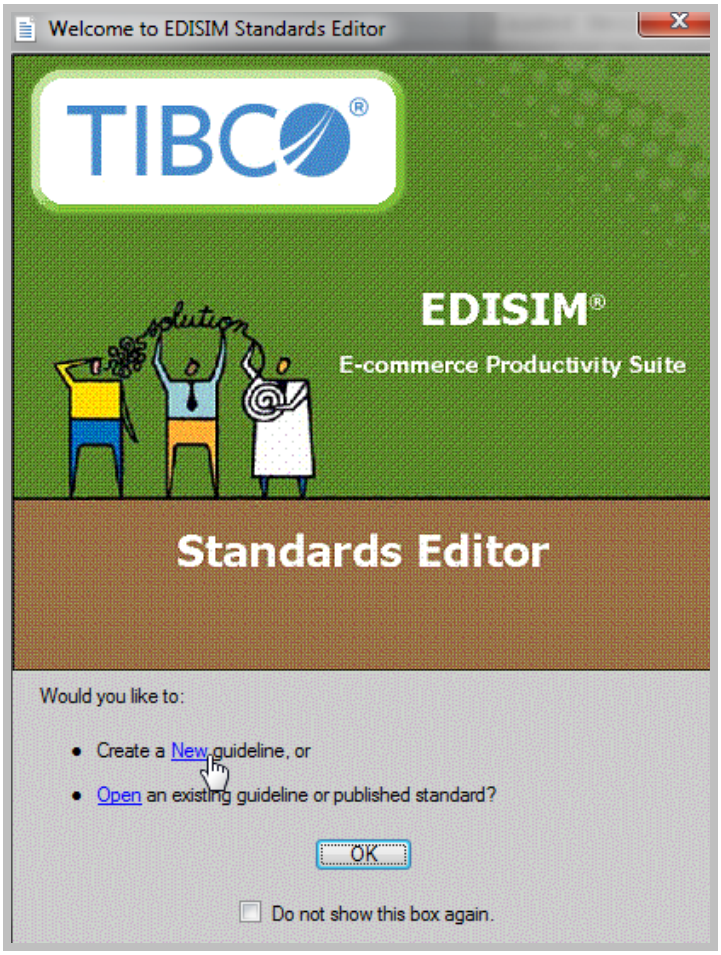
The first step is to select the base standard and transaction set for our new guideline.

START

1. If using the Application Manager, click the Standards Editor icon: 

   Otherwise, choose **Start | Programs | <*TIBCO_HOME*> | EDISIM | Standards Editor**.

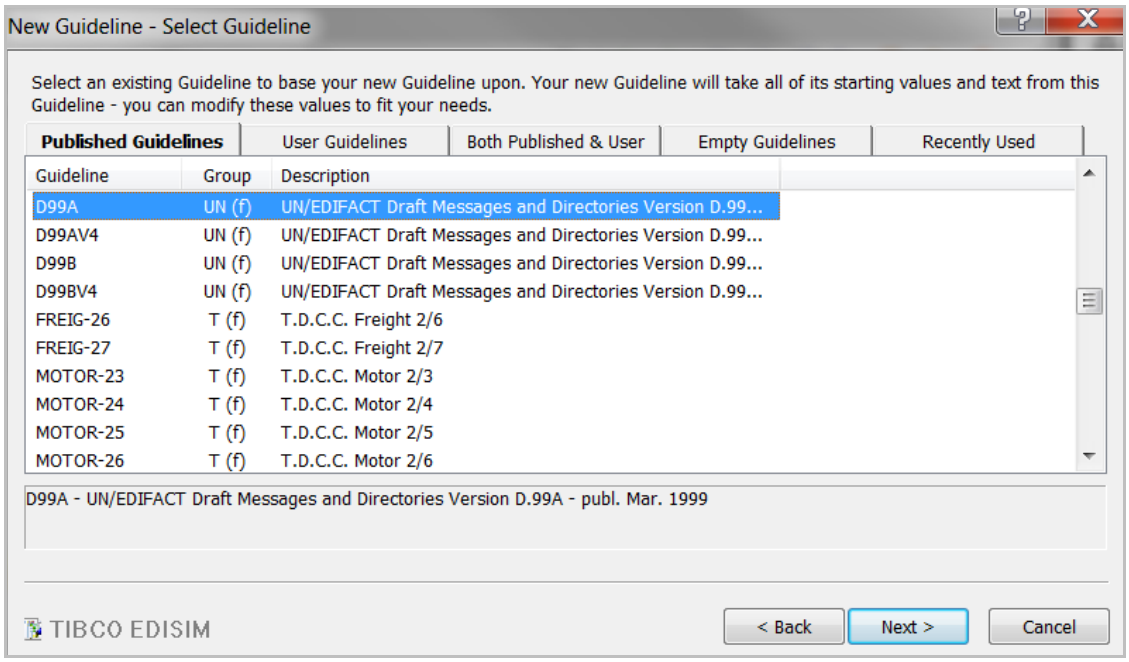
   Standards Editor displays a welcome screen.



2. Choose the **New** link within the “Create a New guideline” text on the welcome screen.

   The standards selected during installation are listed. Notice the tabs at the top of the box.

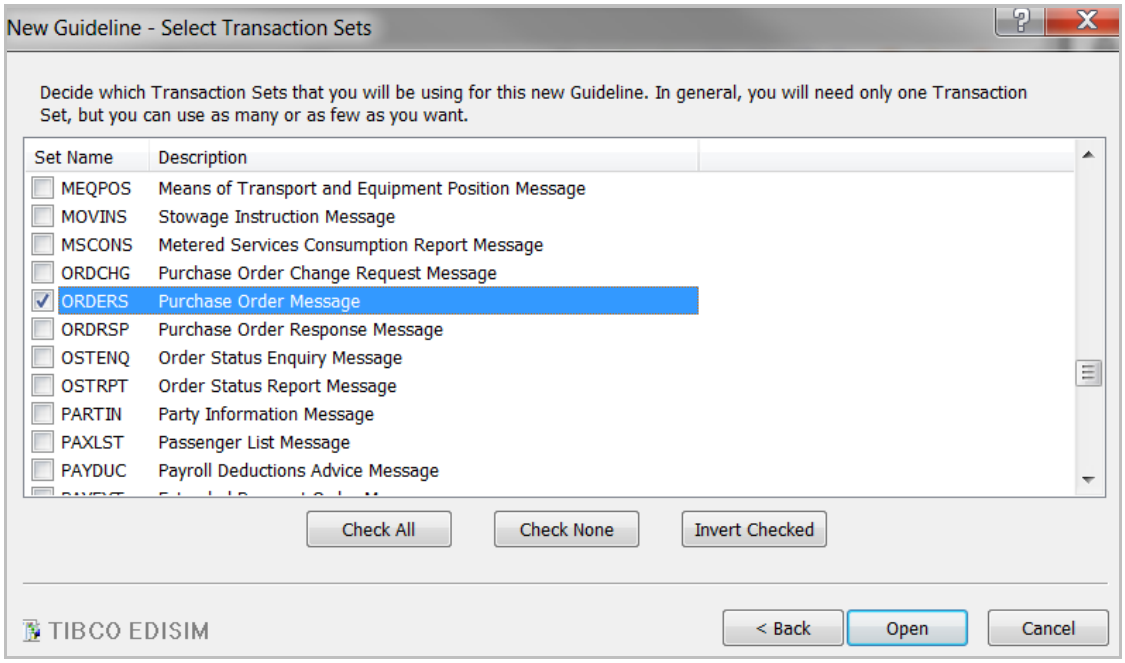
3. Scroll until you see the D99A standard. (Type D99A to speed search to the standard D99A, or press the ***Pg Dn*** key.)



4. Select the **D99A** standard and click **NEXT**.

5. Scroll until you see the D99A **ORDERS** message. (You can type **ORDERS** to speed search directly to the ORDERS message.)

6. Check ORDERS, or use the space bar on your keyboard.



7. Click **Open** in the Selection Transaction Sets dialog box.

   The new standard will be based on the D99A ORDERS

STOP

Comments

- Use **File | New** when first starting a new guideline. Use **File | Open** when editing it later. You can also use the first two buttons on the toolbar (New and Open).
- You can select more than one transaction set. Use *Check None* if you want to edit only the Dictionary Objects. Use *Invert Checked* if you want to use many transaction sets, omitting only a few.
- When moving through lists, you can use the scroll bar, the *Arrow* keys, or the *Page Up* and *Page Down* keys.

Saving Your Work and Customizing Note Levels

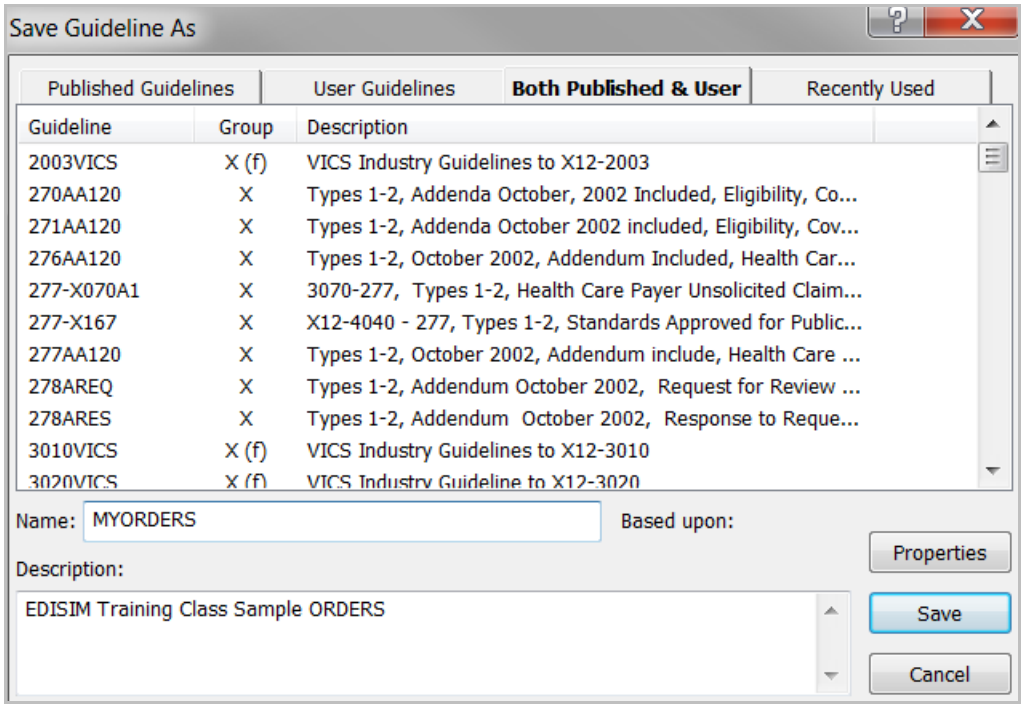
START

1. Choose **File** | **Save As** or use the Save button on the toolbar.

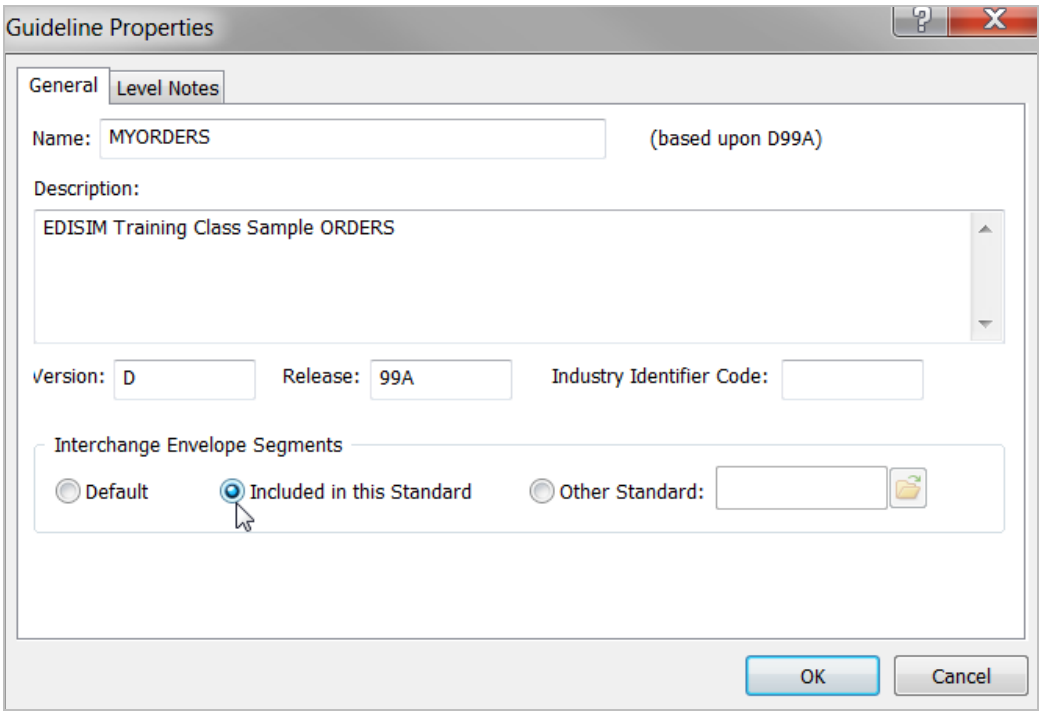
2. In the **Save Guideline As** dialog box, enter **MYORDERS** for the Name (DO NOT PRESS the *Enter* key).

   Note: Guideline names can have up to 128 alphanumeric characters, underscores, and hyphens.

3. Press the ***Tab*** key to move to the Description field, and then replace the existing text with **EDISIM Training Class Sample ORDERS**.

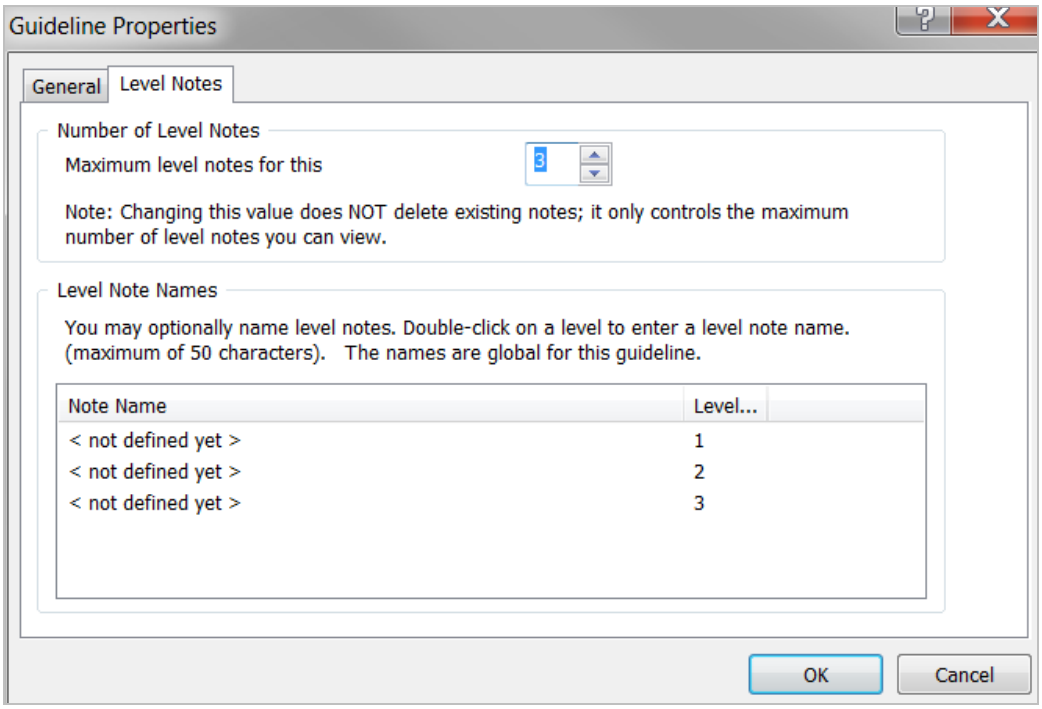


4. Click the **Properties** button to view the Version, Release, and Industry Identifier Code. Keep the default of 'Included in this Standard' for the Interchange Envelope Segments, since they are found in this guideline's dictionary.



5. You may also define how many level notes are available for this MIG by selecting the **Level Notes** tab. (See Advanced Features for more on Level Notes.)

   You can actually have 100 levels. For this exercise, leave the value at **3**.



6. Click **OK**. Then click **Save**.

This MIG is saved under **EDISIM \User Files\Public Guidelines.**

STOP

Comments

- After naming a file, you can also save it by clicking the Save button  on the toolbar.
- The Interchange Envelope Segments selection tells Analyzer and Validator where to find the interchange and group enveloping:
  - Default - use generic enveloping definitions in UN1ICS if enveloping segments are not in the dictionary.
  - Included in this Standard - use enveloping segments in the dictionary.
  - Other Standard - use enveloping segments in standard xxx's dictionary.

  Analyzer and Validator will look at this setting if you select Analyzer's or Validator's "use one standard" option.

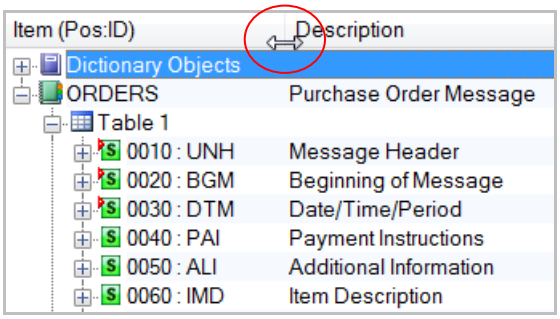
Navigating

START

1. Maximize Standards Editor.

2. Expand the Orders Transaction Set by clicking the **+** symbol that is in front of the rows for **Orders, Table 1**, **0080 Loop**, and **Table 2**.

3. Drag the splitter bar between the first and second columns to the right to show all information that is in the first column:



4. Find and expand the **Table 2** loop **1010**.

5. Look at the column headings just below the toolbar. Depending on your resolution, you may need to use the horizontal scroll bar at the bottom of the pane to see other columns. The last column is **Indicators**.

STOP

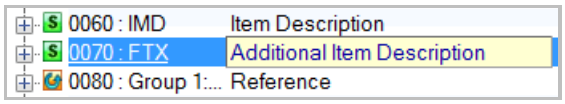
Changing Descriptions

You can change descriptions of any object to better describe their purpose. We are going to change a description for a field that can be used to provide additional descriptive information for the item.

START

1. We would like to rename the 0070 FTX (Free Text) object so that users will know they can provide additional information about the item.

2. Click on the Table 1 0070:FTX *segment's* Description field, and change it to **Additional Item Description**.



3. Click off the field to see your change.

STOP

Comments

- Renaming certain objects helps avoid confusion later.
- Keep your new description close to the assigned description.

Marking Segments as Used/Not Used

When you designate something as not used, you will get an error message if that item is in an EDI file.

- The **Req** column in the top pane shows the requirement according to EDIFACT.
- The **U/A** (user attribute) column shows the requirement agreed upon by you and your trading partner.

You can change the requirement of loops, segments, and elements that are Optional. To mark an object not used or toggle it back to used, you have these choices:

Edit | Mark Used/Not used

 button on the toolbar

Alt+M

Edit | User Attributes

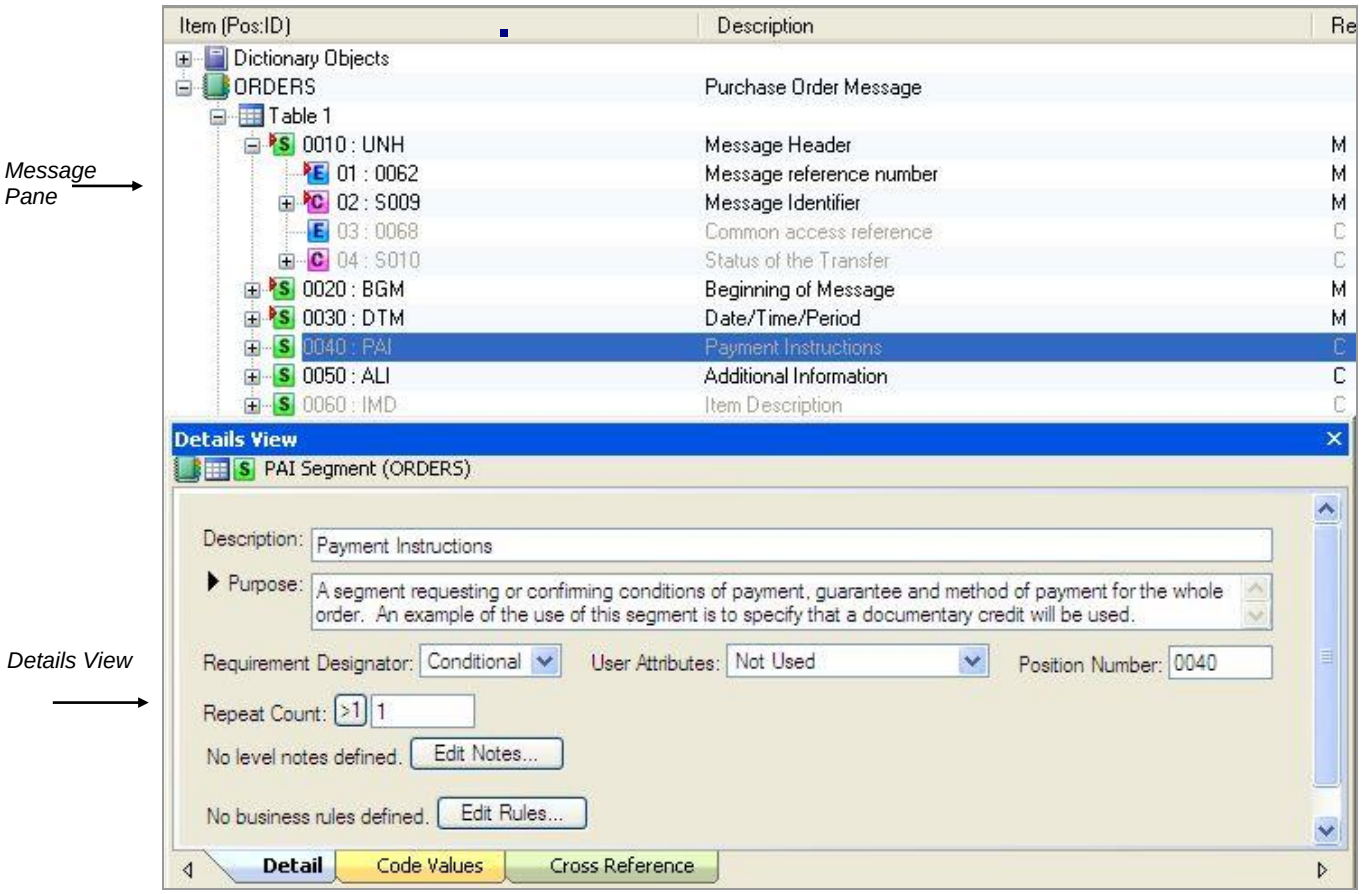
START

1. Expand the ORDERS message to a tree view by clicking the **+** symbol that is before the **ORDERS**, **Table 1**, **Table 2**, **LIN Group**, and **Table 3**. All segments are displayed that are not part of a group.

2. Click the **UNH** segment at position 0010 in Table 1, and using the scroll bar, scroll to the **UNT** segment at position 2350 in Table 3. Holding the Shift Key, select the UNT segment to highlight all the segments between the UNH and the UNT segments.

3. Select **Edit | Mark Used/Not used**. Click in the highlighted area and notice that all the optional segments are faded, and the mandatory segments are not.

4. Scroll to the **ALI** segment at position 0050 in Table 1. Right click and select Mark Used/Not Used. Click on the PAI segment at position 0040 in Table 1 to see that the ALI segment is now highlighted for use.



5. Look at the status bar just below the toolbar to see the Requirement Designator, the User Attributes, the Repeat, and the Indicator for each segment.

6. Select the IMD segment at position 0060. Use the ***Alt+M*** keys on your keyboard to mark it as Used.

7. Click the Save icon to save your work.

Comments

- Normally, use Mark Not Used rather than Delete.
- Leave **View | Show Unused** on when editing.
- A faded segment is not used.
- An N in the U/A column means the segment was specifically marked not used.
- A segment that is faded but has no N in the U/A column is part of an unused loop.

Marking Groups as Not Used

START

1. Scroll down and click on **GROUP 12** at position 0440 to select it, then scroll down until you see GROUP 28 at position 1010. (Use the scroll bar, NOT the *Page Down* key.)

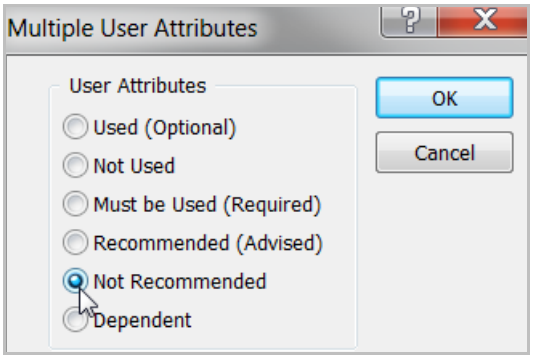
2. Hold down the ***Shift*** key and click on **GROUP 28**.

   Groups 12 through 28 are now selected.

3. Choose **Edit | User Attributes**.

   The Edit User Attributes dialog box appears. This is another way to mark multiple items not used, but it gives you other choices also.

4. Click the **Not Recommended** button.



5. Click **OK**.

6. Click a segment to deselect the previously selected segments.

   The first segment in a group functions as the group “trigger” segment, and its user attributes cannot be changed.

7. Click the **Save** button on the toolbar.

STOP

Comments

- Selecting methods:
  - To select scattered items: *Ctrl*+click.
  - To select a continuous range: Click on the first one, then hold down *Shift* and press the *Down Arrow* to the end, or click one end of the range, then *Shift*+click the other end.
- Clicking the right mouse button also opens the Edit menu selections.

Changing a Segment's Requirement

Search  = Search the ID column

Edit | Find or  = Search the Description column

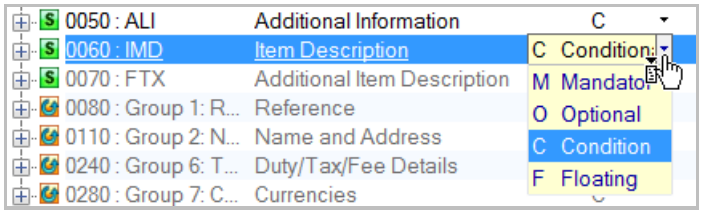
To be found, the item must be showing. It cannot be in a collapsed loop.

START

1. Let's use speed search to locate a segment. Type **IMD** in the speed search box and click the ID icon.

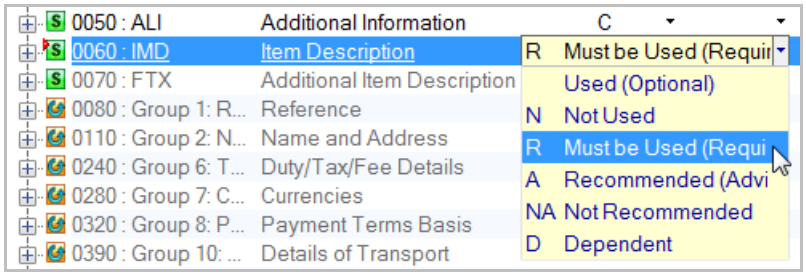
2. Click on the arrow for the **Req** (Requirement Designator) field.

   All requirements (M, O, F, and C) are listed. EDIFACT only uses M (Mandatory) and C (Conditional).

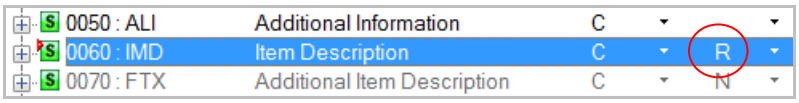


   "C" is the requirement for the IMD segment according to the published EDIFACT standard.

3. In the U/A column, drop the arrow and select **Must be Used (Required)**.



4. You should now see the **User Attribute** indicator has been changed to **R** (Required) for the IMD segment



5. **Save** your work.

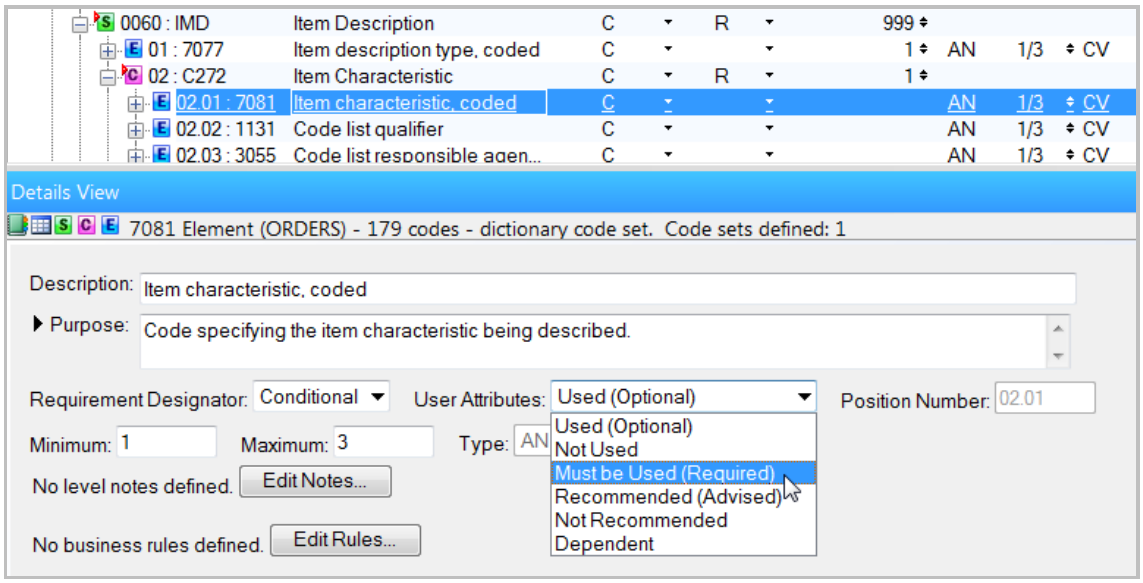
STOP

Marking Elements as Unused and Must be Used

START

1. With the IMD segment selected, click on the **+** symbol to expand the elements. Click on element **7081 Item characteristic, coded** at **IMD02**. (IMD02 = second element (02) in IMD.)

2. Look at the lower pane Detail Tab, click the User Attributes drop down arrow, and select **Must Be Used (Required)**.



3. Notice that the User Attribute in the tree pane now reflects the **R** for requirement.

   Note: Using the bottom pane is another way to change the Properties of elements, segments, or groups.

4. Scroll down in the elements pane and click on element **7383** at **IMD04**.

5. Click the **Used/Unused** button  on the toolbar. This button toggles between used and unused for a selected item.

6. Save your work.

STOP

Comments

- Selecting and marking user attributes en masse works for elements or segments.
- A gray field in the edit dialog box contains read-only information from the dictionary, but it can be copied. A white field shows dictionary information too, but it can be changed.

Working with Composites

A composite is a group of simple data elements that are used together. A composite tag starts with either a C or an S in EDIFACT. A simple data element within a composite is called a “subelement.” Although UN/EDIFACT refers to composites as elements, in this guide we will call them composites.

START

1. In the Tree pane, look at the **C273** composite at **IMD03**.

   Notice that the position numbers of the subelements begin with 03. The subelements’ position numbers have two parts: two numbers representing the position of the composite they belong to, followed by two numbers identifying the position of the subelements.

2. Click on the **+** symbol to expand the **C273** composite.

   The C273 composite has a Req. Des. and User Attributes, but has no Type, Min, or Max. That is because the composite itself does not contain data; its subelements do.

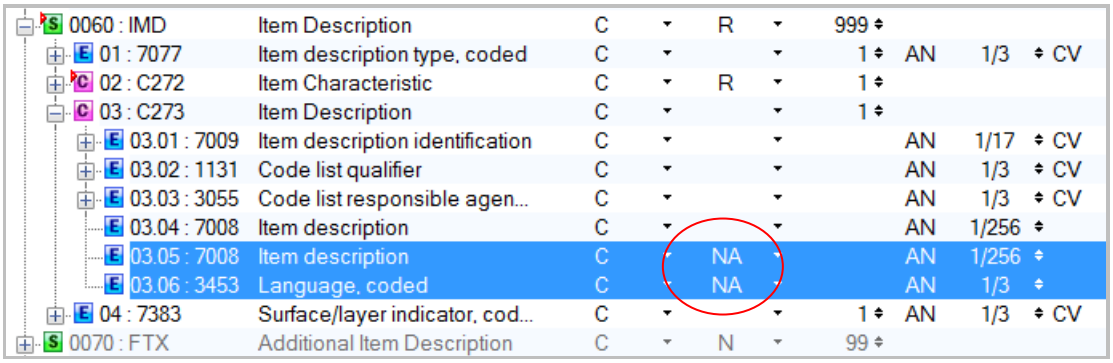
3. Click to select the subelements **7008** at **IMD03.05** and **Ctrl+Click** to select **3453** at **IMD03.06**.

4. Choose **Edit | User Attributes**.

   The Edit User Attributes dialog box appears.

5. Click **Not Recommended**, and click **OK**.

   NA appears in the U/A (User Attributes column). We will not be using them.



6. Save your work.

STOP

Comments

- In the EDI file, sub-element delimiters (:) differ from element delimiters (*). Example: MOA+5:10:DEM:3:7'

Changing Element Length

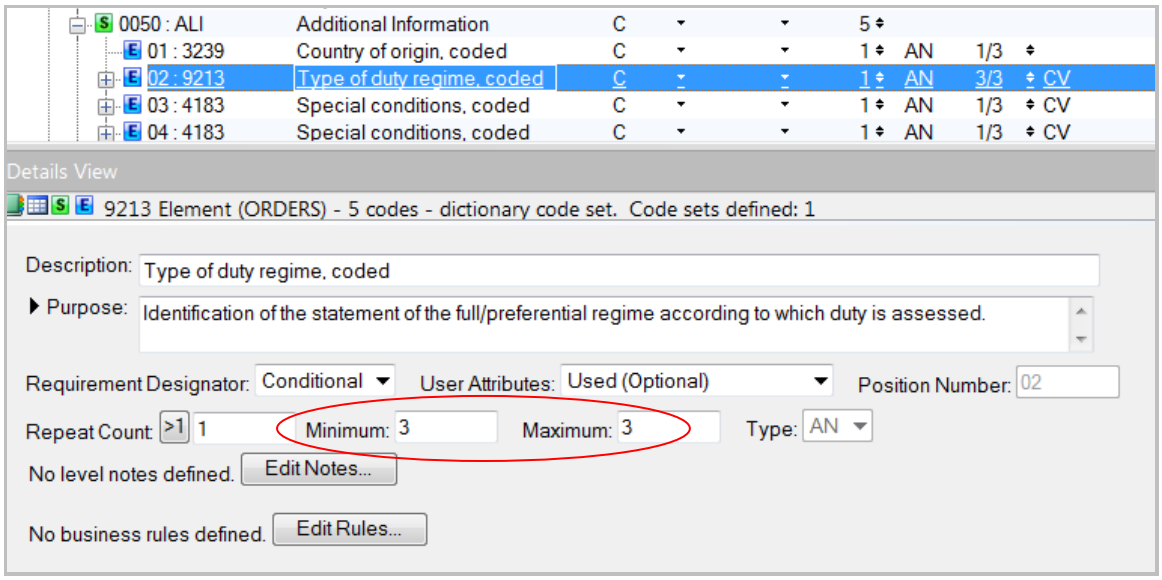
START

1. Click in the Tree pane and speed search for **ALI** at **0050**. Press ***Enter***.

2. Click on the **+** symbol to expand the segment, then click subelement **02**.**9213**.

3. Change the Min and the Max fields to **3**. We want the length of this subelement to require a number that is exactly 3 characters.

   IMPORTANT: To be compliant with the published standard, do not change the maximum length to a number higher than the standard maximum length.



4. Save your work.

STOP

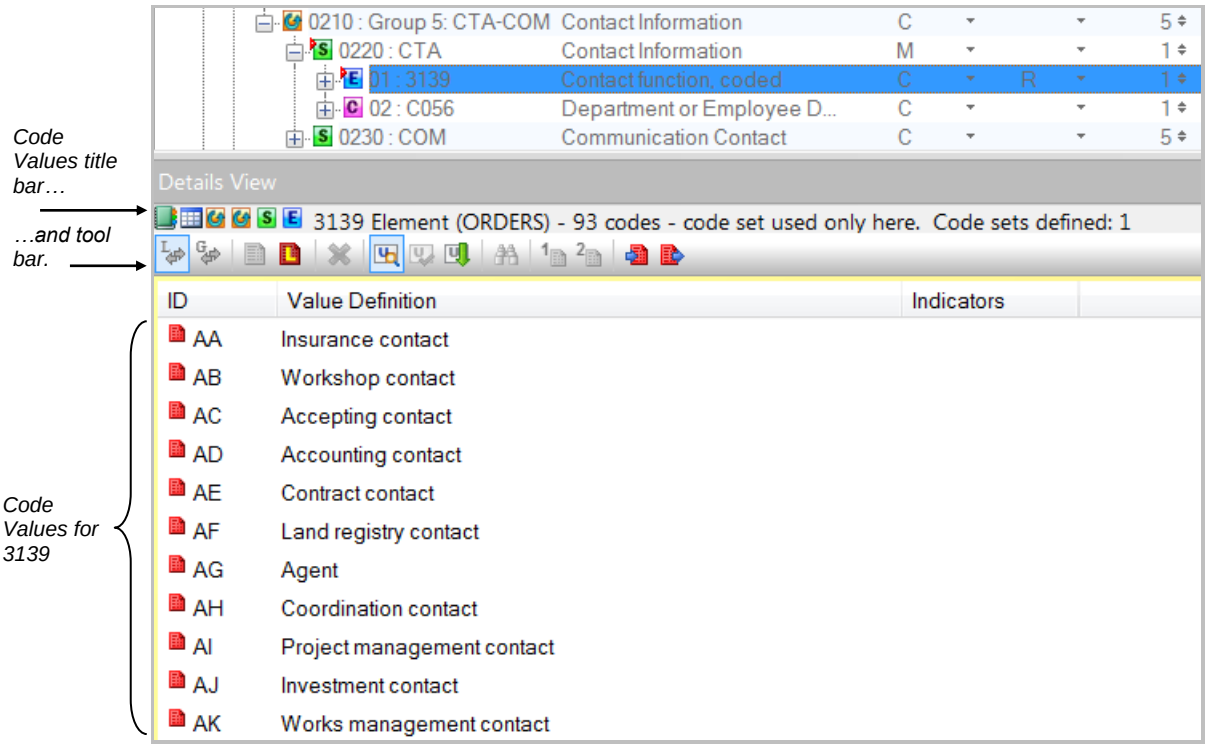
Editing Code Values and Finding Codes

We are going to customize the list of code values for element 3139 in the CTA segment. In addition, we are going to search for a specific word in the code values.

START

1. Click 0110 Group 2 and verify it is marked Used. Under that, click **Group 5**, position 0210. Expand the group, expand the **CTA** segment and click on element **3139** at **CTA01**. In the U/A (User Attributes) column, click **Must Be Used (Required)**, then press ***Enter***.

   Click on the Codes Tab at the bottom to review codes for element 3139.



2. Look at the Code Values title bar for code information.

3. Chose the “Mark all the code values not used” button on the Code Values toolbar below the title bar.

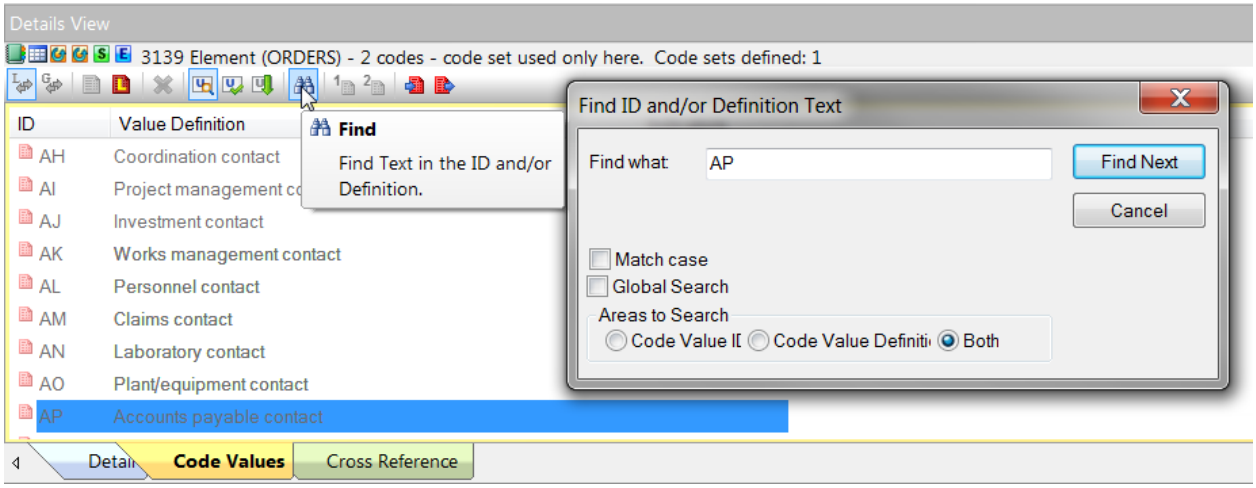
4. All of the dictionary code values become faded. If the codes disappear, select **Show/Hide** from the Code Values toolbar.

5. Let’s search for the code **AP**. Choose Find from the Code Values toolbar. The Find dialog box appears.

6. In the Find what field, type **AP**, and repeatedly press the **Find Next** button.

   Find searches for names or definitions in the current pane.

7. When you've found **AP** click ***Alt+M*** to mark it used.

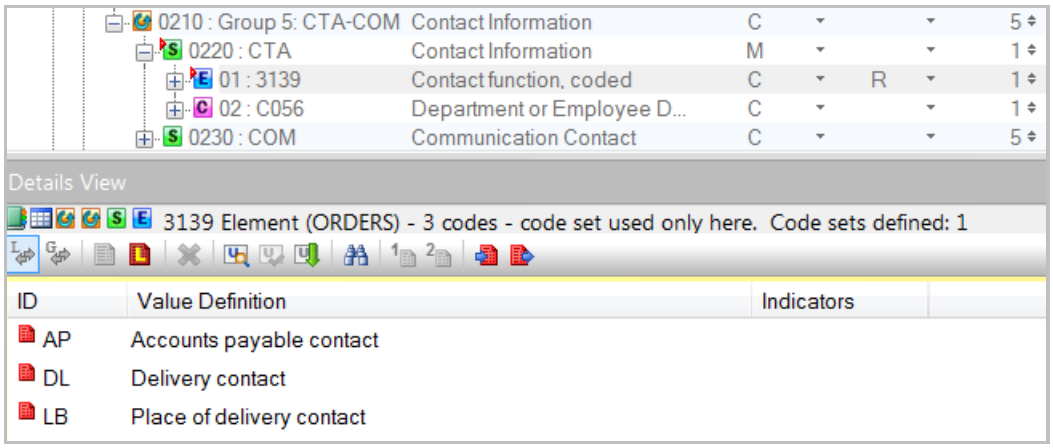


8. Speed search for code **DL,** then ***Alt+M*** to mark used, speed search for code **LB,** then ***Alt+M*** to mark used.

9. From the toolbar, select **Show/Hide** to view the 3 codes that are selected.

10. The Code Values pane title bar now shows that there are 3 codes in the code set. (You may have to click off element **3139** and then click on it again to refresh the title bar.) The AP, DL, and LB dictionary code values display in the Code Values pane.

Selected Code values for 3139 →



11. Now, you realize that you do not want to use the LB code value after all. Click on **LB** and press ***Alt+M*** to mark it not used.

12. Save your work.

STOP

Comments

- In the Code Values dialog box, you can select scattered dictionary codes by pressing *Ctrl*+click on the codes you want to use, then press ***Alt+M*** to mark them used or not used. Use speed search to find codes in a long list.
- When you change the list of codes, you are making a “code set” which will print in Doc Builder.

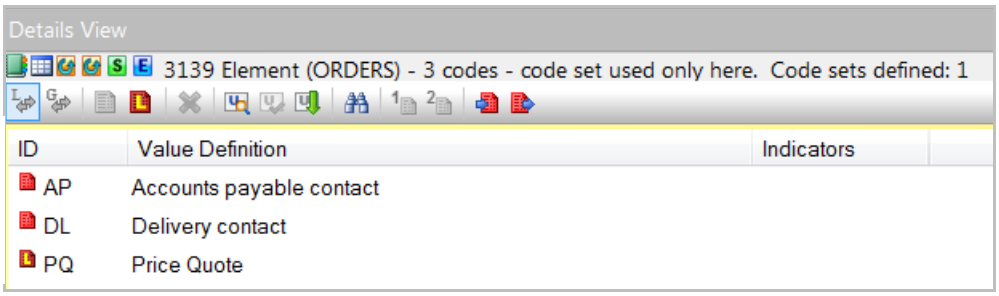
Adding a Local Code

Let's add a local (non-EDIFACT) code value to our 3139 element.

START

1. With the **3139** element at **CTA01.**
2. From toolbar in on the main menu, select **Show/Hide** to show all codes, both used and unused.
3. Choose the Local Code Icon on the toolbar below the Code Values pane title bar.
4. The **Local Code** field appears in the Code Values pane. Type **PQ**.
5. Press the ***Tab*** key and in the Definition field type Price Quote, hit **ENTER**.
6. Click **Show/Hide** to hide unused codes.
7. The PQ code value is added, and the Code Values title bar shows there are 3 codes in the Code Set. There is 1 code set defined.

The local code value displays with a local code icon.



8. Save your work.

STOP

Comments

- Normally, you try to conform to the published standard by using only dictionary code values. For elements without codes, you should put your local values into an application values list.
- Local codes can be added in the dictionary or in the message. Codes added to the element dictionary become dictionary codes and are not marked as local codes.

Adding Level 1, 2, and 3 Notes (or more…)

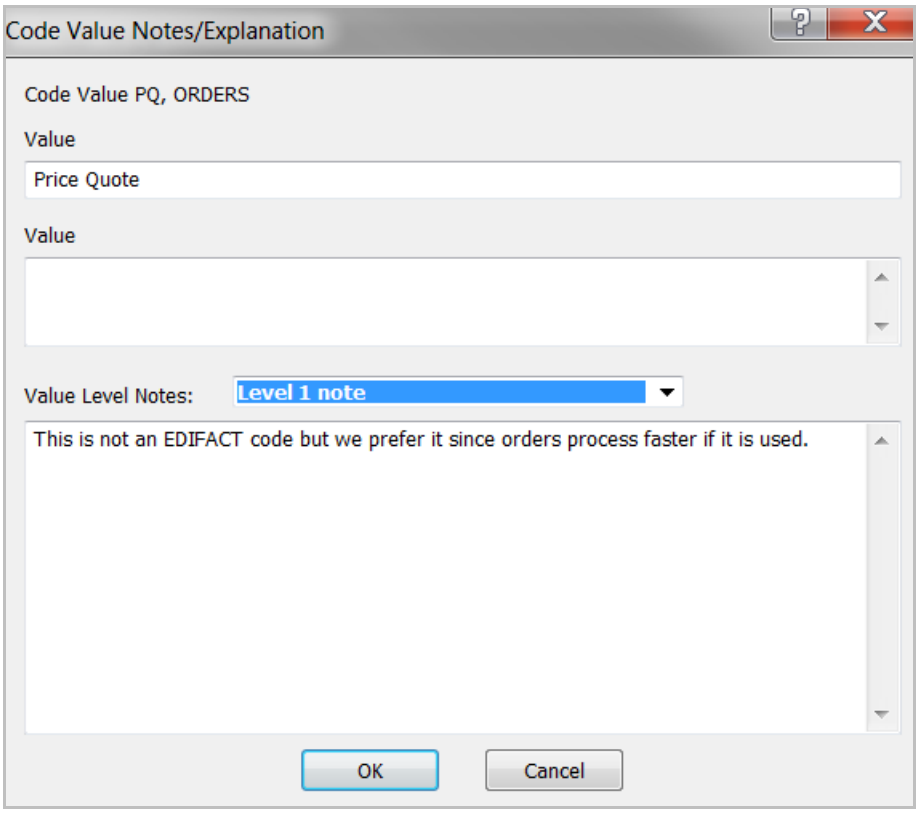
At the start of your MIG, you defined in the Properties how many level notes you want to be able to add. We chose three levels for this MIG. You can add a level 1, 2, or 3 note to any item (messages set, segment, composite, element, or code). You can add notes to EDIFACT groups if the MIG is based on D94A or higher.

START

1. Add a note to our **PQ Price quote** code value. In the Code Values pane, right click on PQ Price quote contact.

   From the menu, select **Level Notes/Properties**.

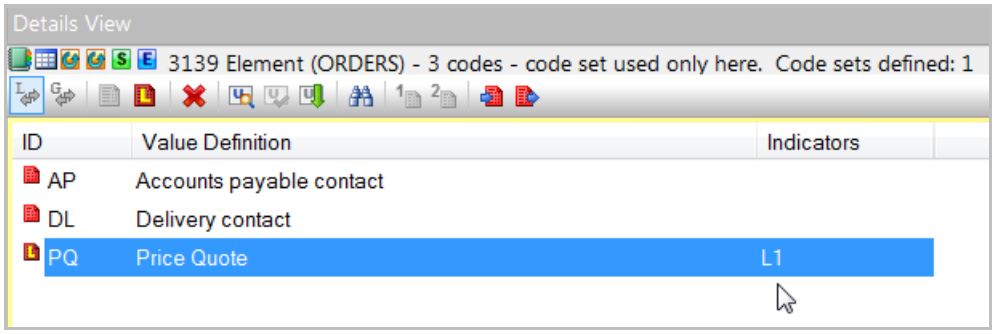
2. Add a Level 1 note to the new code value, type **This is not an EDIFACT code but we prefer it since orders process faster if it is used.**



Notes text goes here →

3. Click **OK**.

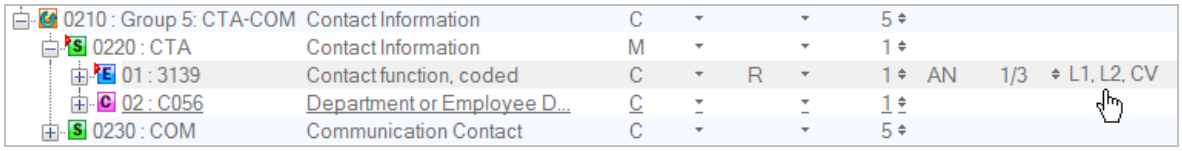
   Look at the Indicator in the Indicators column. It has a **L1** for a Level 1 note.



4. Select element **3139** at **CTA01** in the tree pane and right click, from the menu, select **Level Notes**….
5. Click in the Notes area and type **This element is conditional for EDIFACT but required by Kaver Corp.** Click OK.
6. On the same element, again right click and, from the menu, select **Level Notes**…. Drop the arrow in the Level Notes drop down box and select **Level 2 note**. We are starting a level 2 note, which will be for our own internal use.
7. Click in the Notes area and type **Have we updated the old purchasing system for these codes? Check with Systems Support.**

   The text will automatically wrap to the next line when typing.
8. Click **OK**.

   Look at the Indicator in the Indicators column. It has L1, L2 for Level 1 and 2 notes.



9. Select the **CTA** segment at **0220** and add a level 1 and 2 note. Then add a level 1 and 2 note to the **ORDERS Purchase Order Message**.
10. Save your work.

STOP

Comments

- The usage of notes should be consistent company-wide. For example:

  Level 1 notes - notes to partners

  Level 2 notes - internal notes

  Level 3 notes - additional notes or examples.

- Level notes can also be added from the Detail pane.

- You can print or suppress level 1, level 2, or level 3+ notes in Doc Builder.

- Notes show in Standards Reference.

Adding a Note from an Outside Source (Optional Exercise)

In this exercise, we are going to use level 3 notes for segment examples.

START

1. From Explorer, double-click on the **EDISIM** folder.
2. Click on the **Samples** folder, and double-click on **testpom.txt**.

   Let's copy text from this file into the Level 3 note of the CTA segment.
3. Choose **Search | Find** *or* **Edit | Find**, depending on your software.
4. In the Find what field type **CTA**, then click **Find Next**.

   CTA is highlighted.
5. Select the entire **CTA** line of text, and choose **Edit | Copy** or ***Ctrl*+*C***.
6. Choose **File | Exit** to close the textpom.txt file.
7. Choose **File | Close** to close Explorer.
8. Return to Standards Editor, and right-click on the **CTA** segment at position **0220**, select Level Notes.
9. Select the **Level 3** note from the drop-down box.
10. Click in the Notes area, type **Example:**, press the ***Spacebar***, and press ***Ctrl+V***.

    The CTA line is pasted as an example in our level 3 note.
11. Click **OK**.

    Notice that the Indicator column now has a L3 to indicate a level 3 note.
12. Save your work.

STOP

Comments

- You can copy and paste notes from other places or from another application, such as a word processor, into the MIG. If you copy text into notes, you will probably want to turn on **File | Preferences | Appearance | Wrap Soft Returns in Pasted Text**. This makes it easier to paste text from word processors into notes and other text fields. If this option is selected, new lines in pasted text become single spaces. Two or more consecutive CR/LF result in hard-returns.

Copying and Pasting Within a MIG

You can copy and paste segments and groups. Customizations you have done (user attributes, notes, code list changes, etc.) will copy also.

START

1. Make sure **View | Show Un-used** is on for the Tree pane. It must be on before pasting.
2. In the Tree pane, locate and click on **GROUP 2 NAD** at position **0110**.
3. Close the group if it is open.
4. Choose **Edit | Copy**.

   This will copy the entire group.
5. Leave the GROUP 2 NAD line selected, and choose **Edit | Paste**.

   We pasted a second GROUP 2 NAD right below the first one. It uses the same position numbers, since the groups are consecutive. Pastes always go BELOW whatever is selected when you perform the paste.
6. Turn off **View | Show Unused**.
7. Save your work.

STOP

Comments

- To MOVE segments or groups, use **Edit | Cut** rather than **Edit | Copy**.
- The position numbers of pasted items are reused to keep in sync with EDIFACT. Only groups after D93A have position numbers. To change position numbers, see the exercise “Resequencing Position Numbers” towards the end of the Standards Editor module.
- Business rules are not copied when copying and pasting segments or groups.
- Be careful not to nest a pasted group within another group, unless this is what you are trying to do.

Customizing the Group 2's

Now we want to customize each of the two GROUP 2's so that our printed MIG will clearly show that we have two different types of GROUP 2's. We will change the name of each NAD segment slightly so that we can easily tell them apart and set the requirements specifically for each group.

START

1. Click in the description field in the Tree pane for the first **NAD** segment at **0120**.

2. Change the name to **Name and Address - Payer**. Hit **ENTER**. Notice that the Detail pane reflects this change in the Description. You may have to click on the Detail Tab, in the bottom pane.

3. Click in the **Repeat** column, on the **GROUP 2 NAD,** change the repeat count to 1 and hit **ENTER**.

4. Click in the **User Attributes** column, and set the group to **Must be Used (Required)**.

   The entire GROUP 2 NAD (Payer) must be used.

5. We'll edit the first code value in each group trigger segment so that Analyzer can tell the groups apart. The trigger segment is the first segment in each group. Click on **+** symbol in front of the **Payer NAD** segment. Click element **3035**.

6. Choose Code Values tab at the bottom of the screen.

   Since this is the "Payer" NAD, we need to use only the PR code for element 3035. PR will act as the "group qualifier" and will enable Analyzer to tell the two groups apart.

7. Choose the "Mark all of the code values not used" button on the toolbar of the Code Values Pane to disable all the dictionary codes.

   Note: If all the code values disappear, choose **View | Show unused** from the main menu. All of the code values will display, but will be faded.

8. **Speed Search** in the Code Values pane by typing **PR**, right-click on the ST and select Mark used/Not used, or type ***Alt+M*** to activate the code value.
9. Look at the title bar of the Code Values pane. It shows that one code is used only here, meaning the list of codes here differs from the list of dictionary codes.

10. Scroll down to the next GROUP 2 NAD. Click the **+** symbol to open the NAD group, then again to open the **NAD** segment.

    Click in the Description field in the tree and change the name to **Name and Address - Ship To**, then press ***Enter***.

11. Click in the **Repeat** column on **GROUP 2 NAD**. Change the repeat to **1**, and the **U/A** (User Attributes) column to **Must Be Used (Required)**.

12. Add a Level 1 note explaining the requirement: **The Ship To NAD group is required for customs information.**

13. Click on the first element (**3035**), in the **Ship To NAD** segment, click in the Code Values pane and select the Mark all code values not used button on the toolbar. Then click **Show/Hide** to show unused codes.

14. Speed Search for the **ST** code, and ***Alt+M*** to mark used. Then click **Show/Hide** to hide unused codes.

15. The Code Values pane shows that only the ST code is used. This is the only valid code for this particular NAD's element 3035.

16. Click on **+** symbol in front of composite **C082**, click on subelement **1131 Code list qualifier** at **NAD02.02**.

17. In the Code Values dialog box, click **Mark all code values not used** button then click **Show/Hide** to show unused codes. Speed search code **58, *Alt+M*,** to mark used. Speed search on code **100, *Alt+M*** to mark used. Then click **Show/Hide** to hide unused codes.

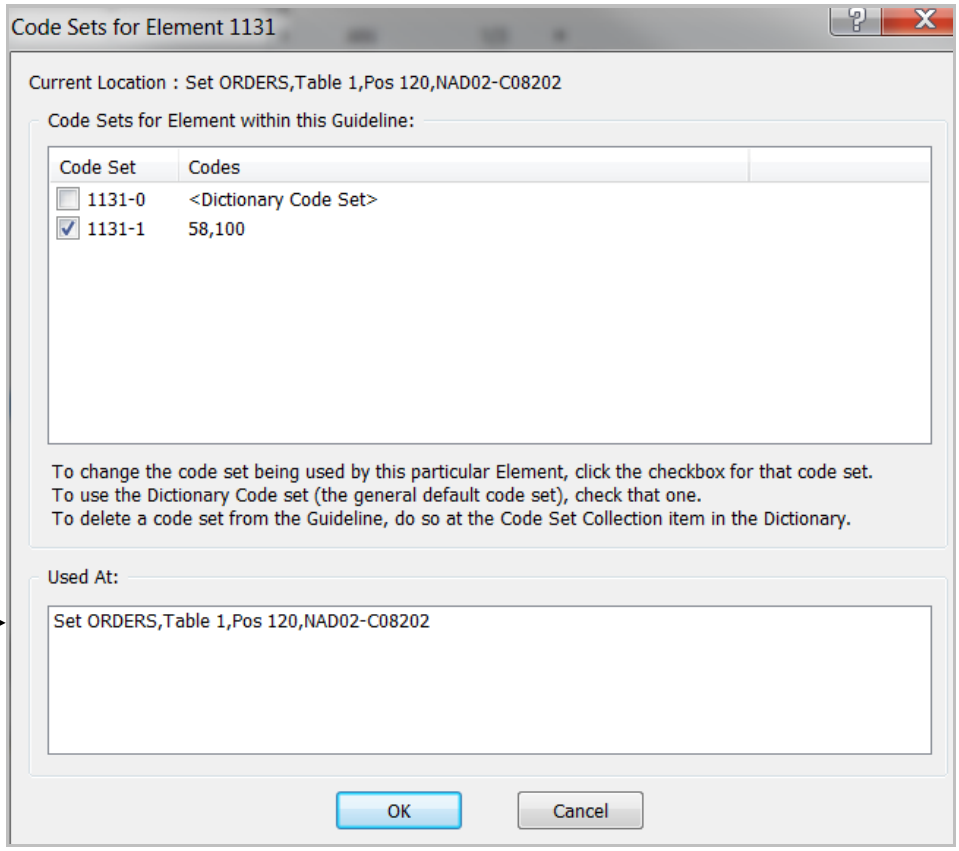
    The two used codes, 58 and 100, now display in the Values pane.

18. Scroll up in the Segments pane and click on the **Payer NAD** segment. Right-click on its subelement **1131,** select **Select Code Set** .

    The Code Sets for element 1131 dialog box appears.

19. Click on the **58,100** code set **1131-1** and look at where it is already used.

Code set selection used at →



20. Check the box 1131-1 to attach the code set to this location.

    You have re-used the codes as defined in the Ship To NAD.

21. In the Tree pane, click on **GROUP 3 RFF** under the Payer NAD group. Then ***Shift***+click on **GROUP 4 DOC**, and press ***Alt+M***.

    We have marked some segments as not used in the Payer group. The segments are faded.

22. Save your work.

STOP

Comments

- Analyzer can match up the NAD group in the data to the correct one in the MIG by looking at the first tag type code in the first segment of each NAD group. Additionally, Analyzer will look at other data elements and attempt to further match if the value in the first qualified element is the same.

- You can change a code set locally or globally. To make a global code change, you must first click on the “changes will be either local or global” icon on the Code Values toolbar. When yellow, changes are local. When red, changes are global.

Attaching Application Values and Regular Expressions

For elements without code values, you can require specific values, or a sequence of values called Regular Expressions. You can do this by attaching a value list to one or more elements in the MIG.

START

1. First, we will define a value list and attach it to a subelement. In the Tree pane, click on the **+** symbol to expand the **LOC** segment at position **0130** in the **Ship To NAD**.

2. Click on the **+** symbol to expand the **LOC02.01** composite, click on sub-element **3225** and choose **Edit | Advanced | Application Values**.

   The Application Data dialog box appears.

3. In the Value List field, type **RECVDOC**.

   This will be the name of our list.

4. ***Tab*** to the Description field, type **Receiving Dock**, and press ***Enter***.

   The cursor moves down into the Value field at the bottom.

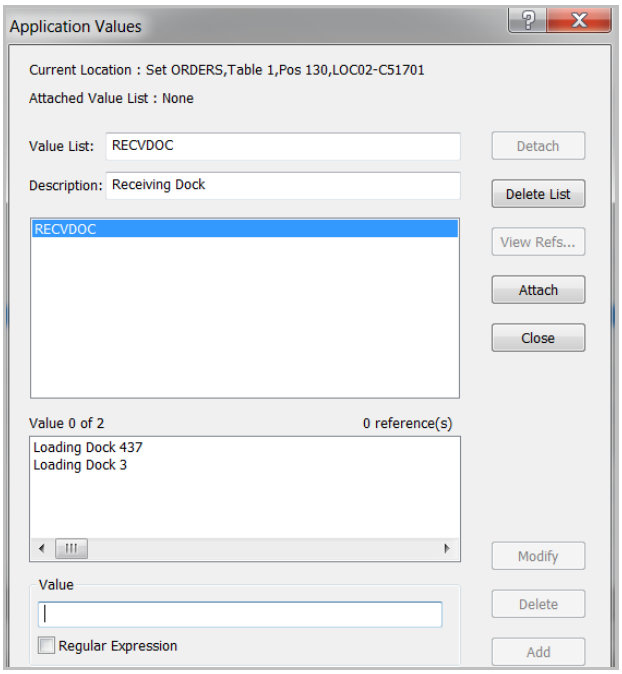
5. In the Value field, type **Loading Dock 437**, and press ***Enter***.

   The value is added to our RECVDOC value list and displays in the Value box. The cursor moves back to the Value field.

6. Type **Loading Dock 3**, and press ***Enter***.

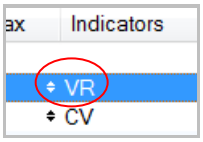
   This value is also added to our list and displays in the Value box.

Values added to the RECVDOC list →



7. Click **Attach** to close the dialog box and to attach our application values.

   The Indicator column for the LOC02.01 now displays the **VR**, Values List Reference.



8. Now, we are going to define another value list that contains a regular expression and attach it to a subelement. In the Tree pane, expand **Group 3 RFF**, expand the **RFF** segment at **0160** under the **Ship To NAD**.

9. Expand the **C506** composite, **RFF01**, click **1154** at **RFF01.02**, and choose **Edit | Advanced | Application Values**.

10. In the Value List field, type **REFID**.

    This is the name of our second list.

11. ***Tab*** to the Description field, type **Reference ID**, and press ***Enter***.

    The cursor moves to the Value field.

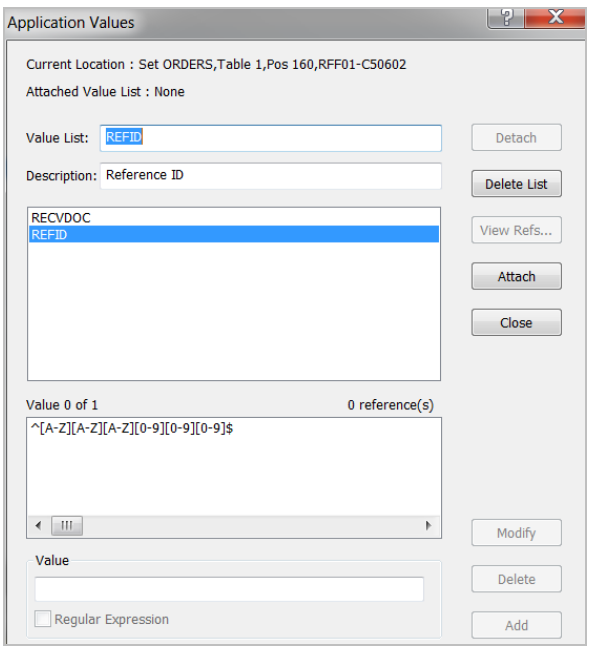
12. In the Value field, type **^[A-Z][A-Z][A-Z][0-9][0-9][0-9]$** (DO NOT PRESS *Enter*).

    This sequence requires a value that consists of 3 alpha characters followed by 3 numeric characters such as **ABC123**. The caret symbol (^) means nothing can appear before the regular expression. The dollar sign ($) means no characters can appear after it.

13. Click the **Regular Expression** check box, and click **Add**.

    The regular expression is added to our REFID value list and displays in the Value box as shown in the following example.

Regular expression added to the REFID list →



14. Click **Attach**.

    Again, the Indicator for the Application Value List (using a Regular Expression is VR, Value List Reference.

15. Add a level 1 note for your trading partners that explains the regular expression. Right-click on the **1154** element, select **Level Notes**. In the notes area type, **The required value for this element starts with ABC followed by 2 alpha characters and 3 numeric characters. Example: ABCAB123**.

16. Click **OK**.

17. Save your work.

STOP

Comments

- Test Data Generator does not yet support Regular Expressions.
- Acceptable values for a regular expression are required by Analyzer. For example, the value ABC123 is acceptable for our MIG, but Analyzer will reject 123ABC or any value that differs from our value sequence.
- For more information about regular expressions, see **TIB_fsp_edisim_<*n.n*>_fseditor.pdf**, the section Customizing Your Guidelines, Regular Expressions.

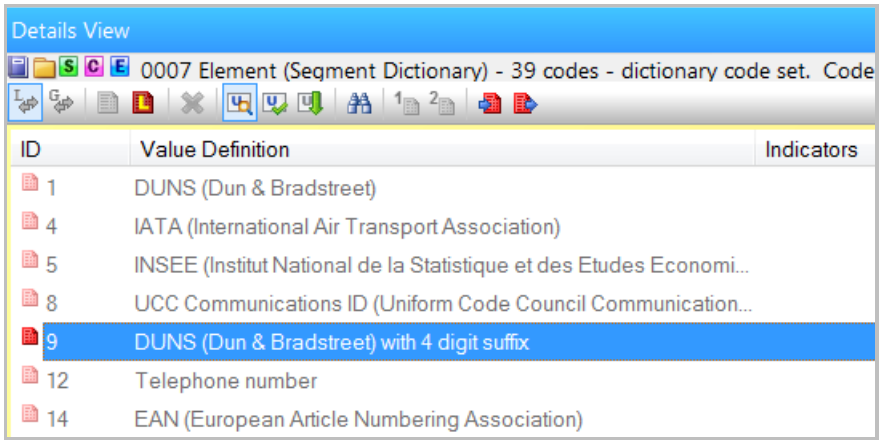
Using the Dictionary

The dictionary is a list of all segments, composites, elements, and codes in the underlying standard.

START

1. Click on the **+** symbol before the **Dictionary Objects** in the Tree pane.
2. Click on the **+** symbol before the **Segments Dictionary**.
3. Search for the **UNB** segment, and expand the segment to view the elements.

   The Dictionary Segments can be edited, including business rules, notes, comments, and dependency notes. These changes carry through to the message.
4. Expand the **UNB02.02** composite. Click on subelement **0007 Partner identification code qualifier**, select the Code Values tab at the bottom.
5. Click on the “Mark all codes not used” button. Click on the Show/Hide button.
6. Speed search on code **9**, **DUNS with 4 digit suffix**, **Alt+M** to mark as used.



7. Save your work.
8. Scroll all the way up in the Segments dictionary and click on the **–** symbol to close. Click on the **+**symbol to expand the **Composites** dictionary.

   All composites in the dictionary appear.
9. Click on a **plus sign** in front of a composite to expand the sub-elements.
10. Click on the **minus sign** to close the sub-elements.
11. Scroll back to the top and close the **Composite** dictionary. Expand the **Elements** dictionary.

    A list of all elements in the dictionary appears.

12. Click on the first element (**0001**).

    Caution: You cannot create a code set in the element dictionary.

13. Let's search for elements containing the word "date." Choose **Edit | Find Descriptive Text**.

    The Find dialog box appears.

14. Type **Date**, and repeatedly press **Find Next**.

    Find will search the names or descriptions in the Tree pane.

15. Click **Cancel** to close the Find dialog box.

16. Click on an element, and look at the choices in the Tree pane and Detail pane.

    You could change things here and these changes would carry through to this element wherever it is used.

17. Click **Cancel**.
18. Click on **–** symbol to close the **Element** dictionary.

STOP

Comments

- Enveloping segments for Analyzer can be customized in the dictionary.
- Changes in the dictionary carry through to the message.

Creating a Business Rule to Set Usage

You can create a business rule to reflect a particular requirement for your MIG. These are “if-then” conditions that you designate. Let’s create our first rule: If the code value for sub-element 1001 at BGM01.01 = 71 then the PAI is required.

START

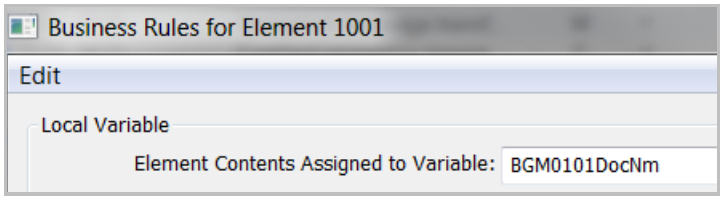
First, we’ll define the variable on the SBR02.

1. Expand the segment BGM at 0020.

2. Expand the composite C002, right click on element **1001 Document/message name, coded** at **BGM01.01**, and choose **Business Rules**.

3. Click in the **Local Variable** text box and type the variable name **BGM0101DocNm**.

   It’s a good idea to use the segment and the element position as part of the name so that it will be easier to locate later. You can enter up to 16 alphanumeric characters.



4. Click **OK** at the bottom of the dialog box.

   Notice the **BV** (Business Variable) in the Indicators column.

Now, create the business rule on the item that is affected by the BGM1.01.

1. Open the segment **PAI** at **1160**. Right-click on the **PAI** *segment* and choose **Business Rules**.

2. Click **New** in the Business Rules dialog box to set up the new rule.

3. In the When to Run the Rule area of the dialog, click the **Conditionally** radio button to indicate this rule will run when certain conditions are met.

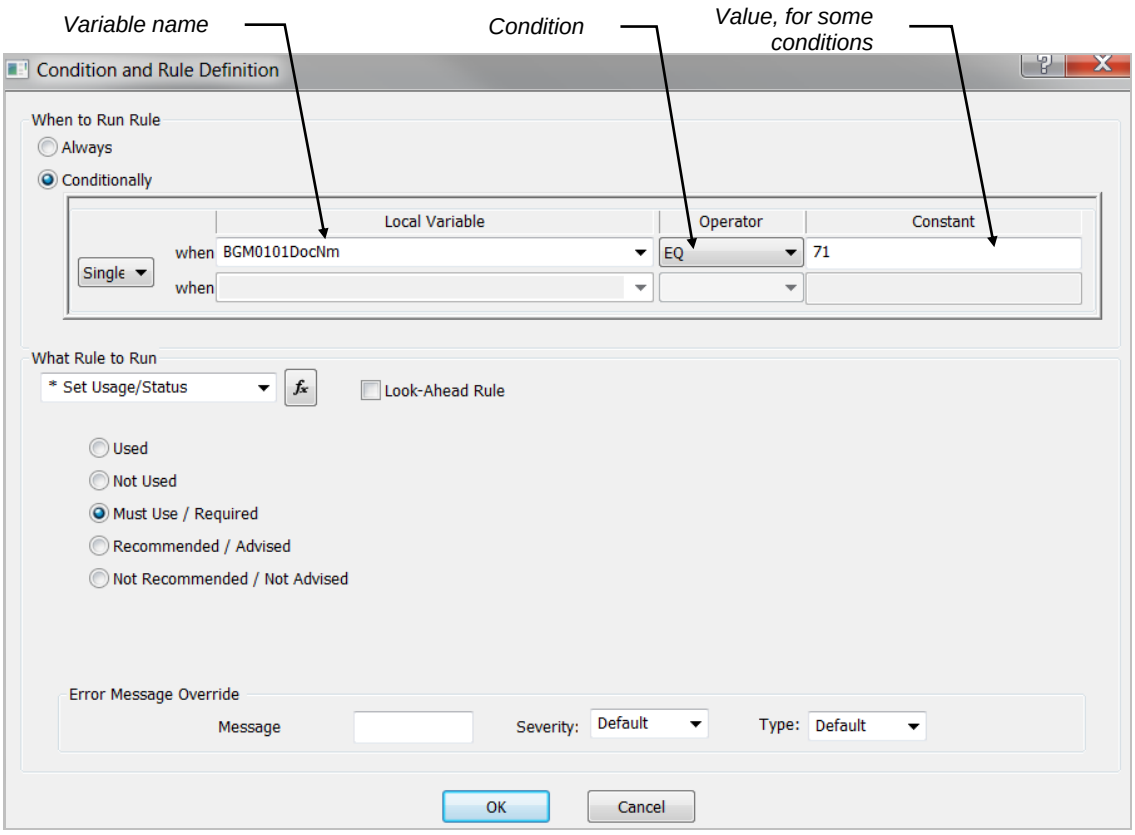
4. Select **Single** from the drop box.

5. Click the drop box for the **Local Variable** field and select the variable name **BGM0101DocNm** that we previously created.

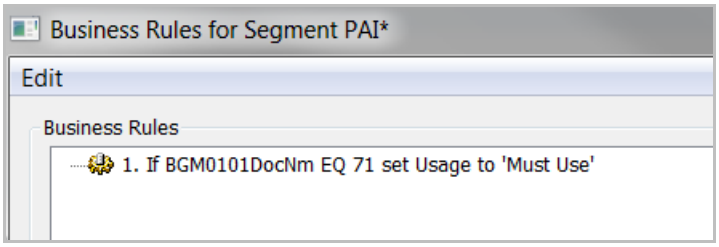
6. Click the drop box in the **Operator** field and select **EQ** (equals), then type **71** in the third field.

7. In the What Rule to Run area of the dialog, select **Set Usage/Status** from the drop box.

8. Select the **Must Use / Required** radio button.



9. Click **OK** and check the rule:



This indicates that, if a value of 71 appears in the data for BGM1.01, then this segment must be used.

10. Click **New** and create a second rule indicating that, if the BGM1.01 doesn't equal 71, then the segment must NOT be used.

11. Click **OK**.

    Notice that the Indicator Column now has a BZ to indicate a Business Rule has been attached.

12. Save your work.

STOP

Comment

- A variable must be located above the item to which the rule applies. In this exercise, notice that the BGM segment is located above the PAI segment.

Creating a Business Rule to Use Application Values

You can create an application values list, then set up a business rule that uses the values list based on the conditions that you choose.

Let's create another business rule: If the code value for element 3227 at LOC01 = 7 then use the Application Value List RECVDOC at LOC02.01, sub-element 3225.

START

1. In the Tree pane, click on **LOC** at **0270, Group 10 TAX-MOA-LOC**.
2. Expand the segment **LOC**, click on element **3227** at **LOC01**, and choose **Edit | Advanced | Business Rules**.

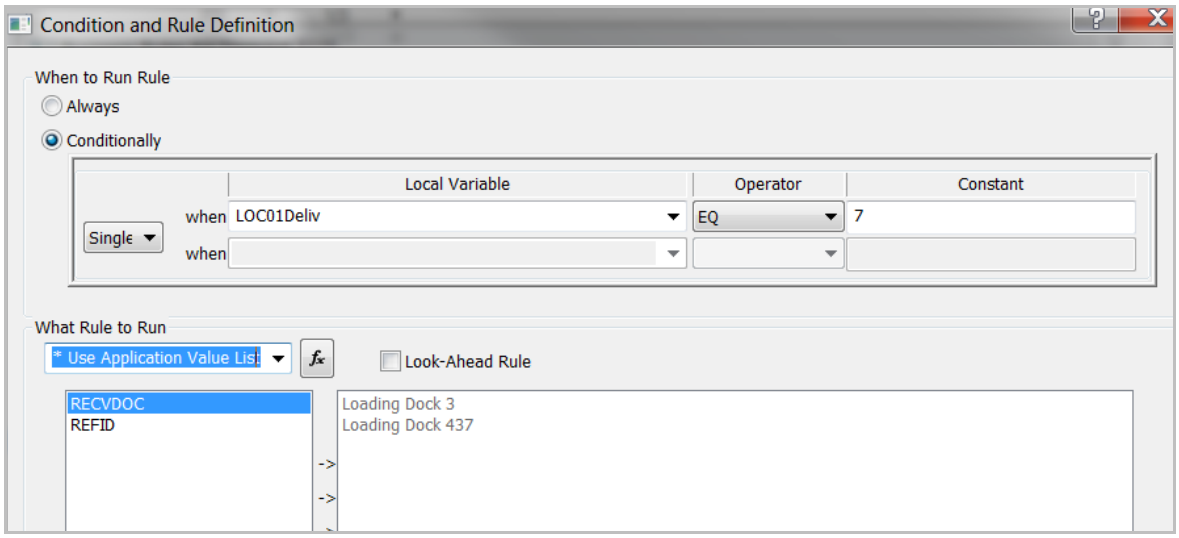
   The Business Rules dialog box appears.
3. Click the **Variable** button.

   The Define Variable dialog box appears.
4. Type **LOC01Deliv** as the variable name.
5. Click **OK** in the Define Variable dialog box, and again in the Business Rules dialog box.
6. Expand composite **C517**, click on sub-element **3225** at **LOC02.01**, and choose **Edit | Advanced | Business Rules**.

   The Business Rules dialog box appears.
7. Click the **New** button to set up the rule.

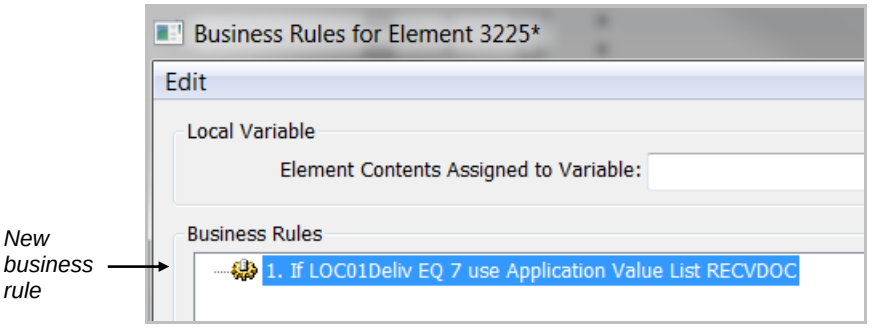
   The Condition and Result Definition dialog box appears.
8. Click the **Single** Relationship button.
9. Select the variable **LOC01Deliv** in the first field, select **EQ** in the second field, and type **7** in the third field.
10. Click the **Use Application Value List** button under What Rule to Run dropdown.

    This will allow us to use our application value list. Your selections should look like the following example.



11. Click on **RECVDOC** in the Application Values Lists field, and click **OK**.

    The new rule is added.



12. Click **OK** to close the Business Rules dialog box.

    Analyzer will enforce this condition.

STOP

Creating a Business Rule to Verify Date Formats

You can create a rule to verify the date and time format. You can set a business rule in the dictionary (recommended) or set it on a certain segment or element in the message.

START

1. To have Analyzer verify that the Date and Time values follow the format chosen, click on **the +** symbol for **Dictionary Objects,** then expand the **Element** dictionary.

   If you set the Date and Time checking function through the dictionary, it will be validated every time it encounters a Date and Time element.

2. Speed search for element **2380**, and press ***Enter***.

3. Choose **Edit | Advanced | Business Rules**.

   The Business Rules dialog box appears.

4. Click the **New** button.

   The Condition and Result Definition dialog box appears.

5. Select the **Always** Relationship button, if not already selected.

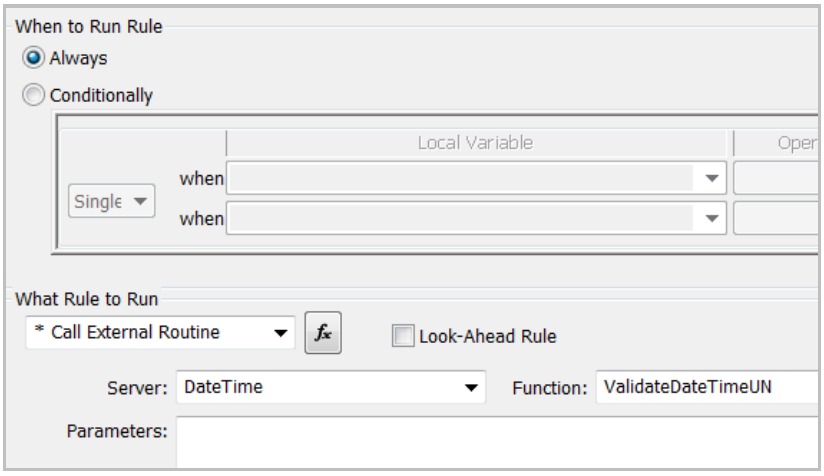
6. Select the ***Call External Routine** button under What Rule to Run.

7. Select **DateTime** as the Server Name under External Routine.

8. Select **ValidateDateTimeUN** as the Function Name.

   This will invoke the DateTime check every time Analyzer finds element 2380 in the message. Analyzer will make sure that the date matches the format specified by its qualifier, element 2379.

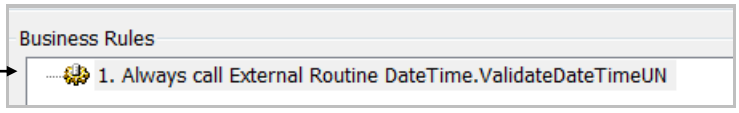
   Leave the Parameters field blank. Your selections should look like the following example.



9. Click **OK**.

   The new rule is added.

New business rule



10. Click **OK** to close the Business Rules dialog box.

11. In the Tree pane close up the **Dictionary Objects**, click the **ORDERS Purchase Order Message**.

    You have exited the Dictionary and are back in your message.

12. Find the segment **DTM** at **0030** and expand it.

13. Right click on element **2379** at **DTM01.03**, click on the **Code Values** tab.

    Click on the "**Mark all codes not used**" button.

    All code values are now faded.

14. Type **102, *Alt+M*** to mark used**.**

    The Analyzer will verify that dates in the DTM at position 0030 are in CCYYMMDD format.

15. Save your work.

STOP

Comments

- You can set up the ValidateDateTimeUN function on individual element 2380s in the message if you don't want to invoke the Date and Time check for all element 2380 occurrences.
- The DTM segment is used in this exercise; however, you can invoke the DateTime check for any segment that has 2379 and 2380 elements.
- Business rules also print in Doc Builder and Standards Reference.
- Test Data Generator does not make default data that follows business rules.

Using Spell Check

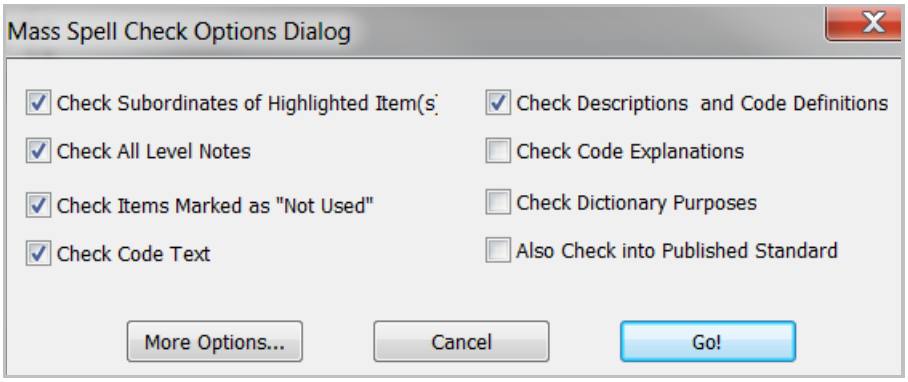
You can check the spelling of an entire document or items in a document. In this exercise we'll check the spelling in a group and a dialog box.

START

1. In the Tree pane, select **GROUP 5** and choose **Edit | Advanced | Spell Check**.

   The Mass Spell Check Options dialog box appears.

2. Clear all of the check boxes *except* for **Check Subordinates of highlighted item(s)**, **Check All Level Notes**, and **Check Descriptions and Code Definitions**.

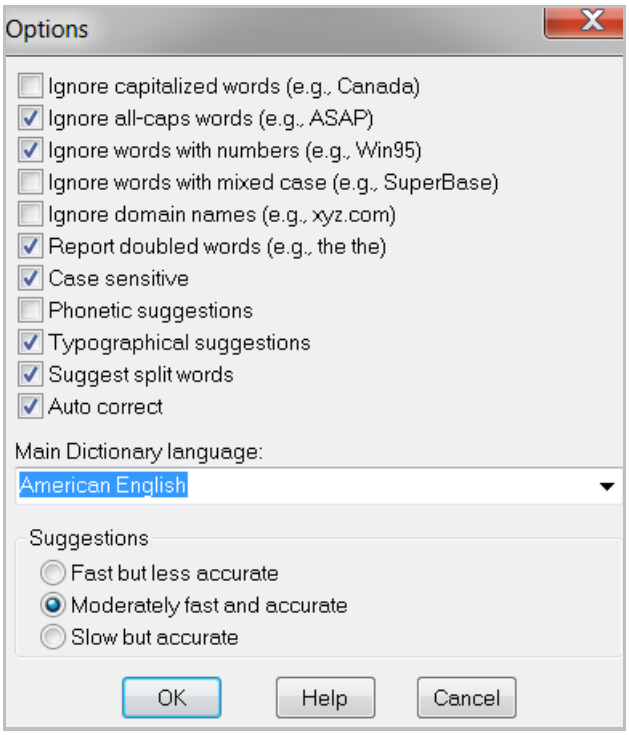


3. Click **More Options**, and review the available options.

   Note: Any selections you would make in the Options and the Mass Spell Check dialog boxes remain in effect for ALL MIGs or messages until you change them.

4. Select **Ignore all-caps words (e.g., ASAP)** and **Ignore words with numbers (e.g. Win95)**.

   Your selections should look like the following example.



Click **OK**.

5. Click **Go!**

   The spell checking function activates.

   a) If no misspelled words are found, the “Spell Check Complete” appears.

   b) If a word is found that is not in the spell check dictionary, the Mass Spell Check Progress window appears. It displays the location of the misspelled word and highlights it within the text. The Check Spelling dialog box also appears. Select the correct word and click the appropriate button for the detected word. When finished, “Spell Check Complete” appears.

6. Click **OK**.

STOP

Comment

- You can spell check using any language listed under Main Dictionary language.

Exporting a MIG

There are several reasons why you would want to export your MIG:

- Share it with others
- Create a backup copy
- Create a Standard Exchange Format (SEF) file to populate a SEF-ready translator

START

1. Save your work.

   You must save before you can export. Otherwise, Export Current Guideline is faded on the menu.

2. Choose **File | Export | Export Current Guideline**.

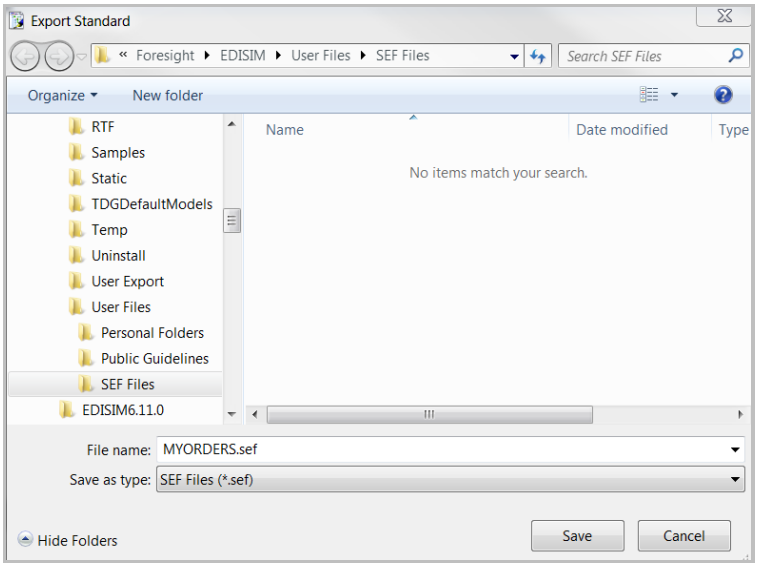
   Notice there are various formats you can export to (e.g., igML, Schema). For our exercise, we will choose “To SEF…”

   Note: If you have multiple guidelines or MIGs, you may export several at a time by choosing Export Multiple Guidelines. A mass export can take a long time, so be prepared.

   The Export Standard dialog box appears. The default folder is the name of the computer’s local drive.

3. Navigate to the **User Files** folder.
4. Create a new folder and type **SEF Files** for the folder name.
5. Press ***Enter*** twice so that ‘SEF Files’ appears in the Save in field.

   Notice that the File name field is displaying MYORDERS.SEF. We are creating a SEF file. This file is going to serve as a backup.



6. Click **Save**.

7. Choose **File | Close**.

STOP

Comments

- SEF is being used to exchange guidelines or MIGs and to automatically load translators. Be sure to zip it first if sending it electronically, since it contains some long lines, which can be broken during email transmissions.

- Use **File | Import** to import a SEF file or a DocStarter file.

Exiting Standards Editor

START

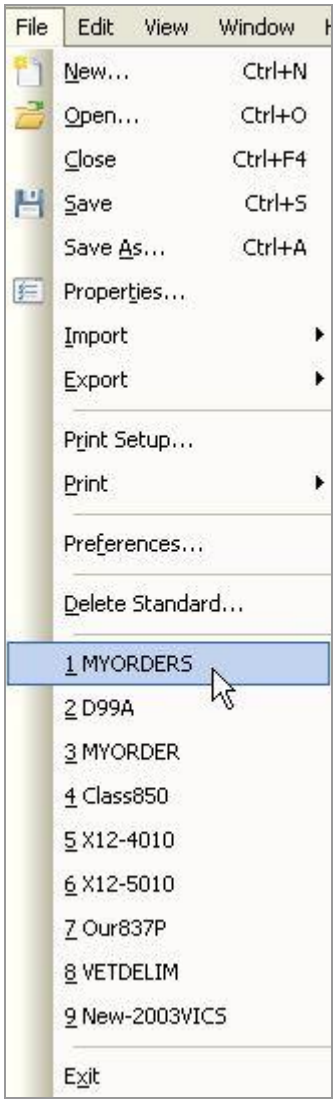
1. Choose **File | Exit**.

STOP

Re-Opening a MIG

START

1. Enter Standards Editor again.
2. Choose **File** and look at the menu.



3. Choose **MYORDERS** from the menu, or choose **Open | Both Published & User | MYORDERS**.

   Note: Do not choose **File | New** on a MIG you've already created unless you want to create a separate MIG and can reuse much of the work.

STOP

Resequencing Position Numbers (Optional Exercise)

Since we pasted our GROUP 2 NAD below the original GROUP 2 NAD, EDISIM assumes that we are defining an alternate variation of the original group. Therefore, it re-uses the same position numbers for each group. This keeps them aligned with the position numbers in the originating EDIFACT standard.

However, there may be times when you want to change position numbers of segments or groups of segments. You would normally resequence only if you are part of a standards committee that is building new messages or models.

START

1. Expand the **ORDERS Purchase Order Message**.

2. Expand the **Group 2 NAD**, click on the **Ship To NAD** segment.

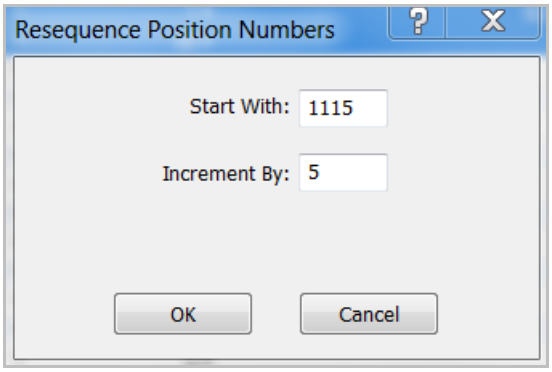
   Look at the Pos. No. field in the Detail pane at the bottom. This is where you would type in a new position number. Do not actually change the position number.

3. In Table 2, select **MOA** at **1100** through **FTX** at **1170**, and choose **Edit | Advanced | Resequence**.

   The Resequence Position Numbers dialog box appears.

   You can type a starting number for the first segment in the selected range and an increment number, which will determine the position numbers of the other segments in the selection.

4. Change the Start With field to **1115**, and change the Increment By field to **5**.



5. Click ***OK***.

   Note the new numbering sequence.

STOP

Cutting and Pasting (Optional Exercise)

Cut and Paste is similar to Copy and Paste. You must have **View | Show Unused** turned on for the Segments pane before you can paste. You'll use Cut and Paste mostly to correct mistakes.

Let's assume that we want our Payer NAD to appear *after* the Ship To NAD in our MIGs document. We can use cut and paste for this.

START

1. Select **View | Show Unused** and verify that it is turned on. If not, click it to turn it on.
2. Select the **GROUP 2 NAD** line just above the Payer NAD segment. Be sure that you have the Group 2 NAD line, not a segment.

   If you cut a Group line, the entire group will be cut.
3. Choose **Edit | Cut**.

   EDISIM suggests that you might want to mark them as unused instead of cutting.
4. Click **Yes**.

   The group disappears.
5. Click on the **Group 2 NAD**, then choose **Edit | Paste**.

   The Payer NAD is pasted at that location, just after the Ship To NAD and is now the **Group 6 NAD**.

IMPORTANT: DO NOT SAVE this exercise. We want to leave our groups in the same order.

STOP

Adding and Deleting Items (Optional Exercise)

Just for practice, let's add and then delete a group.

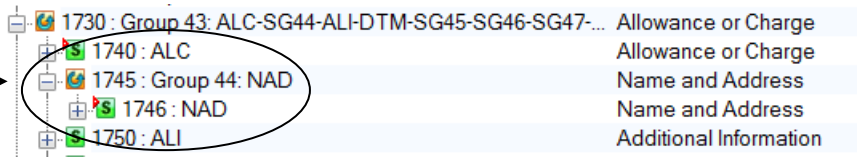
START

1. Search and select the **Group 43 ALC-ALI-DTM…** segment at **1740**.
2. Choose **Edit | Add Item | Group**.

   A drop down list appears with the full segment dictionary.
3. Type N to go to the beginning of the N's, select **NAD**, hit **ENTER**.

   We have added a GROUP 44 NAD, a NAD segment.

Here is our new Group 44 NAD



4. Now we need to insert another segment into the group. Click on the NAD segment at position 1746, and choose **Edit | Add Item | Segment**.
5. Type an L to go to the beginning of the L segments. Select **LAN,** hit **ENTER**.

   We've now added a second segment to our newly added NAD group. This is an alternative to copying.
6. We don't really want this group, so let's delete it. Select **GROUP 44 LAN-NAD** and press the **Delete** key (or choose **Edit | Delete**).
7. When asked to confirm, click **Yes**.

   The entire group is deleted.

STOP

Comments

- To Insert a table, you must be on a closed table or on the Message (ORDERS) first, then use **Edit | Add Items | Tables** and then select its first segment. Delete a table by deleting or moving all of its segments.
- When inserting a group or table, you always have to choose its first segment at the same time.

Exiting without Saving

START

1. Choose **File | Exit**.
2. DO NOT SAVE. Click **No** when asked if you want to save.

STOP

Backing Up Files

What type of backup should you choose?

There are two different types of backups. Consider using both.

- Copy all files in the \EDISIM\User Files\Public folder.

  You can zip them to save space.

- Export an individual MIG or multiple MIGs using **File | Export** from within Standards Editor.

How often should you back up?

That depends on how much work you are willing to lose in case of equipment problems, an accidental deletion, etc. Get in the habit of backing up frequently. You can write a small batch file to do this for you, then give it an icon, if that would encourage frequent backups.

Where should you store backups?

Store them off your machine to protect your data if your hard drive fails. Do not overwrite your last backup when creating a new one, and don't entrust all backups to the same diskette or tape.

Doc Builder

Purpose

The purpose of this exercise is to generate a MIG document from the standard that we customized in the last exercise. You can print this MIG document from Doc Builder, export it as an RTF (Rich Text Format) file for input into word processors, or create an HTML file to post on a Web page.

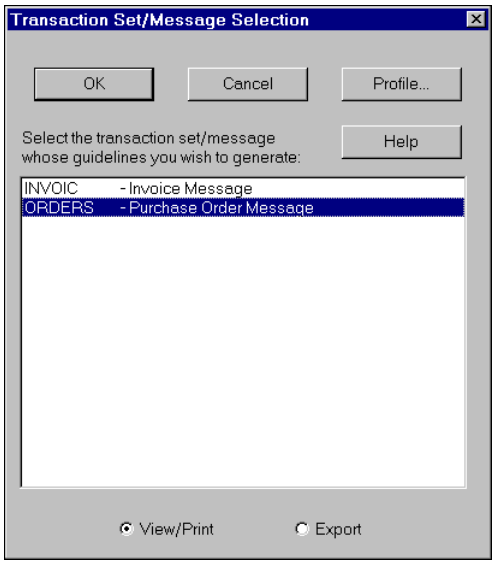
Selecting a MIG, Message, and Profile

START

1. Choose **Start | Programs | <*TIBCO_HOME*> | EDISIM | Document Builder**.
2. Maximize your screen, if necessary.
3. Choose **File | Open Standard**.

   The Open Standard dialog box appears. Only MIGs (not published standards) are available in Doc Builder.
4. Select **MYORDERS**, the one that you just created in Standards Editor.
5. Click **OK**.

   The Transaction Set/Message Selection dialog box appears.
6. Click on **ORDERS Purchase Order Message**.



7. Click the **Profile** button.

   The Doc Builder Options dialog box appears. Leave it open for the next exercise.

STOP

Comments

- Formatting Options:
    1) Default Page Header (none)
        a. Page Header = line at top of page
- Default footer:
    a. Left: name of MIG and base standard
    b. Center: page number (not changeable–no field is available)
    c. Right: today's date
- You can save a profile (Save button) and reuse a saved one (Load button).

Changing Doc Builder Options

We are going to change some of the Doc Builder options, so that our generated document is customized the way we want it.

START

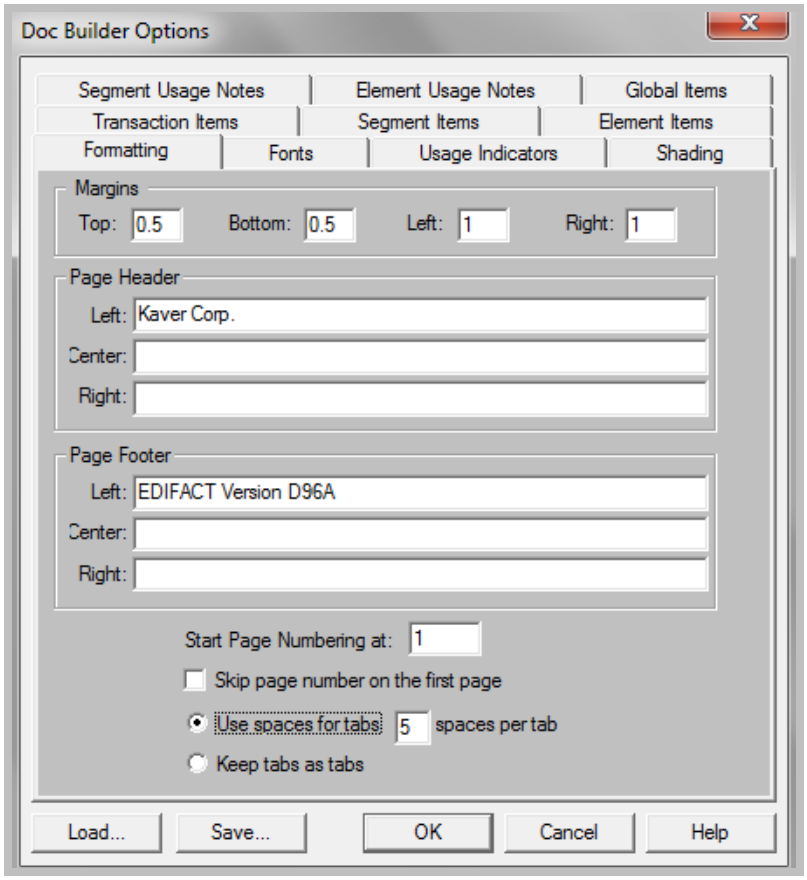
1. In the Doc Builder Options dialog box on the **Formatting** tab, click in the Page Header field and type **Kaver Corp**.

   Kaver Corp. will appear at the top of each page of your generated document.

   Note: All items under Formatting will be viewable in Print Preview once you generate the document.

2. In the Left Footer field, type **EDIFACT Version D99A**.

   The version text will appear at the bottom left of each page.

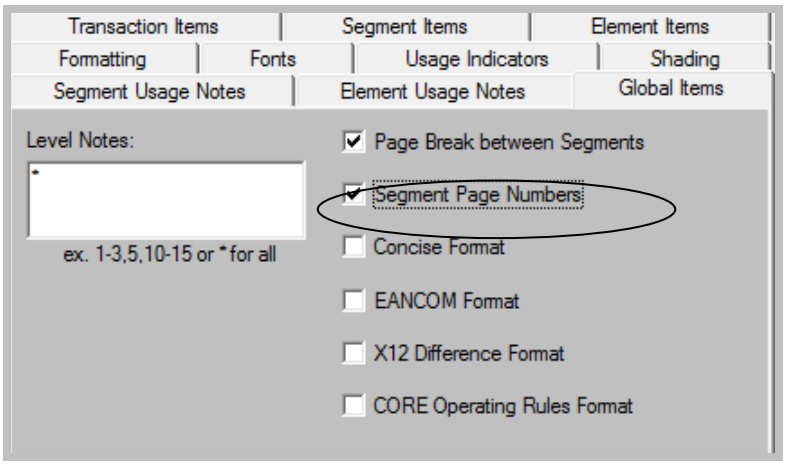


3. Click the **Global Items** tab.

   Notice that some of the check boxes are selected. All selected items in the Doc Builder Options dialog box will be included on your generated document.

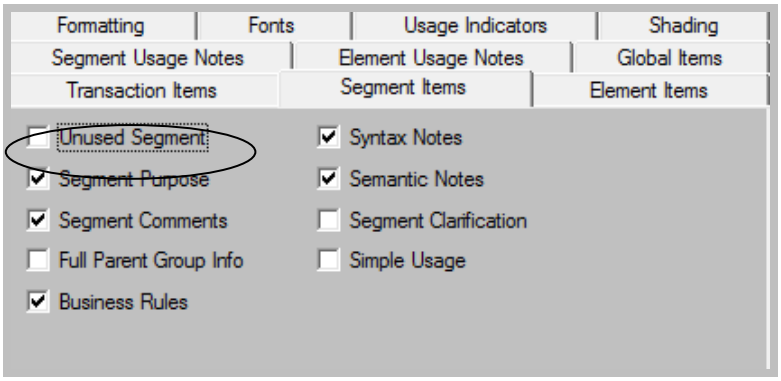
4. Select the **Segment Page Numbers** check box.

   This will create a column of page numbers on the left side of the Segments Table in the document. The Segments Table serves as a table of contents.



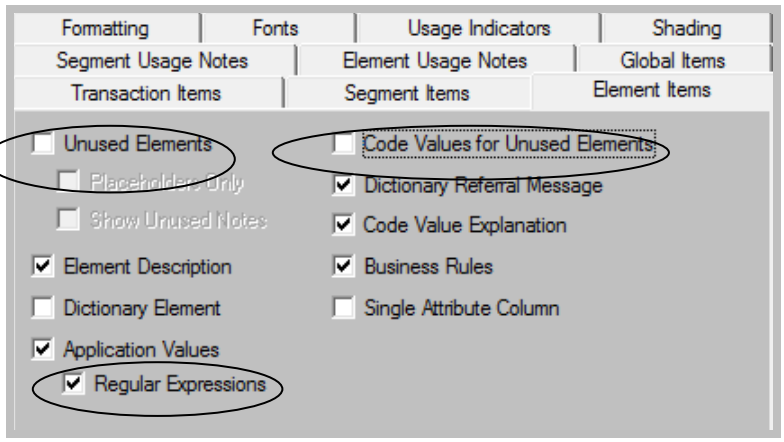
5. Click the **Segment Items** tab and clear the **Unused Segment** check box.

   This will eliminate unused segments in the Segments Table of the document.



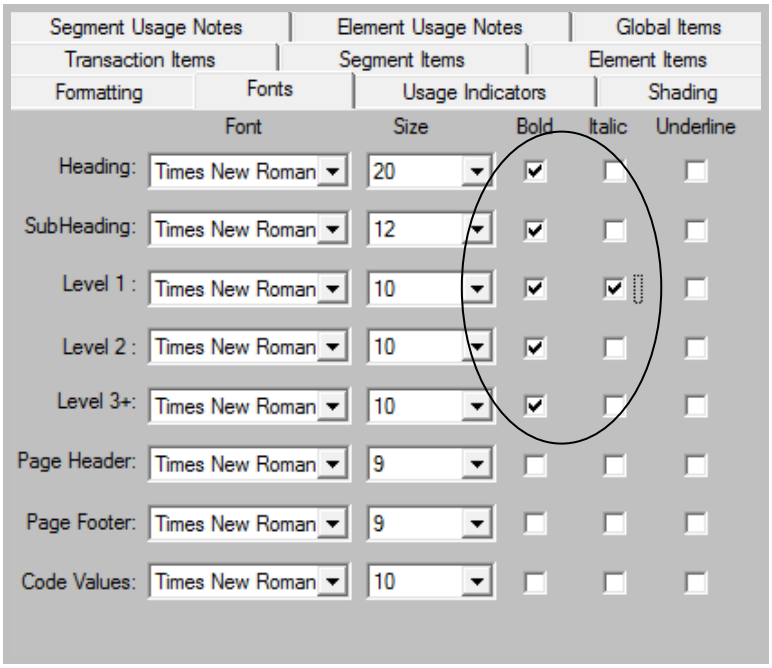
6. Click the **Element Items** tab and clear the check boxes for **Unused Element** and **Code Value for Unused Element**. Then, select the **Regular Expression** check box.

   This will eliminate unused elements and code values for unused elements, and will include the regular expression you created in Standards Editor.



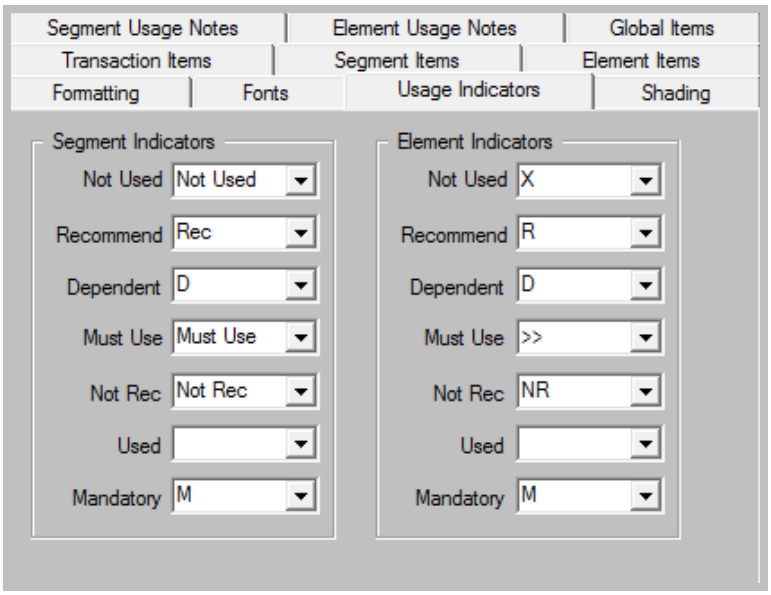
7. Click the **Fonts** tab and select the **Bold** and **Italic** check boxes for **Level 1** notes. Also, select only the **Bold** check boxes for **Level 2** notes and **Level 3+** notes.

   The italic font will make your Level 1 notes be more noticeable than the other notes.



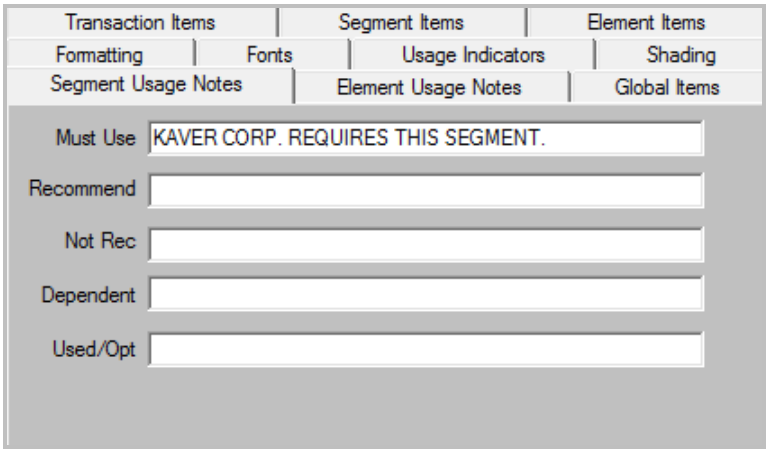
8. Click the **Usage Indicators** tab.

   This is where you decide what text to use for your usage indicators. For each indicator, you could select from the drop-down list, type your own, or choose to leave it blank. Leave the default settings for all items. Notice that Mandatory items will be represented by an M.

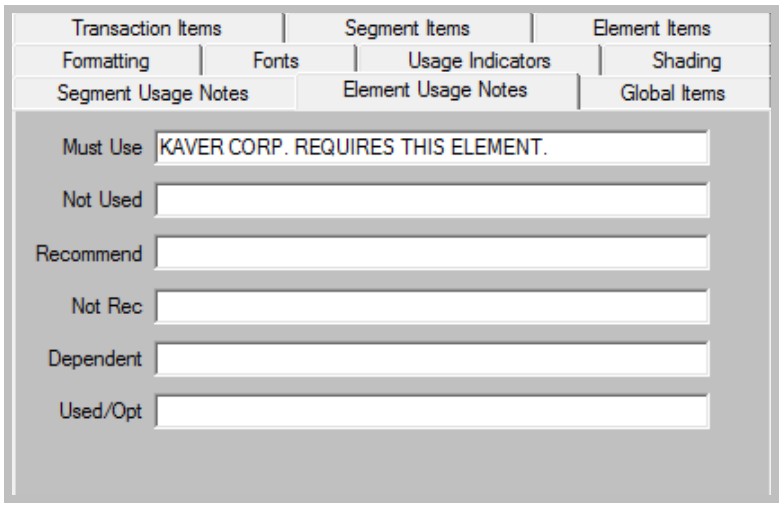


9. Click the **Segments Usage Notes** tab, and in the **Must Use** field, type **KAVER CORP. REQUIRES THIS SEGMENT.**

   This is where you can add your own text to further explain requirements.



10. Click the **Element Usage Notes** tab, and in the **Must Use** field, type **KAVER CORP. REQUIRES THIS ELEMENT.**



STOP

Comments

- For descriptions of the items on the Doc Builder Options dialog box, choose **Help | Index | Contents tab | Doc Builder Details**.

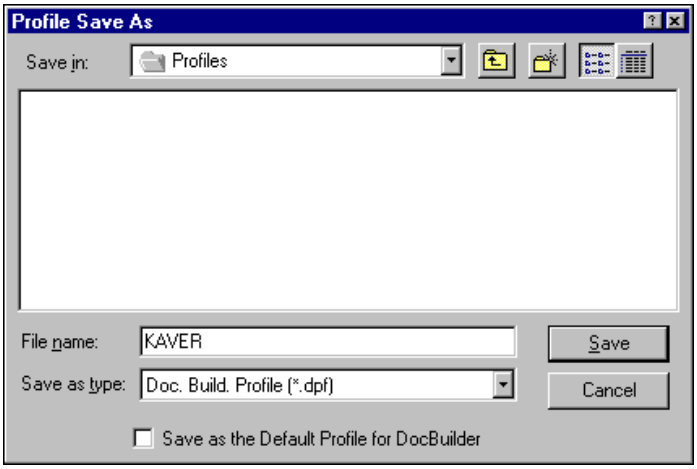
Saving Doc Builder Options

Now that we have customized our Doc Builder profile, we want to save it for future use. This not only saves time, but it ensures that our documents are consistent.

START

1. Click the **Save** button on the Doc Builder Options dialog box.
2. In the Profile Save As dialog box, click the **Up One Level** button  to go the **User Files** folder.
3. Click **Create New Folder** button  , name the folder **Profiles**, and press the ***Enter*** key twice so that 'Profiles' appears in the Save in field.
4. Type **KAVER** in the File name field.

   Notice that the profile will be saved as a .dpf file type. Also, notice that you could select the check box at the bottom to save your profile as the default, but do not select it. You would only save a customized profile as the default if you plan to use it for most of your documents.



5. Click **Save**.

   This profile will remain in effect until you close Doc Builder. The next time you open Doc Builder and want to use the saved profile, click the Load button on the Doc Builder Options dialog box and select the file.

   Note: You can load a profile before or after generating a document. If you load the profile afterwards, Doc Builder will ask if you want to regenerate it.

6. Click **OK** to close the Doc Builder Options dialog box.

7. Verify that the **ORDERS** Message is selected. Click **OK** in the Transaction Set/Message Selection dialog box.

   There will be a pause while the document generates.

   The ORDERS Purchase Order document displays in the EDISIM Doc Builder window. Leave it open for the next exercise.

STOP

Comments

- If a profile is saved as the default profile, it will be used every time you use Doc Builder or until you load a different profile. You can reset it to the original default settings by copying the $fsdeflt.dpf file located in the Bin folder to the Profiles folder (or where you save your profiles). Then, load $fsdeflt.dpf from your Profiles folder, select it as the default, and save it. This will prevent you from saving over the original default file in the Bin folder.

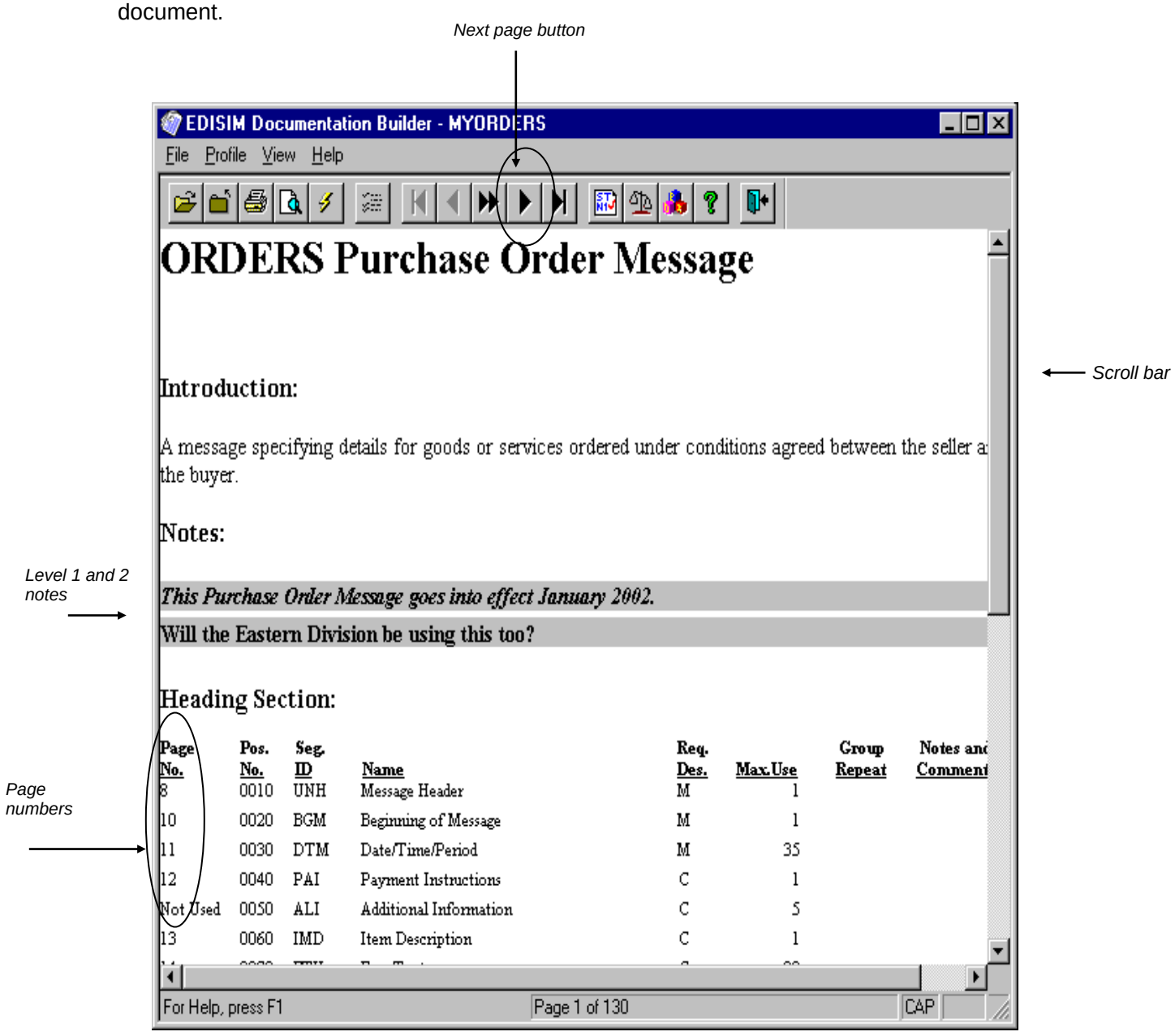
- You may want to also save the profile to a network drive, so others can use the same options.

Viewing the MIG Document on the Screen

The first page you see in your generated MIG is the beginning of the Segments Table, which serves as the table of contents for your document.

START

1. Look at the notes that you put on the message just under the Introduction section.
2. Click the **Next Page** button on the toolbar to advance one page at a time through the document.



3. Use the **Scroll bar** to view the rest of a page.

   Note: If a page has no scroll bar, all text is showing on the screen.

4. Find the toolbar buttons that take you to the last page, the first page, and that let you go to a particular page.

5. Click the **First Page** button  to go to page 1 and look up the page number of the **Ship To NAD**.

6. Use the **Go To** button  to go to the **Ship To NAD** page.

   Review the changes that you made there. Notice the shaded usage notes that you set.

7. Look up the page number for **CTA** at position **0220**, and use the **Go To** button to go to it.

8. Look at the **3139** element. It should be marked as “C” (Conditional) and have our customized list of code values.

9. Look for our notes on the **CTA** itself, on the **3139**, and on the **PQ** code.

STOP

Comments

- The *Pg Up* and *Pg Dn* keys move up and down on the same page.
- *Ctrl+Right Arrow* keys move to the next page; *Ctrl+Left Arrow* keys move to the previous page.
- You now have full documentation with minimal effort.

Using Print Preview

It's a good idea to check your document with Print Preview before printing, especially if you have changed fonts, headers, footers, or margins.

Note: Your computer must have a printer defined to use this feature.

START

1. Choose **File | Print Preview** or click the **Print Preview** button  .

   The full page is viewable and the cursor is now a magnifying glass.

2. Click 2 times on the page to see the two levels of magnification.

3. Notice that the cursor is now an arrow. Click once more to return the page to full view.

4. Use the **Next Page** and **Prev Page** buttons at the top.

   Notice the header and footers.

5. Click **Close** to return to normal view.

STOP

- You can print from Print Preview or from **File | Print**.

Exporting a Guideline or MIG to a Word Processor Format

You can export your MIG as an RTF (Rich Text Format) file. With an RTF file, you can add a company logo or watermark, send an electronic document to a trading partner, or make further formatting changes.

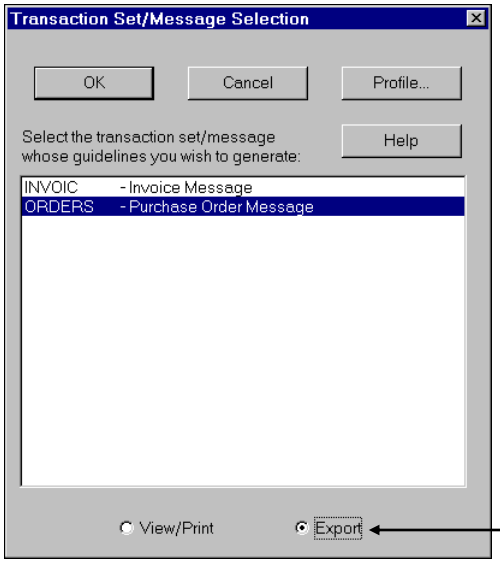
START

1. Choose **File | Generate Guidelines**.

   The Transaction Set/Message Selection dialog box reappears.

2. Click on **ORDERS Purchase Order Message**.

3. Click the **Export** button at the bottom.



4. Click **OK**.

   The Export Guidelines dialog box appears.

5. Click the **UP One Level** button to go to the **User Files** folder.

6. Click the **Create New Folder** button, name the folder **RTF Files**, and press ***Enter*** twice so that 'RTF Files' appears in the Save in field.

7. Name the file **MYORDERS.rtf.**

8. Click **Save**.

   You will again see the Generating Guidelines dialog box. You now have an RTF file saved in the RTF Files folder under EDISIM\User Files.

STOP

Comments

- You can edit the RTF document with a word processor, but be aware that any changes made to the document will not be made in EDISIM and cannot be imported. Do not make structural changes in an RTF file; make them in Standards Editor and then regenerate the document.

- If you want to share RTF files with others, you could export them to a folder on your public network.

- When exporting to RTF, there are a number of options available on the Export Guidelines Box. See Exporting to RTF in **TIB_fsp_edisim_<*n.n*>_fsdocbld.pdf** for more information.

Creating HTML Files

You can quickly create an HTML (Hypertext Markup Language) document from within Doc Builder. Your HTML document can be made available to others via the World Wide Web.

START

1. Choose **File | Create HTML**.

   The Create Web Page Options dialog box appears.

2. Click the **Up One Level** button to go the **User Files** folder.

3. Click the **Create New Folder** button, type **HTML Files** for the new folder name, and press ***Enter*** twice so that 'HTML Files' appears in the Save in field.

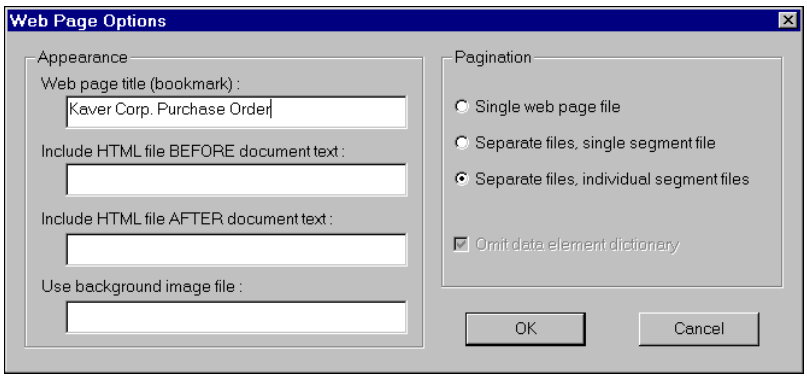
4. Name the file **MYORDERS.htm**, and click **Save**.

   The Web Page Options dialog box appears.

5. In the Web page title (bookmark) field, type **Kaver Corp. Purchase Order**.

6. Click **Separate files, individual segment files**, if not already selected.

   The Segments Table will be in one file and each segment will be in its own separate file.



7. Click **OK**.

   Document Builder generates the document.

8. Close your MIG and exit Doc Builder.

9. Open Explorer and locate the **MYORDERS.htm** document in the **EDISIM \User Files\HTML Files** folder.

10. Double-click **MYORDERS.htm** to view the file.

    Notice that there are no page numbers, but there are links to each segment's page. Also, notice that the entire Segments Table is on one page.

11. Click the link for the **CTA** and view your customized changes.

12. Scroll down to the bottom of the page and view the links. The **Previous** link will go the previous page, **Next** will go the next page, and **Up** will return to the Segments Table. Click these links several times, then return to the Segments Table.

13. Scroll down and click on the **Ship To NAD** link and again look at your customized changes.

14. When you have finished viewing the HTML file, choose **File | Close**.

15. Close Explorer.

STOP

Comments

Review the Web Page Options dialog box.

- Appearance

  1) Web page title (bookmark):

     (Optional but recommended) Enter text that will serve as the Bookmark or Favorites title. This is also used by web search engines and on browser's top title bar.

  2) Include HTML file BEFORE document text:

     Text or graphics, such as a logo or instructions, will print at the top of each HTML page. It is a file containing HTML fragments, not a complete HTML file. You must use the full directory address unless it is coming from the same folder where you are saving. Example: C:\ EDISIM \User Files\HTML Files\before.htm

  3) Include HTML file AFTER document text:

     Same as #2 above, except the text or graphics will print at the bottom of each page.

  4) Use background image file:

     Use this for a watermark. You must use the full directory address unless the image is coming from the same folder where you are saving. Example: C:\EDISIM\User Files\HTML Files\background.gif

- Pagination

  1) Single web page file:

     There will be one single web page. This method takes a long time to open in a web browser.

  2) Separate files, single segment file:

     Segment Table in one file with details of all segments in a second file.

3) Separate files, individual segment files:

   Segment Table menu file with each segment detail in its own separate file.

4) Omit data element dictionary (optional check box):

   Omits or includes entire data element dictionary in one file. Must have first turned on the dictionary element under **Profile | Options | Optional Items**.

Comparator

Purpose

The purpose of this exercise is to compare our MIG MYORDERS, which is based on D99A, to a MIG based on D01C. We then migrate our user-created changes to the new MIG. Changes that EDIFACT made between D99A and D01C cannot be migrated.

Preparing a D01C MIG for Migration

Let's assume that we are now preparing to migrate some MIGs to EDIFACT D01C. We do not want to start over from scratch–adding notes and defining code sets. We are going to compare our current ORDERS with a brand new MIG in D01C, then migrate all user-created changes to the new one.

START

1. Choose **Start | Programs | <*TIBCO_HOME*> | EDISIM | Standards Editor**. Maximize the window, if it isn't already maximized.

2. Standard's Editor opens. Let's create a new ORDERS based on D01C.

3. Choose **File | New**, select **D01C** from the Build New Standard dialog box, and click **Next**.

   The Select Messages from Standard File dialog box appears.

4. Type **ORDERS** and select by clicking in the box, and click **Open**.

5. Expand the **ORDERS Purchase Order Message** and Table 1.

6. Click on the **GROUP 2 NAD** at **0120**.

7. Choose **Edit | Copy**, and then choose **Edit | Paste** to insert an additional GROUP 2 NAD.

   This creates a MIG with the same number of GROUP 2's for our comparison. This also allows information from your second NAD group to be migrated.

8. You do not have to make any further changes to it at this time. Just save it by choosing **File | Save As**, and name it **NEWORD**.

9. Close the MIG and exit Standards Editor.

10. Choose **Start | Programs |<*TIBCO_HOME*> | EDISIM | Comparator**. Maximize the window, if it isn't already maximized.

STOP

Comments

- Migrate to a MIG rather than a published standard. Although, if you are simply going to compare, this is not important.

Choosing What to Compare

START

1. In Comparator, choose **File | New**.

   The Comparator Input dialog box appears.

2. In the Base Guideline list, select **MYORDERS**.

   This is the "present" standard, guideline, or MIG.

3. In the Base Transaction field, select **ORDERS**.

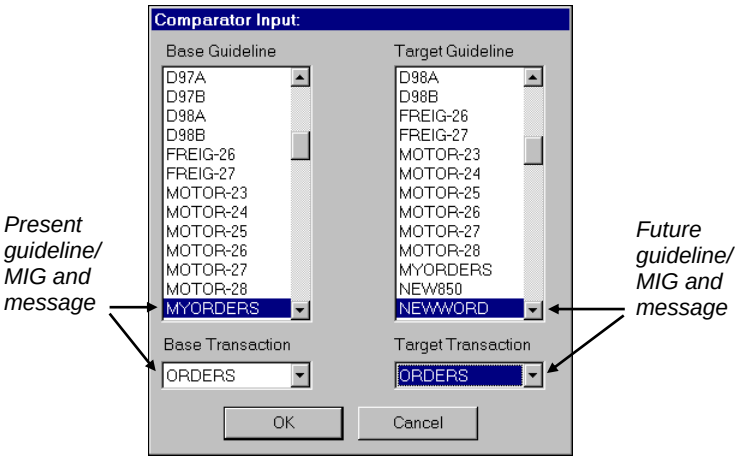
   This is the "present" message.

4. In the Target Guideline list, select **NEWORD.**

   This is the “future” standard, guideline, or MIG.

5. In the Target Transaction field, select **ORDERS**.

   This is the “future” message.



6. Click **OK**.

   The Input area at the top left now shows what is being compared (MYORDERS and NEWORD).

STOP

Changing Item Options

Before comparing the two MIGs, we'll need to designate the items that we want to compare. Turning off options that are not of interest will speed up the Comparator processing. Only compared options can be migrated.

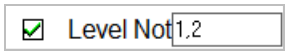
START

1. Click the **Msg** (message) item button in the Control pane on the left.

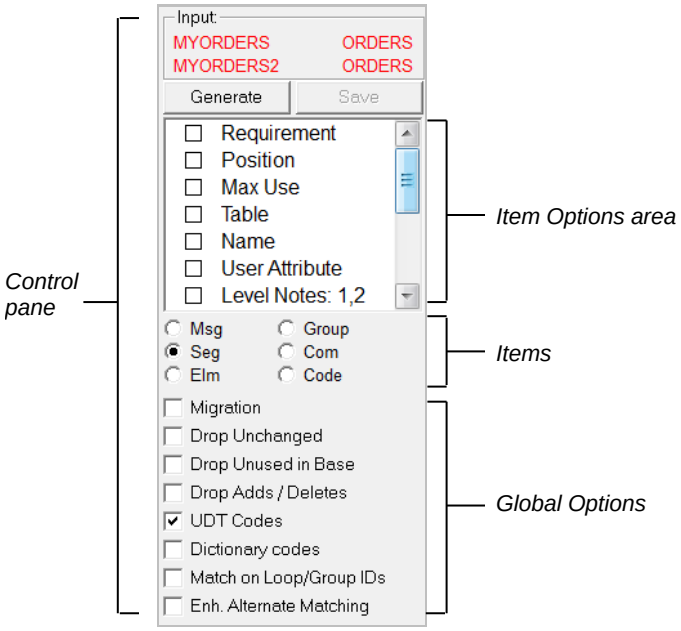
   The options for message comparisons appear in the Item Options area (the white area just above the item buttons).

2. Double-click in the Item Options area to clear all of the check boxes at once.

3. Select the **Level Notes** option to select the check box and then click to the right of the Level Notes label. An entry box appears. Enter **1,2**.

   For the message itself, we want to compare only level 1 and 2 notes. If we had left the * entry (the default), all levels of notes would be compared



4. Click the **Seg** (segment) item button. Double-click in the Item Options area again to clear all options, and select the following options (scroll down to see all of the options):

   Requirement
   Max Use
   Name
   User Attribute
   Level Notes (select levels 1, 2, 3)
   Business Rules

Important: Double-check your selections. Otherwise, if items are not selected, your NEWORD guideline will not contain all of your customized changes.

5. Click the **Elm** (element) item button and clear all options. Select the following options:

   Requirement
   Min Length
   Max Length
   User Attribute
   Type
   Level Notes (select levels 1, 2, 3)
   Appl Values
   Business Rules

6. Click the **Group** item button and clear all options. Select the following options:

   Requirement
   Max Use
   User Attribute
   Level Notes (select levels 1, 2, 3)

7. Click **Com** (composite) and clear all options. Select the following options:

   Requirement
   User Attribute
   Level Notes (select levels 1, 2, 3)

8. Click **Code** and clear all options. Select the following options:

   Used
   Level Notes (select levels 1, 2, 3)
   Description
   Subsets

STOP

Comments

- When migrating, if the base and the target both have a code set for a particular element, selecting the Merge Values option for the Code item will combine both code sets. If the merge option is not selected, the base code set will overwrite the target code set. This is also true for application values (Merge Appl Values) for the Elm item.

Setting Global Options and Generating a Comparison

Below the Item Options buttons are some check boxes that let you select global options. These are not tied to a particular kind of item such as a segment. Global options also affect what will be compared.

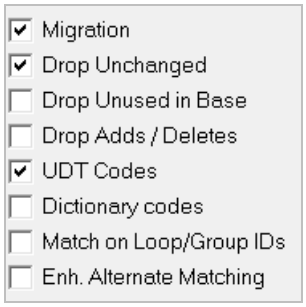
START

1. Select the **Migration** check box.

   This option must be on before you generate, if you intend to migrate.

2. Select **Drop Unchanged**, so only changes between the base and the target are included.

3. Select **UDT Codes**, so User Defined Transactions (code sets) will be compared.



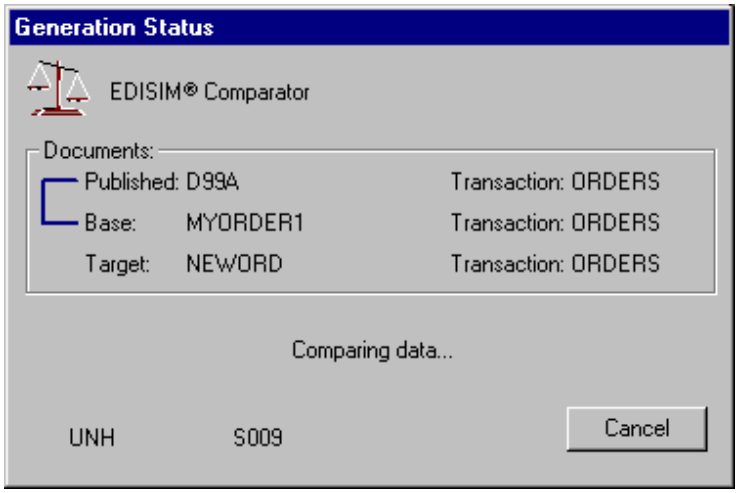
4. Now, we want to save our settings. Choose **File | Save As**.

5. In the Save in field, open the **Profiles** folder under **EDISIM \User Files**, and name the file **ORDERS.dlt**. Click **Save**.

   This saves the options. You can use **File | Open** to re-use these settings.

6. Click the **Generate** button in the top left area.

   The standards being compared display throughout the generation process.



STOP

Comments

- Global Options:

    1) Drop Unchanged - if compared items are the same in the base and target, do not display the item at all. Select this option for migration.

    2) Drop Unused in Base - if an item has been marked unused in the base, do not compare, display or migrate the item. Turn this off when migrating.

    3) Drop Adds/Deletes - only display items that are in both the base and the target.

    4) UDT (User Defined Transaction) Codes - compare the selected options for code sets used in the message. It will not compare unchanged EDIFACT code lists.

    5) Dictionary Codes - compare the selected options for dictionary codes. This will compare code lists from the underlying standard, not your UDT codes. This uses lots of processing time.

    6) Match on Loop/Group IDs – match loops and groups based on their Loop ID.

    7) Enhanced Alternate Matching – match alternate segments (consecutive segments with the same tag) based on their first code value(s).

- First, Comparator finds differences between the base and its underlying published standard, and then it compares this list of user changes to the target. These user changes are eligible for migration.

Viewing the Comparison

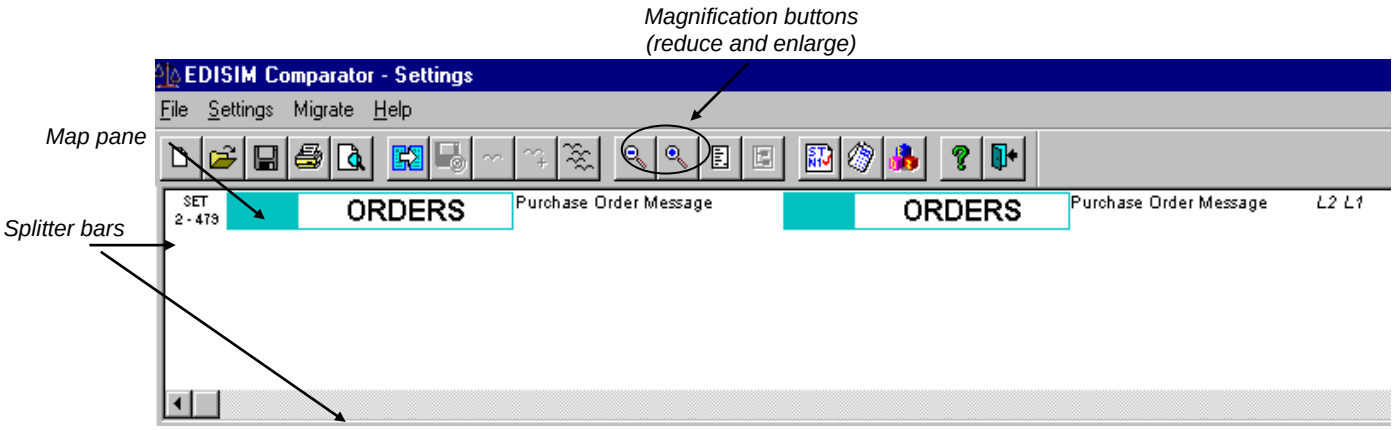
START

1. Adjust the Map pane, so that you can see the entire **ORDERS Purchase Order Message** line:

   a) Drag the vertical splitter bar to the left as far as possible.

   b) Click on the **ORDERS Purchase Order Message** line with the blue rectangles, then adjust the size by repeatedly clicking the magnification buttons on the toolbar.

   c) You may also move the horizontal splitter bar downward to view more of the upper pane.



2. Choose **Settings** | **Expand All Nodes**.

   You will see the groups and segments in the ORDERS. Comparator is showing MYORDERS (base) on the left and NEWORD (target) on the right. Notice the red DEL and the blue INS in the first column, which indicate deleted and inserted EDIFACT items in the new MIG.

3. Double-click on the **ORDERS** to close all nodes.

4. Double-click on the **ORDERS** to expand the segments and groups.

5. Scroll down to the first **NAD** group (Payer) and double-click on it. Double-click on the **CTA** group at **0210**, and also the **CTA** segment at **0220**.

6. Look in the first column of the line containing the **CTA** segment. You will see this information:

   SEG
   3 – 9

   This means CTA is a segment, there are 3 changes to the CTA itself, and there are 9 changes to the subordinates of CTA.

7. Let's find some of the changes. With the **CTA** segment at **0220** selected, look at the bottom half of the screen in the Detail pane.

   The CTA has Level 1, 2, and 3 Notes in MYORDERS but not in NEWORD.

8. Let's find the changes to the subordinates of CTA. Double-click on composite **C056** (CTA02), then click on subelement **3413**. Look in the Detail pane to see the changes to the Max Length and Min Length.

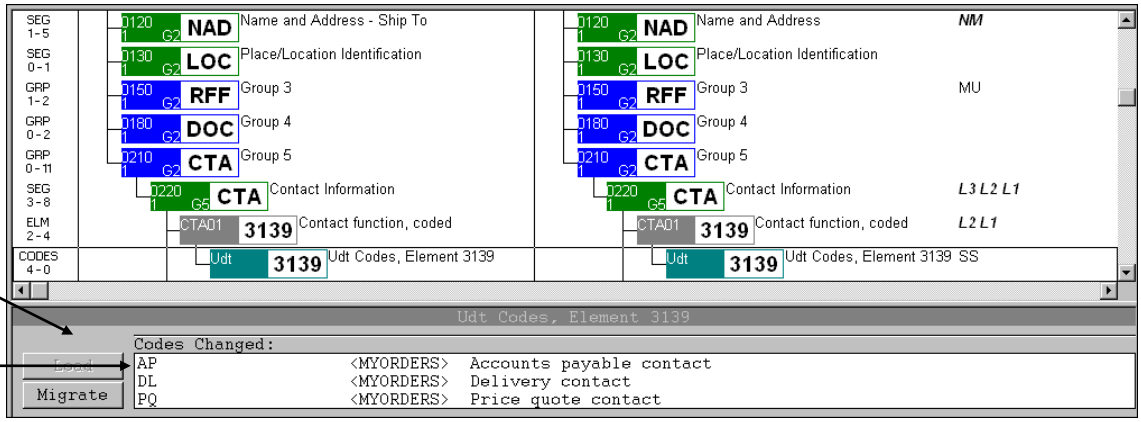
9. Click on element **3139** (CTA01). Look in the Detail pane to see the changes.

   It has a level 1 and level 2 note, and it is marked "Must Be Used".

10. Double-click on element **3139** to expose the Udt codes.

11. Click on the **Udt codes** line for the **3139**, and press the **Load** button in the Detail pane.

    Changed codes appear in the Codes Changed area as shown in the following example.



Detail pane

Changed codes for 3139

12. These three codes are in a code set in MYORDERS, but not in NEWORD.

    We made these code changes to MYORDERS in Standards Editor.

    If a code in this area had displayed <modified>, you would have to click on it to see the modification in the Detail of Changes area.

STOP

Changing Views (Optional Exercise)

Before migrating your changes, you may want to review a text report that shows the differences between your base and target. You could use the text report as a worksheet.

Note: Your computer must have a printer defined to use the Print Preview feature.

START

1. Choose **Settings | Text Mode View** or click the **Text View** button  on the toolbar.

   You will see a text version of the comparison.

2. Choose **File | Save Text Report**.

3. Create a new folder named **TXT Files** in the **User Files** folder, and press ***Enter*** twice so that 'TXT Files' displays in the Save in field. Change the File name to **ORDERS.txt**. Click **Save**.

4. Choose **File | Print Preview**.

   The text file displays in Print Preview.

5. Click the **Zoom In** button twice to zoom in completely.

   An asterisk (*) indicates that an item is different between and base and the target.
   A plus sign (+) indicates that an EDIFACT item has been added to the target.
   A minus sign (-) indicates that an EDIFACT item has been removed from the target.

6. Click the **Next Page** button several times and view the report.

   Notice the Print button – you could print the report from here.

7. Click **Close** to exit Print Preview.

8. Choose **Settings | Graphic View** or click the **Graphical View** button  to return to the previous view.

STOP

Migrating User-Created Changes

Almost any user changes to the base can be migrated to the target. Exceptions: Group Tags and dependency rule changes. Code sets migrate, but a user-created code set used in multiple places will migrate as separate code sets.

START

1. Since we want to have these code changes in NEWORD, let's just migrate them now. Click the **Migrate** button.

   The entire code set migrates. The codes disappear, since the migration made the base and target the same for these codes.

2. Select the **CTA** segment (not the CTA group) in the Payer NAD group.

   The notes that we added appear at the bottom.

3. We cannot see the entire text of the notes, so we don't know if we want to migrate them. Double-click on **Level 2 Notes**.

   The Text Viewer dialog box appears.

4. You can edit the note before you migrate it, if you choose. The edits would affect only the target (the base is never changed by Comparator). Click the **Migrate** button on the Text Viewer dialog box.

   The level 2 note migrates and the Text Viewer closes. If you didn't edit the note, Comparator will no longer display it, since the level 2 note for both the base and the target are the same. If you did edit the note, Comparator displays level 2 notes for both the base and the target since they differ.

5. Select **Level 1 Notes**, and click **Migrate**.

   The level 1 note migrates and Comparator no longer displays it.

6. Select the **CTA01** (**3139**). There are level 1 and 2 notes and the user attribute is marked "Must be Used" in the base. Hold down the ***Shift Key*** and the left mouse button and drag slowly across the gray box in front of the 3139 in the base. When the cursor turns into a little bottle with a **+**symbol, drag it to the corresponding gray box in the target and watch the bottle "dump." Then release the mouse button.

   This method migrates changes for the segment and all its subordinates all at once.

   Two other ways to do this are selecting the Migrate Current and Subordinates button  on the toolbar or choosing **Migrate | Current**.

7. Let's migrate the remaining user-created changes. Click the **Migrate All** button  or choose **Migrate | All**.

   All user-created changes migrate. Only EDIFACT changes are left.

STOP

Comments

- Ways to use Comparator:
  1) Compare a published standard (D99A) to another published standard (D01A).
  2) Compare a MIG to a published standard to see the changes that have been made.
  3) Compare MIG to MIG.
- Benefits of comparisons:
  1) View required changes.
  2) View new or deleted elements and segments.
  3) Assess the impact and make decisions before making changes.
  4) Determine what the trading partner is requesting.

Saving the Target MIG

START

1. We want to save the target guideline now that we've migrated our customized changes. Choose **File | Save Target** or click the **Save Target** button  .

   The Save As New Standard dialog box opens.

2. Leave the Standard Name as **NEWORD** and replace the Description text with **D01C ORDERS for EDISIM Training Class**. Click **OK**.

3. Choose **File | Exit**, answer **Yes** to Save the changes dialog box.

4. Open **NEWORD** in Standards Editor

   Let's review the changes that were migrated.

5. Expand the **ORDERS Purchase Order Message, Edit | Level Notes** to review our Level 1 note, then click **OK**.

6. Expand the **Ship To NAD**. Click on the **3035** element at **NAD01,** select the Code Values tab and look at the migrated code set.

   Notice the migrated segment name change – the inclusion of Ship To.

7. Expand the **CTA** segment in the **Payer NAD** group. Click on element **3139** at **CTA01** to see the migrated code set.

8. Close your MIG and exit Standards Editor.

STOP

Analyzer

Purpose

The purpose of this exercise is to check EDI test data. First, we will see if it complies with EDIFACT requirements, and then if it complies with requirements in our locally-created MIG MYORDERS. The same approach can be used with production EDI data - inbound or outbound.

Why use Analyzer to check compliance?

- Analyzer checks against published standards (EDIFACT, EANCOM, etc.) or against company MIGs.
- Some personnel who have responsibility for testing trading partners' data do not have ready access to the translator. The EDISIM Analyzer provides "desktop" checking.
- Analyzer's dual-panel display is intuitive and easy to navigate.
- Analyzer's diagnostic messages are complete and easy to understand, and the printed reports with interleaved EDI data and diagnostic messages are clear.
- Analyzer checks thoroughly, especially with its ability to realign despite missing segments, to identify whether a segment is inside or outside a group, to check whether "not used" segments and elements are there, and to check application values and business rules.

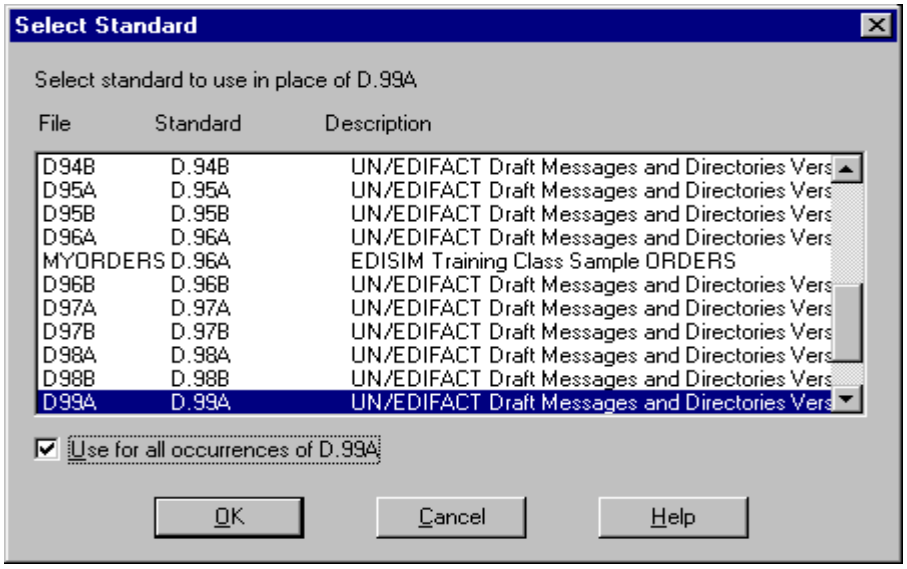
Choosing a File and a Standard

Whenever you have a new MIG, you'll always want to analyze it against a published standard.

START

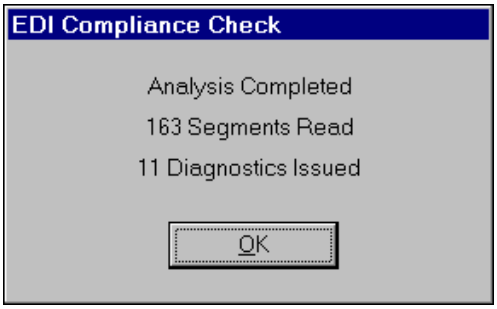
1. Choose **Start | Programs | <*TIBCO_HOME*> | EDISIM | Analyzer**.
2. Choose **File | Open**.
3. Go to the **EDISIM \Samples** folder.
4. Click on **testpom.txt**, and click **Open**.

   The Select Standard dialog box appears. Analyzer prompts you to select a standard (or MIG) against which to compare the data. The list is sorted by the Standard column.
5. Single click on **D99A**.
6. Click the **Use for all occurrences of D.99A** check box. Otherwise, Analyzer would stop and ask for a version each time it encounters a UNH segment in the file. Your selections should look like the following example.



7. Click **OK**.

   The EDI Compliance Check dialog box appears.



8. When the analysis is complete, click **OK**.

STOP

Comments

- Sources of data for Analyzer:

  1) A trading partner test or rejected transmission inbound.

  2) A test from the translator for an outbound message.

- You can drag a MIG or standard file from Explorer and drop it directly into the Analyzer window.

Checking the Diagnostics

START

1. Maximize the Analyzer window, if necessary.
2. Scroll to the top in the **Diagnostics** pane.
3. Press the ***Down Arrow*** through the diagnostics, while noticing how Analyzer highlights the corresponding segment in the Segments pane at the bottom.
4. Choose **File | Close**.

STOP

Comments

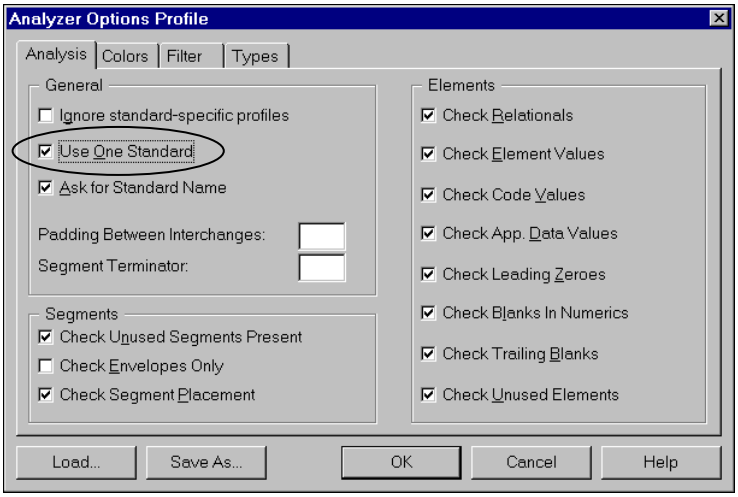
- Each message in this exercise is from the D99A perspective.
- Diagnostic message explanations are located in the Help file under the Contents tab.
- You can adjust the splitter bar between the panes, and you can also zoom a pane.

Changing Analysis Options

START

1. Click **Options | Analyzer Profile**. The Analyzer Profile dialog box appears. On the Analysis tab, select the **Use One Standard** check box in the General area.

   "Use One Standard" will use the enveloping segments within your MIG dictionary in place of the default enveloping. Do not turn this on if you are analyzing multiple versions within one file.



2. Press **F1** to go to Help. Click on the **Index** tab, type **Analysis Options**, and click **Display**.

3. Scroll through the Analysis Options Help information, then exit Help.

4. Click **OK**.

STOP

Analyzing Against Our MYORDERS MIG

START

1. Choose **File | Open**.
2. Select **testpom.txt** again, and click **Open**.
3. Click on **MYORDERS** in the Select Standard dialog box.

   The option “Use for all occurrences of D.99A” does not appear since we chose “Use One Standard” in the Profile Options.
4. Click **OK**.

   The Edit Compliance Check dialog box appears and analyzes the data.
5. When the analysis is complete, click **OK**.
6. Press the ***Up Arrow*** to go the top of the Diagnostics pane.
7. Press the ***Down Arrow***, while noticing how Analyzer highlights the corresponding segment in the Segments pane.

STOP

Comments

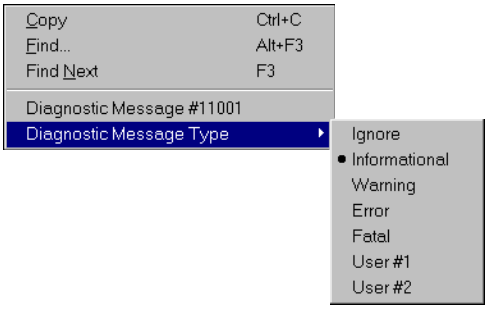
- Each diagnostic message in this exercise is from the MYORDERS perspective.
- Diagnostic Message explanations are located in Help > Index under the Diagnostics and in the Diagnostics section of **TIB_fsp_edisim_<*n.n*>_fsanalyze.pdf**.

Changing Diagnostic Message Colors (Optional Exercise)

By default, Analyzer displays all diagnostic messages, and they have pre-assigned colors representing their severity level or type. You can change the colors of diagnostic message types.

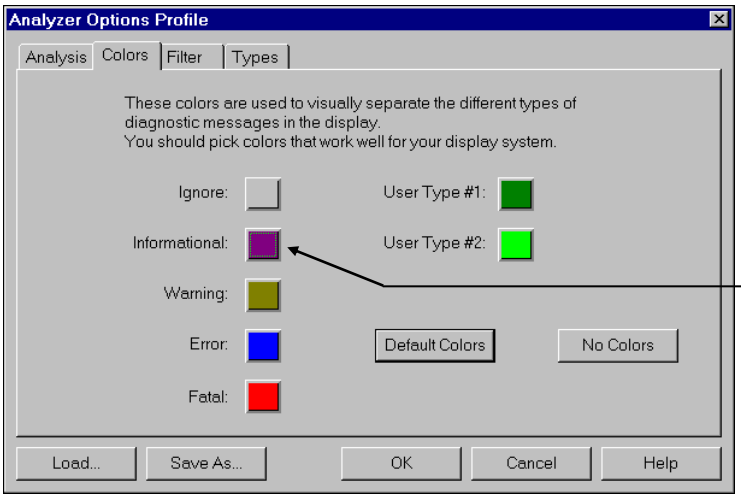
START

1. Before changing the display of a diagnostic message, let's determine its "type." Click the right mouse button on any dark gray message in the Diagnostics pane, then highlight **Diagnostic Message Type**. On the submenu, a bullet precedes "Informational" indicating that this is the message type.



2. Now let's change the color of the Informational type diagnostic messages. Choose **Options | Analyzer Profile | Colors** tab.

3. Click on the **dark gray square** beside **Informational**. In the Color selection box, click the color **purple** (the next to last color in the fourth row), and click **OK**.



The Informational color square displays in the color purple.

4. Click **OK** again.

   Analyzer displays all informational messages in the color purple.

STOP

Comments

- The Default Colors button resets the colors to the original settings.
- The No Colors button displays all diagnostic messages in black.

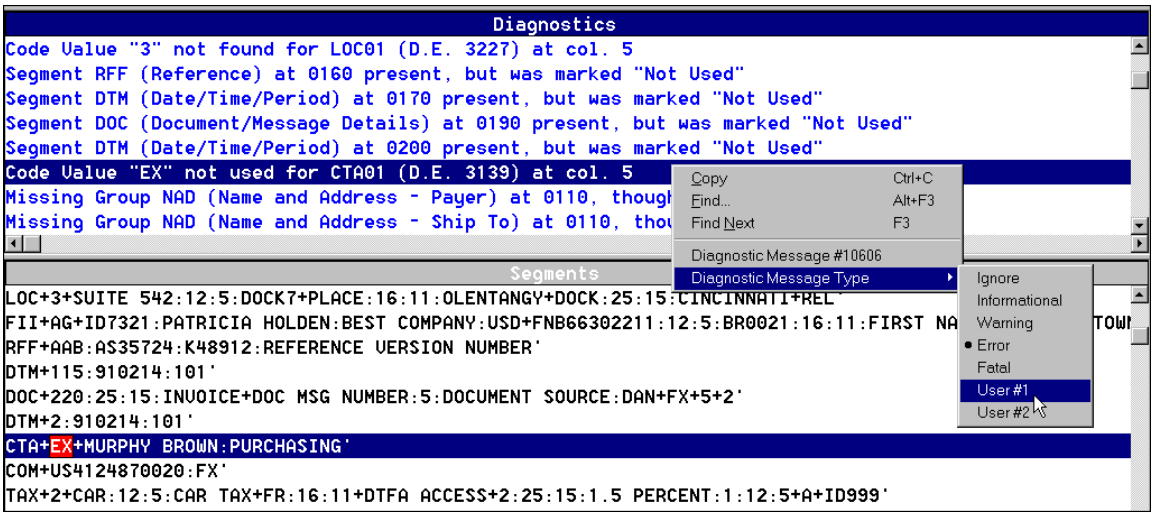
Changing Diagnostic Message Types (Optional Exercise)

You can change the type of a diagnostic message. Let's assume that we are paying particular attention to code value violations and want them to appear in their own color.

START

1. In the Diagnostics pane, scroll down until you see **Code Value "EX" not used for CTA01 (D. E. 3139) at col. 5**. Click the right mouse button over this message, then highlight **Diagnostic Message Type**. It is currently an Error type, which displays in blue.

2. Select **User #1** on the submenu.



3. Click on another message so you can view the message that you just changed.

   The code value changed from an Error type to a User #1 type, and now displays in dark green text. Notice that other code value messages also display in dark green.

4. Right-click on any code value message and notice the diagnostic number is 10606.

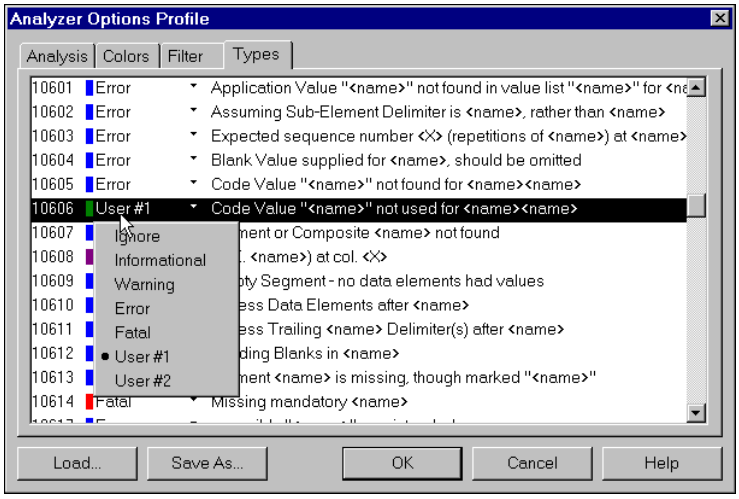
   Whenever you change a diagnostic message type, any message with the same diagnostic number is affected.

5. Choose **Options | Analyzer Profile | Types** tab. All diagnostic messages are listed in numerical order.

6. Scroll down until you see **#10606** (the message we just changed). Notice that the message shows a green bar and User #1.

7. Click *directly* on **User #1**.

   A popup menu displays the types. This menu is another way of changing a diagnostic message type.



STOP

Comments

- Diagnostic messages are pre-assigned to one of these types:

ANALYZER DIAGNOSTIC MESSAGE TYPES

Type	Default Color	Default Meaning
Ignore	Light gray	For your use.
Informational	Dark gray	Non-standard diagnostics such as cancellation of analysis.
Warning	Olive	Minor EDI data errors such as leading zeros in numerics, or assumptions that Analyzer had to make when data was ambiguous.
Error	Blue	Typical EDI data errors such as wrong codes, missing mandatory items, etc.
Fatal	Red	Error that prevents analysis from completing.
User #1	Dark green	For your use.
User #2	Light green	For your use.

Saving an Analyzer Profile

Now that we have customized the options for the Analyzer Profile, we'll want to save the profile for future use.

START

1. Click the **Save As** button.

2. Open the **Profiles** folder under the User Files folder, and name the file **Myprofile**.

   Notice that the profile will be saved as an .apf file type. Also, notice that you could select the check box at the bottom to save your profile as the default, but do not select it. You would only save a customized profile as the default if you plan to use it for most of your files.

3. Click **Save**.

   This profile will remain in effect until you close Analyzer. The next time you open Analyzer and want to use the saved profile, choose Load Profile from the Options menu and select the file.

4. Click **OK** to close the Analyzer Profile dialog box.

STOP

Comments

- If a profile is saved as the default profile, it will be used every time you use Analyzer or until you load a different profile. You can reset the profile to the original default settings by copying the $fsdeflt.apf file located in the Bin folder to the Profiles folder (or where you save your profiles). Then, load $fsdeflt.apf from your Profiles folder, select it as the default, and save it. This will prevent you from saving over the original default file in the Bin folder.

- You can save the profile to a network drive, so others can use the same options.

- You may want to save multiple profiles, depending on what you are analyzing. For example, you may want a MIG-specific profile, a standard-specific profile, or a trading partner-specific profile.

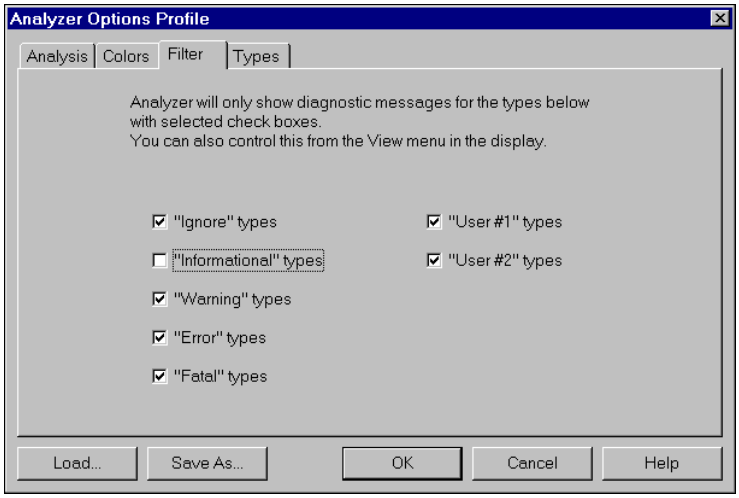
Filtering Diagnostic Messages (Optional Exercise)

You can hide or “filter” the display of diagnostic message types. Filtering lets you view or print only the message types that you choose.

START

1. Let’s no longer display the informational messages. Choose **Options | Analyzer Profile | Filter** tab.

2. Clear the **Informational** check box, so that your screen appears like the following example.



3. Click **OK**.

   Analyzer no longer displays the informational messages.

STOP

Comments

- You can also hide or filter a particular diagnostic number. Choose **Options | Analyzer Profile | Types** tab. Change that diagnostic number’s type to one that you are filtering.

Printing from Analyzer

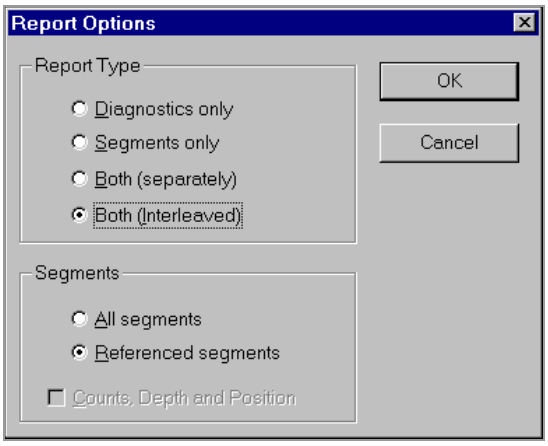
Analyzer provides the ability to print the diagnostics and segments to a printer or to a file.

Note: Your computer must have a printer defined to use the **File | Print** feature.

START

1. Choose **File | Print to File**.
2. Click **Both (Interleaved)**, if not already selected.

   These are the recommended options. The diagnostics and segments will print paired together, with each diagnostic message printed after the segment to which it refers.



3. Click **OK**.

   The **Text File Format** dialog box appears. This is where you would make page format changes such as margins and page width.

4. Click **OK**.

   The Print Text File dialog box appears.

5. In the **Save As** dialog box, select the **TXT Files** folder under **EDISIM \User Files** and change the file name to **Diagmsg.txt**. Click **Save**.
6. Open Windows Explorer and open **Diagmsg.txt**. View the diagnostic messages.
7. Now, let's go over how to send the output to a printer. Choose **File | Print**.

   The Report Options dialog box appears.

8. Click **Both (Separately)** and click **Referenced segments**, if not already selected.

   Both the Diagnostics and Segments will print separately.

9. Click **OK**, but click **Cancel** in the Print dialog box since we are not actually printing right now.
10. Close the file.

STOP

Comments

- For details on report options, search for “Report Options box” in Help under the Index tab.

Using DocStarter

DocStarter lets you start with one or more EDI data files and create an EDISIM MIG based on them. The MIG can contain one or more messages, with segments, elements, and codes that are customized to match the data in the EDI file.

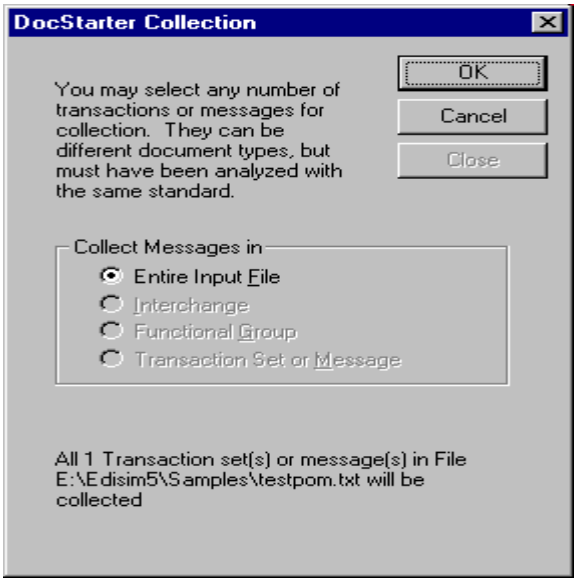
Creating a DocStarter Collection File

START

1. Choose **File | Open**, and select **testpom.txt** from the Samples folder, then click **Open**.
2. In the Select Standard dialog box, select **D99A** and click **OK**.

   For a new MIG, always check against a published standard.
3. Click **OK** in the EDI Compliance Check dialog box.

   View the diagnostics to see if you want to base the MIG on this data.
4. Choose **File | DocStarter Collection**. The DocStarter Collection dialog box appears.



5. Leave the default setting of **Entire Input File** selected.

   This setting collects information from all interchanges.
6. Click **OK**.

   The Open DocStarter Collection File dialog box appears.

7. Create a new folder named **COL Files** in the User Files folder, and press ***Enter*** twice so that 'COL Files' displays in the Save in field. Name the file **Collect.col** and click **Save**.

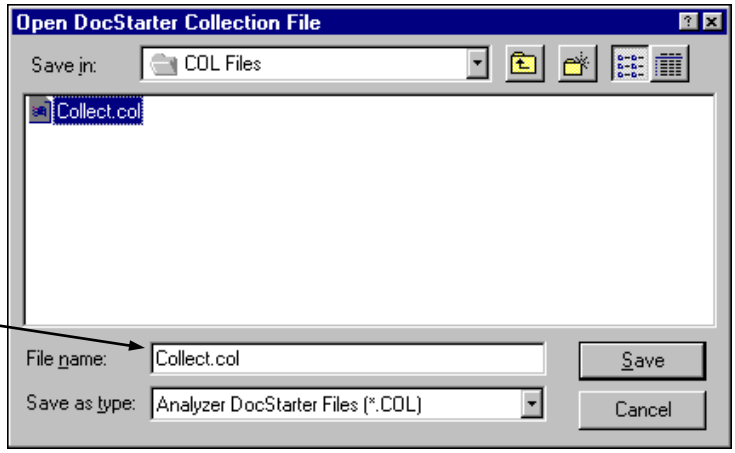
   This file starts a new collection using the data from testpom.txt.

8. When "Collection Complete" displays, click **Close**.

9. Let's add another file to the DocStarter collection that we just created. Repeat steps 1-6, except in step 1 select the file **testpom1.txt**, instead of textpom.txt. After you're finished, continue with step 10.

10. Open the **COL Files** folder under the Database folder, and click on **Collect.col** so that it appears in the File name field.

Save to the same collection file



11. Click **Save**.

    All of the data from testpom1.txt is added to the collection. Now, you have a collection file that contains data from two files: testpom.txt and testpom1.txt.

    Note: You could continue repeating this process, analyzing other EDI files that can help you define the MIG, and saving the results to the collection file. More collected files yield better MIGs.

12. Close the DocStarter collection dialog box.

13. Choose **File | Exit** to exit Analyzer.

STOP

Importing a DocStarter Collection File

Now, let's import the collection file we just created into Standards Editor.

START

1. Open Standards Editor.
2. Choose **File | Import | Import Analyzer DocStarter File**.
3. Click the **Up One Level** button, and select **EDISIM \User Files\COL Files\Collect.col**, then click **Open**.

   The DocStarter Analysis Options dialog box appears.
4. Leave the check box for **Remove Unused Transactions or Messages** selected.

   DocStarter will only include transactions that it finds in the EDI data.

   Note: Retain Local Code Values includes all user-created code values from the data. Be aware that these code values may not be EDIFACT compliant.

5. Click **OK**.

   Standards Editor imports the DocStarter collection.
6. Choose **File | Save As**. Save the file as Standard Name **TESTORD**. Replace the description text with **D99A Purchase Order created from test data**.
7. Select **Included in this Standard** under Interchange Envelope Segments. Click **Save**.
8. Look at the codes in the **PAI** segment. These codes came from the EDI data files.
9. Exit Standards Editor.

STOP

Comments

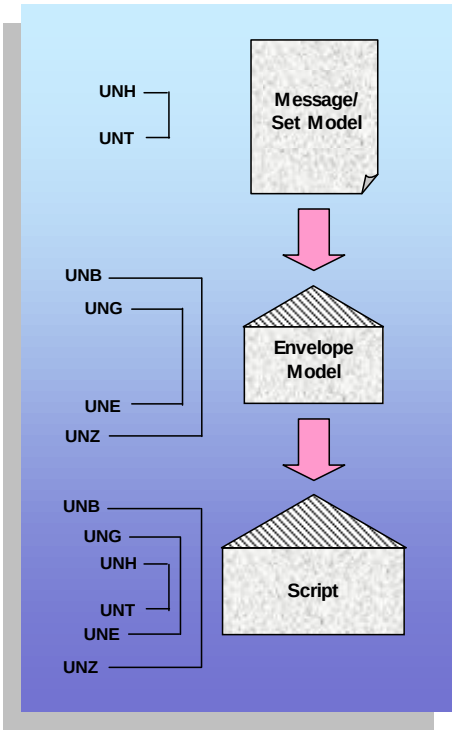
- The MIG will be as complete and accurate as possible, based on the EDI data files. However, you will probably want to further refine it by editing it in Standards Editor.

  Important: If you customize the imported DocStarter MIG in Standards Editor, then collect more files using DocStarter, you must save the newly collected files under another name in Standards Editor. Then use Comparator to compare the two MIGs and migrate the changes to the customized MIG.

Test Data Generator

Purpose

The purpose of this exercise is to create test data that is based on our MIG NEWORD. To create data, we will need to create both a message/set model and an enveloping model and insert these pieces into a script. This script is then processed to create our test data.



Creating a Message Model

First, let's create a message/set model.

START

1. Choose **Start | Programs | <*TIBCO_HOME*> | EDISIM | Test Data Generator**.

   Test Data Generator opens.

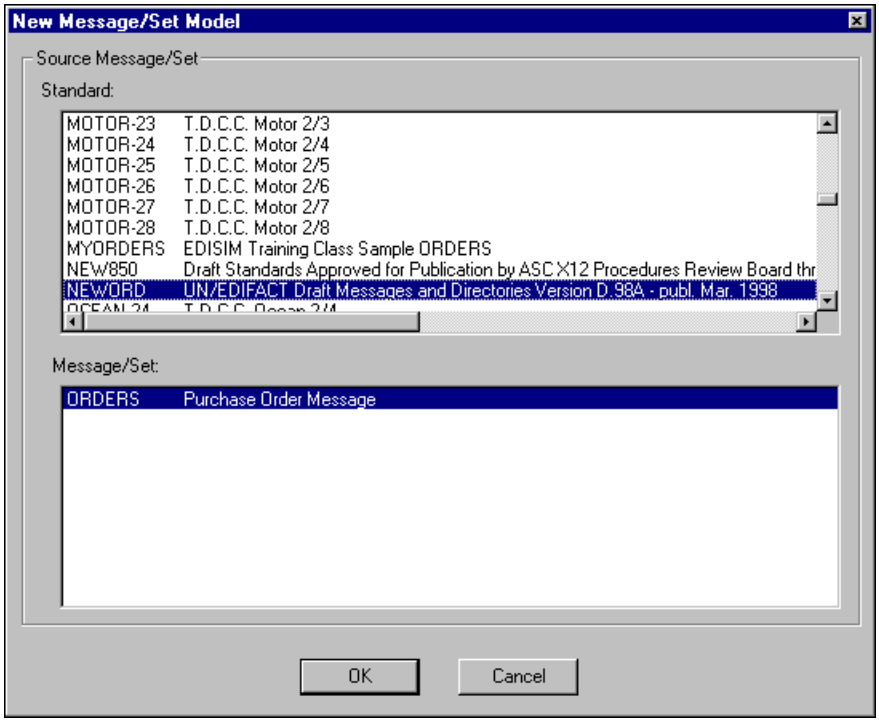
2. Maximize the Test Data Generator window, in necessary.

3. Click **File | New | New Message/Set Model**.

   The New Message/Set Model dialog box appears.

4. Select **NEWORD** in the Standard list.

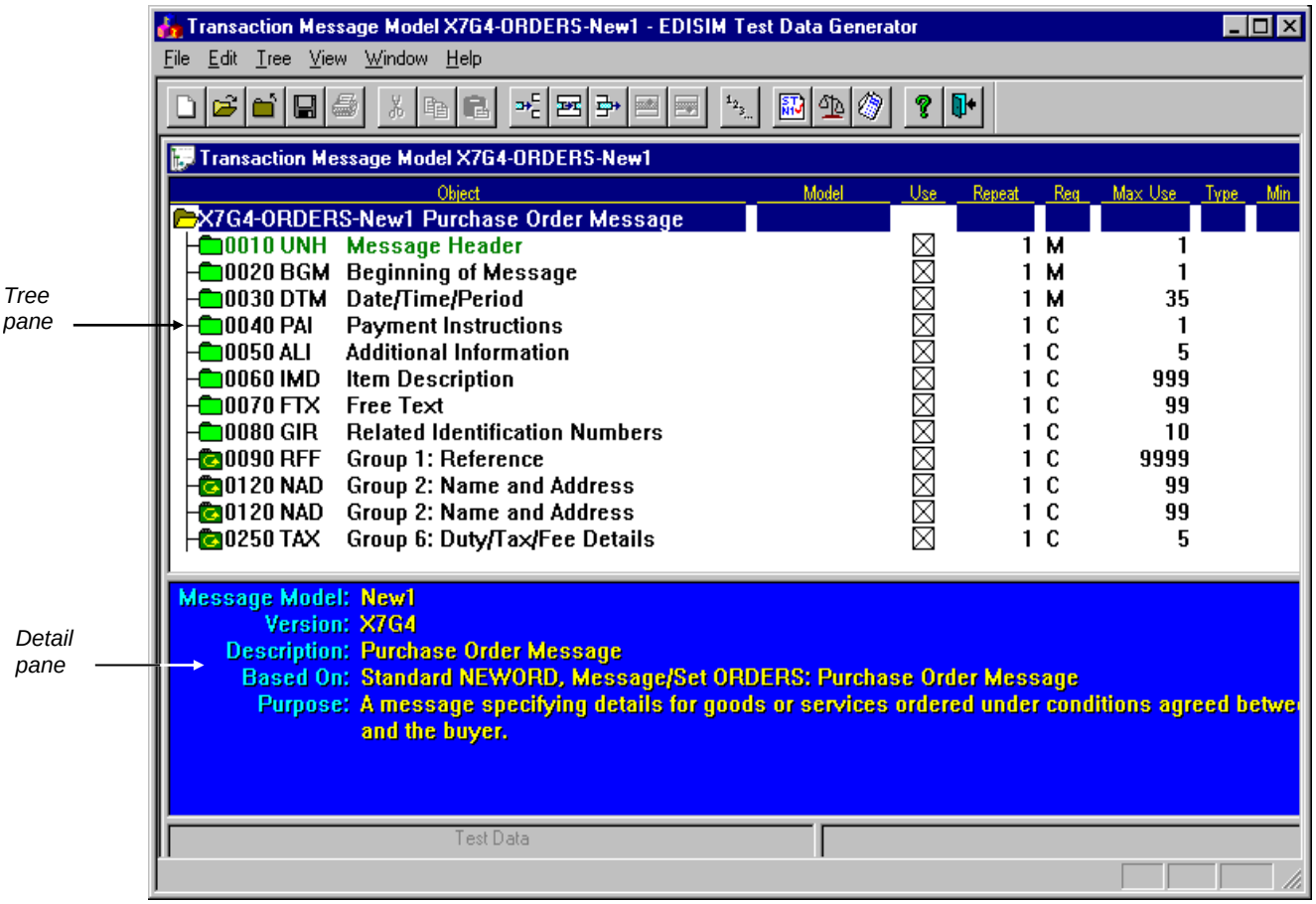
   You will see a list of messages for the chosen standard in the Message/Set list at the bottom. There is only one: ORDERS.



5. Click **OK**.

   Test Data Generator (TDG) displays the message model.

6. If necessary, maximize the Transaction Message Model window. Drag the splitter bars until the screen is similar to the following example.



The ORDERS MIG items appear in the Tree pane at the top. The Detail pane in the middle displays information about what is highlighted in the Tree pane (ORDERS Purchase Order).

Notice that TDG assigns a version number to a new Message/Set model. Version numbers are only for identification purposes.

7. Choose **File | Save As**. In the Save Model As dialog box, type **TEST1** in the Name field, and replace the text in the description field with **Test 1 - Default Test Data**. Click **OK**.

STOP

Viewing the Data

Let's look at the data the TDG is using in our message model.

START

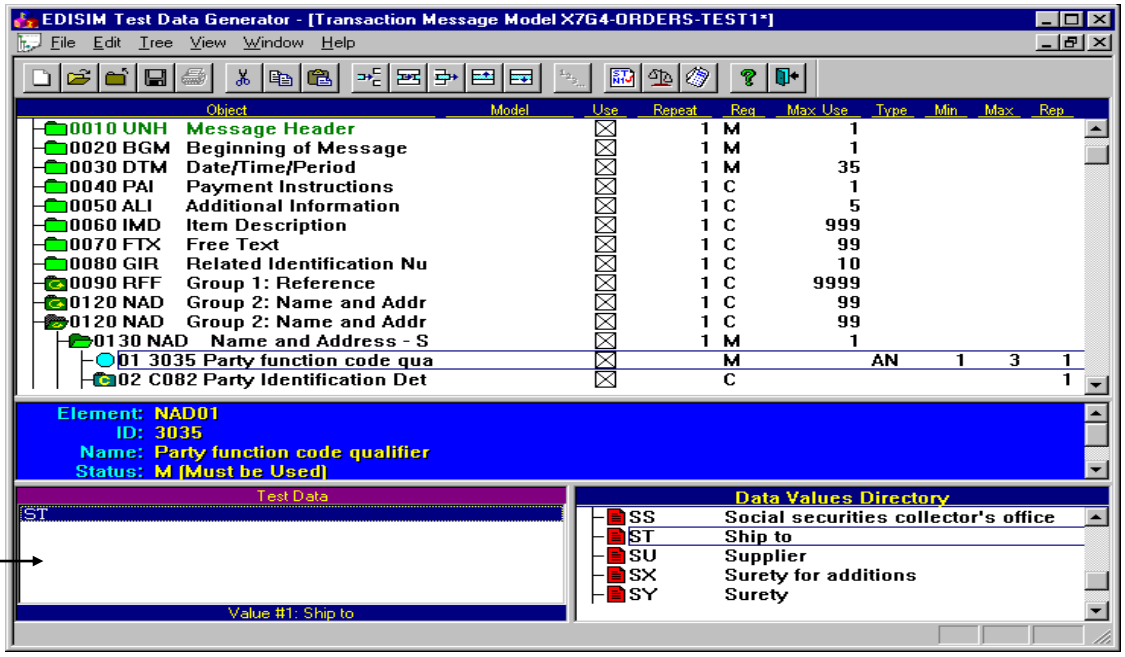
1. Groups are symbolized by dark green folders with a looping arrow. Double-click on the **Group 2 Ship To NAD** group to open it.

   The subordinates of our customized NAD group display below it.

2. Segments are symbolized by bright green folders. Double-click on the **Ship To NAD** segment at **0120** under the group.

3. Elements are symbolized by light blue circles. Single click on element **3035**.

   Data displays in the bottom two panes. For now, look at the Test Data pane; it displays your code set of ST. TDG creates default data that uses values from the code sets you created in Standards Editor.



Test Data pane →

4. Double-click on the **C082** composite at **NAD02**. Click on subelement **3039** and look at our default data.

   Some default values have been filled in by the Test Data Generator since the MIG had no code sets or application values here.

5. Double-click on the **Ship To NAD** segment to close it.
6. Double-click on the **LOC** segment at **0140**, and double-click on the **C517** composite.

7. Click on the **3225** subelement and review the test data values.

   These are application values from Standards Editor.

8. Scroll down to the **CTA** group under the Ship To NAD, and open it. Open the **CTA** segment.
9. Click on element **3139** to see the test data values.

   These are the values from our code set.

STOP

Comments

- The first item in each table displays in green text.
- TDG already has default data, and we can generate test data at any time. Default values come from:
  - EDISIM defaults.
  - values you enter.
  - functions (when appropriate).
  - selected code or application values (where available).
- In the Test Data Generator, you can further customize your data by:
  - reflecting a particular occurrence.
  - inserting an intentional error.
  - generating different quantities.
- At present, TDG does not create default data that matches the requirements of the regular expressions.

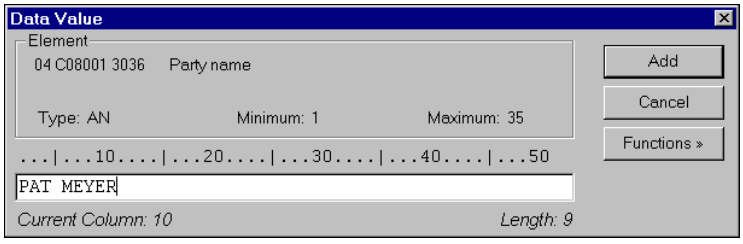
Adding New Values

Although all elements are preloaded with values, you will want to test for certain conditions. This means you will want to change some of the default values provided by the Test Data Generator.

START

1. Double-click to open the **BGM** segment at **0020**.
2. Click on the **4343** element at **BGM04**, and look at the values in the Test Data pane.
3. Double-click on the **Code Values** folder in the Data Values Directory to open it.

   All code values for element 4343 appear.
4. Double-click on code **AG** to add it to the test data list.
5. Close the **BGM** segment.
6. Open the **Ship To NAD** segment.
7. Open the **C080** composite, and click on the **3036** subelement at **C080.01**.
8. Click on the top name in the Test Data pane, and choose **Edit | Insert**.
9. In the Data Value dialog box, type your name in UPPERCASE letters.



10. Press ***Enter***.

    This adds your name to the Test Data pane and clears the Data Value field to add another value.
11. Click **Cancel** to leave the Data Value dialog box.
12. Close the **Ship To NAD** segment.
13. Double-click on the **RFF** group at **0160** in the Ship To NAD group. Double-click on the **RFF** segment.
14. Double-click the **C506** composite at **RFF01**. Click the **1154** subelement.
15. Click on the top value in the **Test Data** pane.

16. Choose **Edit | Insert** and enter **RFF423** in the Data Value dialog box, then click **Add**.

    We've added a value to fit our Application Value requirement (the regular expression that we added in Standards Editor).

17. Click **Cancel**.

18. Close the **Group 2 Ship To NAD** group.

19. Save your work.

STOP

Comments

- The commands on the menus operate according to the pane where your cursor is placed.

Using Functions to Provide Values

Let's set up two functions. The first function creates today's date every time we generate test data from this model, and the second function generates a random number.

START

1. Double-click on the **DTM** segment at **0030**, and double-click on the **C507** composite.
2. Click on the **2380** sub-element.
3. Highlight the default value in the Test Data pane and press ***Delete***.
4. Open the **Functions** folder in the Data Values Directory pane.
5. Double-click on the **{DATE}** function to add it to the Test Data pane.
6. Close the **DTM** segment.
7. Open the **Group 6 TAX** group at **0250**, and open the **MOA** segment at **0270**.
8. Open the **C516** composite, and click the **5004** subelement.
9. Highlight the default values in the Test Data pane and press ***Delete***.
10. Open the **Functions** folder and scroll down to **{RAND(min,max)}**. Double-click to add it to the Test Data pane.
11. Click on the **{RAND(min,max)}** value in the Test Data pane and choose **Edit | Replace** to edit our value.
12. In the Data Value dialog box, replace **(min,max)** with **(1,10000)** so the function looks like {RAND(1,10000)}. Click **OK**.

    The changed function displays in the Test Data pane. This function will generate a random number between 1 and 10,000.

    Note: If a function contains lowercase text, this means that you should replace it with information that you need.
13. Close the **Group 6 TAX** group.

STOP

Marking Composite Sub-elements as Unused

START

1. Open the **Group 28 LIN** group at **1030**, and open the **IMD** segment at **1060**.

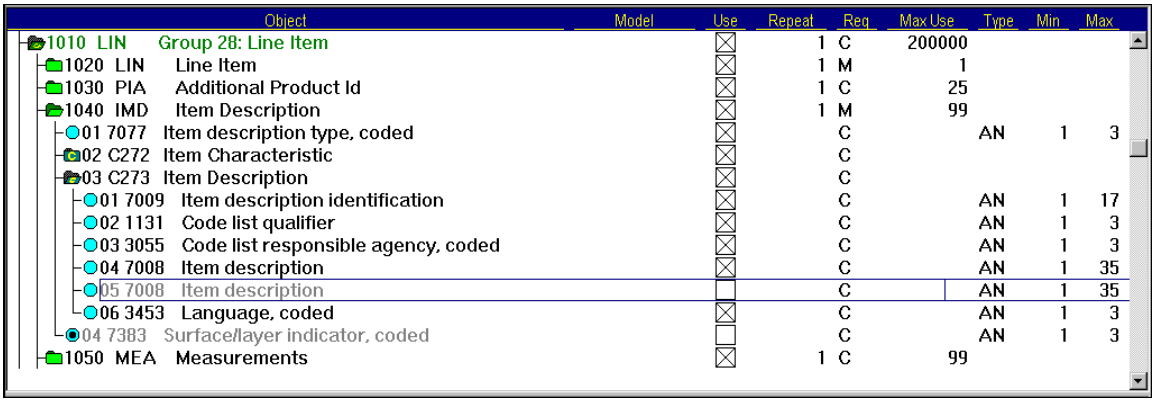
   Notice that the text for element 7383 is faded and the box in the Use column is empty. Test Data Generator knows that we marked this as unused in Standards Editor, so it is unused here also.

2. Double-click on the **C273** composite at **IMD03**.

3. Scroll down and select the **7008** sub-element at **IMD03.05**.

4. Click in the **Use** column for this sub-element.

   The X disappears in the Use column, indicating that Test Data Generator will not include this sub-element in the data.



Object	Model	Use	Repeat	Req	Max Use	Type	Min	Max
1010 LIN Group 28: Line Item		☒	1	C	200000			
1020 LIN Line Item		☒	1	M	1			
1030 PIA Additional Product Id		☒	1	C	25			
1040 IMD Item Description		☒	1	M	99			
01 7077 Item description type, coded		☒		C		AN	1	3
02 C272 Item Characteristic		☒		C				
03 C273 Item Description		☒		C				
01 7009 Item description identification		☒		C		AN	1	17
02 1131 Code list qualifier		☒		C		AN	1	3
03 3055 Code list responsible agency, coded		☒		C		AN	1	3
04 7008 Item description		☒		C		AN	1	35
05 7008 Item description		☐		C		AN	1	35
06 3453 Language, coded		☒		C		AN	1	3
04 7383 Surface/layer indicator, coded		☐		C		AN	1	3
1050 MEA Measurements		☒	1	C	99			

5. Close the **IMD** segment.

6. Save your work.

STOP

Comment

- IMD03.05 will not be generated. Although IMD03.05 was not marked unused in the MIG, it is not required nor mandated either, so we have not created an error.

Marking Items Not Used

START

1. In the **LIN** group, click the **Use** column to turn off generation of segment **PIA** at **1050**.

   Turning off an optional element or segment is allowed.

2. Scroll down to the **ALI** segment at **1100** and turn off the **Use** column.
3. Scroll down to the **GIR** segment at **1150** and turn off the **Use** column.
4. Close the **LIN** group.
5. Save your work.

STOP

Comments

- TDG allows segments that are not mandatory to be turned off, but a warning will appear if you choose to turn off mandatory segments.

Changing the Repeat for Segments and Groups

So far, our test data file will have one of each segment and group. We can change the quantity on the screen.

START

1. Click on the **Repeat** column for the **RFF** group at **0090**.
2. Type **2**, and press ***Enter***.

   The data will now include two RFF groups at this location.

STOP

Creating an External Model (Optional Exercise)

You can customize and give a name to a group, segment, or element. You can then reuse this “named model” elsewhere. This produces consistency and saves work.

START

1. Open the **Ship To NAD** segment, and the **C080** composite. Click sub-element **3036** at **NAD04.01**.

   The test data includes our name that we added. Let’s make this a model to use again.

2. Choose **File | Save Externally (Name)**.

   The Save As External Element Model dialog box appears.

3. Enter **3036NAME** for the model name and replace the text for the description with **3036 Name List**, then click **OK**.

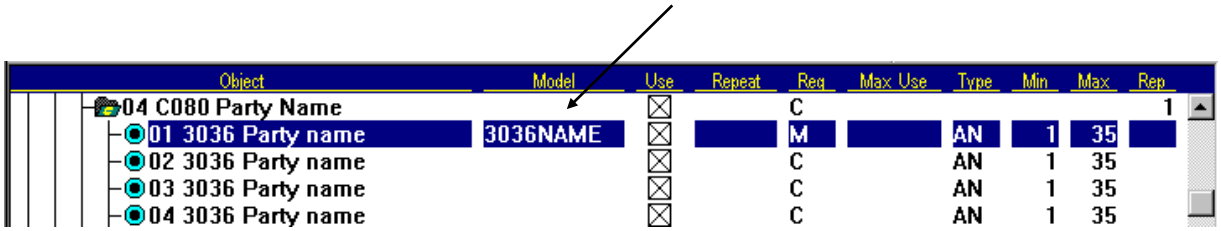
   We have created a model of sub-element 3036.

4. With sub-element **3036** still selected, choose **Edit | Replace**.

5. From the Select Item dialog box, select **3036NAME**, and press ***Enter***.

   The model replaces the original sub-element.

Test Data Generator shows the name of the external model here. The entire model has a shaded background.



6. Save your work.

STOP

Comments

- We saved a specific definition of sub-element 3036. This model will be available to other 3036 sub-elements in any message/set model.

- Reasons to create external models:
  - Saves effort
  - Assures consistency
  - Improves database efficiency
- To change a named model, double-click on the model's name. You cannot change it directly on the screen.
- You cannot change the repeat count of an external model of a group or segment. You must edit the model to do this.

Copying Values from an Outside Source (Optional Exercise)

In this exercise, we copy company names from a word processor file. Since we may want to use this list elsewhere, we save an external model of the element that uses the company names.

START

1. In the **Payer NAD** segment, double-click on composite **C080**, and click sub-element **3036** at **NAD04.02**.

   Look at the Test Data. We want to add our list of Payer customers.

2. Let's go to Explorer and use a word processor file that contains customer names. Right click on the Windows **Start** button, and choose **Explore**.

3. In the Explorer window, double-click on **EDISIM \Samples\Customer.rtf**.

   Our word processor opens the customer list.

4. Select all of the names in the Customer Name list, and choose **Edit | Copy**.

5. Close the word processor and Explorer.

6. Click in the Test Data pane in Test Data Generator, and choose **Edit | Paste**.

   Our list of names is pasted into our Test Data pane.

7. Click on sub-element **3036** at **NAD04.02** again, and choose **File | Save Externally (Name)**.

   This will create another named model of customer names for sub-element 3036.

8. Type **CUSTNAME** for the model name, and replace the text for the description with **Element 3036 Customer**, then click **OK**.

   We have now created a new external element model consisting of this variation of sub-element 3036.

9. With 3036 still selected, choose **Edit | Replace**. In the Select Item dialog box, select **3036-CUSTNAME**, and click **OK**.

   The Model column shows that we're using a model CUSTNAME here. Click on another item and notice that the model has a shaded background.

STOP

Using and Changing an External Model (Optional Exercise)

If you change an external model, the change affects all places where the model is used.

START

1. In the **Ship To NAD** segment, in the **C080** composite, click on the **3036** sub-element at **NAD04.02**.

2. Choose **Edit | Replace**.

3. In the Select Item dialog box, select **3036-CUSTNAME**, and click **OK**.

   We are re-using the previously created element model.

4. With **3036** still selected, click in the **Test Data** pane, and choose **Edit | Insert**.

   TDG will not allow you to add a new name to the model this way. You must edit the model.

5. Double-click on **CUSTNAME** in the model column.

   The model opens. Notice the title bar.

6. Click the top name in the Test Data pane, and choose **Edit | Insert**. In the Data Value dialog box, type **JOHN SMITH**. Click **Add**, and click **Cancel** to close the Data Value dialog box.

7. Choose **File | Close** to close and save changes to the element model.

   The model has been updated with your changes.

8. In the **Payer NAD** segment, click on the **3036** sub-element at **NAD04.02**. It has also been updated with JOHN SMITH.

STOP

Comments

- You can also make external models of segments or groups.

Detaching a Model

TDG also allows you to detach a model from an element.

START

1. With the **3036** sub-element selected, choose **File | Save Locally (Embed)**.

   This severs the element from the model. The shading is gone. From now on, any changes made to the model will not change the Payer NAD sub-element 3036 and vice versa.

2. Click the top name in the Test Data pane, and choose **Edit | Insert**.

3. In the Data Value dialog box, add **ABE LINCOLN**, then close the box.

4. Scroll to the **Ship To NAD**, and click on **3036** at **NAD04.02** again.

   ABE LINCOLN was not added to our list.

5. Choose **File | Close**. Answer **Yes** to the question about saving.

STOP

Setting Up Enveloping

So far, we have customized a message from the UNH to the UNT. We now need to customize the enveloping.

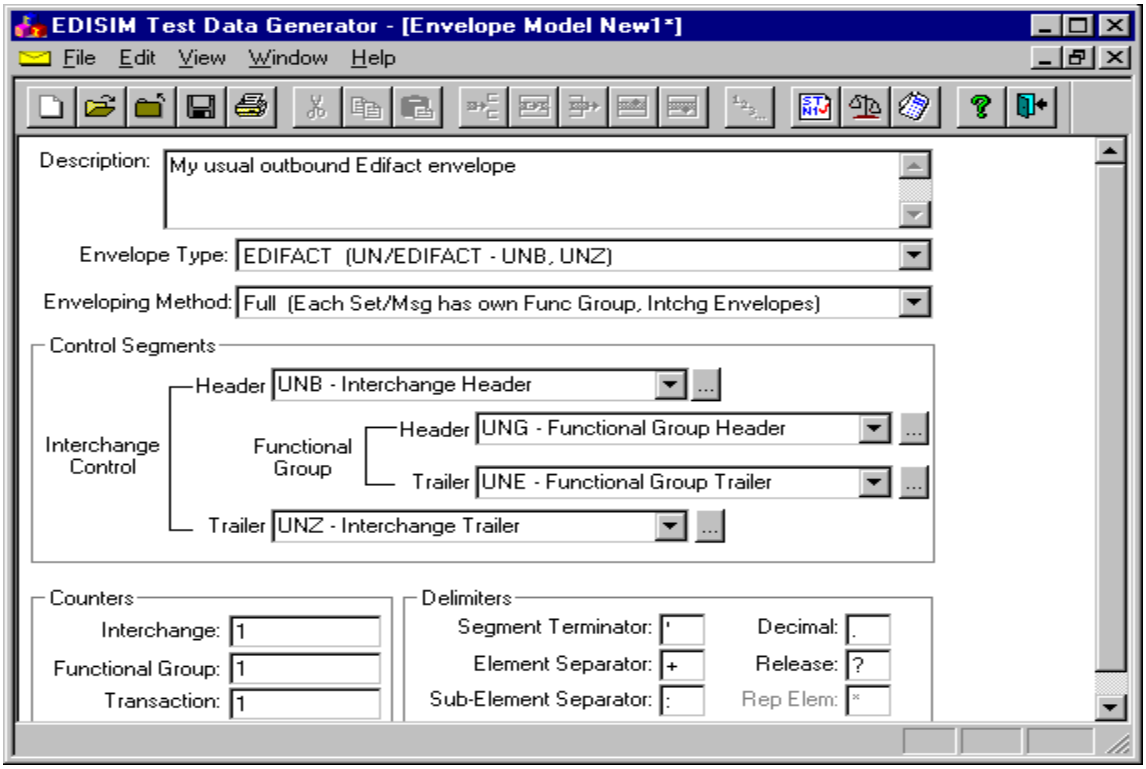
START

1. Choose **File | New | New Enveloping Model.**

   In this screen, you can define or change enveloping. These fields are explained in Customizing Your Enveloping Model in Help.

2. For the Description, type **My Usual Outbound EDIFACT Enveloping**.

3. For Envelope Type, choose **EDIFACT (UN/EDIFACT-UNB, UNZ)**.



4. Choose **File | Save As**, type **EDIFOUT** for the name, and choose **OK**.

   You now have your own copy of EDIFACT enveloping, and can use it over and over. You probably will not need to change it very often.

5. Click the arrow for the **Enveloping Method** field, and look at your choices. Leave it at Full.

6. Click the **Ellipsis** button (three dots) at the end of the **Interchange Control Header** field (UNB line).

   You will see the UNB elements and can edit them just as you did in the Message/Set Model.

7. Double-click on the **S002** composite to view its elements.

8. Delete the value for **0004 Sender identification**, insert our Duns+4 number **7825012250001**, and press the ***Spacebar*** twice to add two spaces, so that we have 15 characters.

9. Select sub-element **0007**. Select the code in the Test Data pane, choose **Edit | Replace**, and change it to **9**.

10. Save the UNB.

11. Click **File | Close**.

12. Save the Envelope.

13. Click the **arrow** for the **Functional Group Header** field (UNG line).

    You will see other pre-defined UNGs and other enveloping segments that you can choose. Leave it at the UNG - Functional Group Header.

14. Choose **File | Close**.

STOP

Comments

- You probably will not have to change your enveloping models very often.
- You can change the Delimiters.
- TDG comes with sample enveloping models. You can see them by choosing **File | Open** and selecting Enveloping Models in the Object Types field.

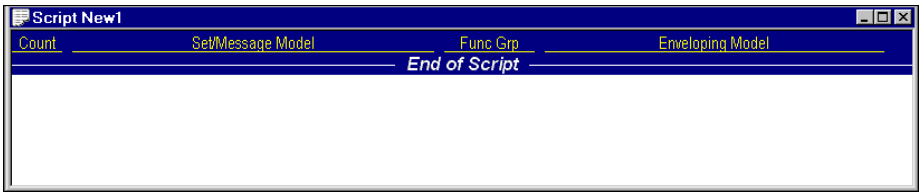
Creating a Script

Let's create a script that consists of our customized message model and enveloping model.

START

1. Choose **File | New | New Script**.

   The Script dialog box appears with no scripts listed.



2. Choose **Edit | Insert**.

   The Script Line dialog box appears. We will insert a line into the script.

3. Click the **Ellipsis** button after the **Message/Set Model** field.

   A list of all message/set models appears.

4. Select **TEST1**, and click the **Select** button.

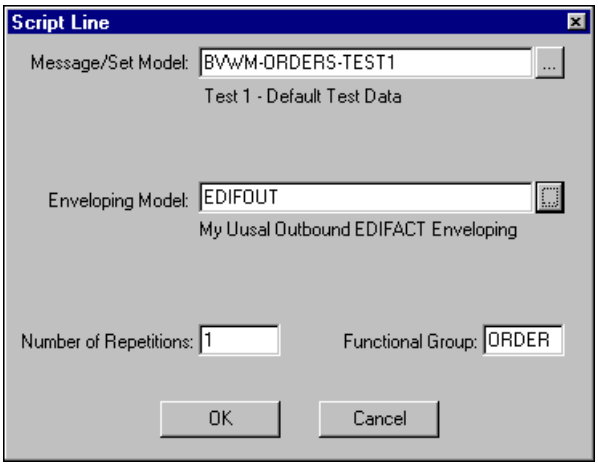
5. Click the **Ellipsis** button after the **Enveloping Model** field.

   A list of enveloping models appears.

   Note: The "$EDIFACT" enveloping model is generic EDIFACT enveloping provided by TIBCO Foresight.

6. Select **EDIFOUT**. Click **Select**.

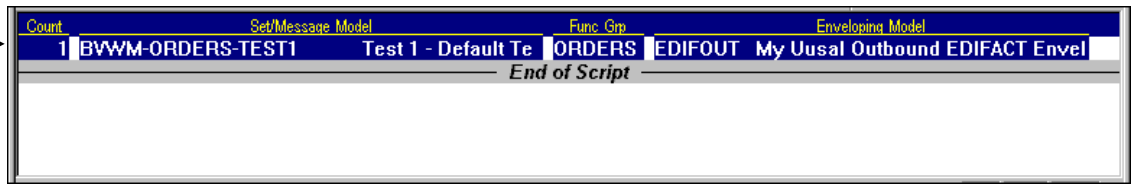
   This is our EDIFACT enveloping. The Script Line dialog box now appears like the following example.



7. Click **OK**.

   We return to the Script dialog box, which now lists one line.

New script line →



8. Choose **File | Save As** and name the script **TEST1**. For the description, type **Basic functional test of ORDERS Purchase Order Message**. Click **OK**.

STOP

Comment

- Each script makes one test data file, regardless of how many messages it includes.

Generating Test Data

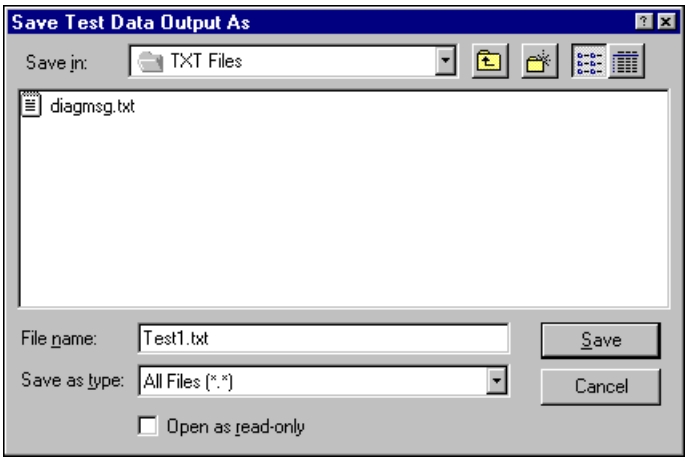
Now that we created the script using our customized message model and enveloping model, we're ready to generate the test data.

START

1. Choose **File | Generate Test Data**.

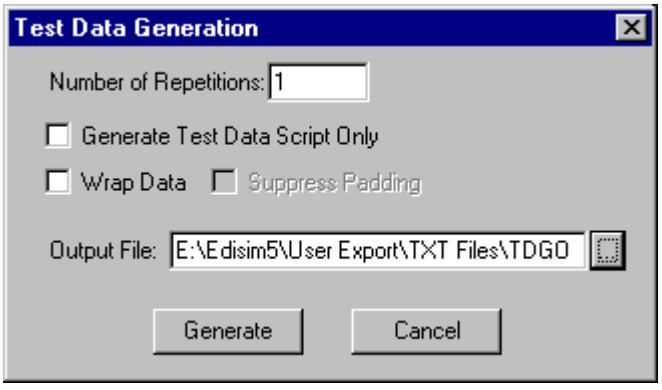
   The Test Data Generation dialog box appears.

2. Click the Ellipsis button for the Output File field and open the **TXT Files** folder under **EDISIM \User Files**. Name the file **Test1.txt**.



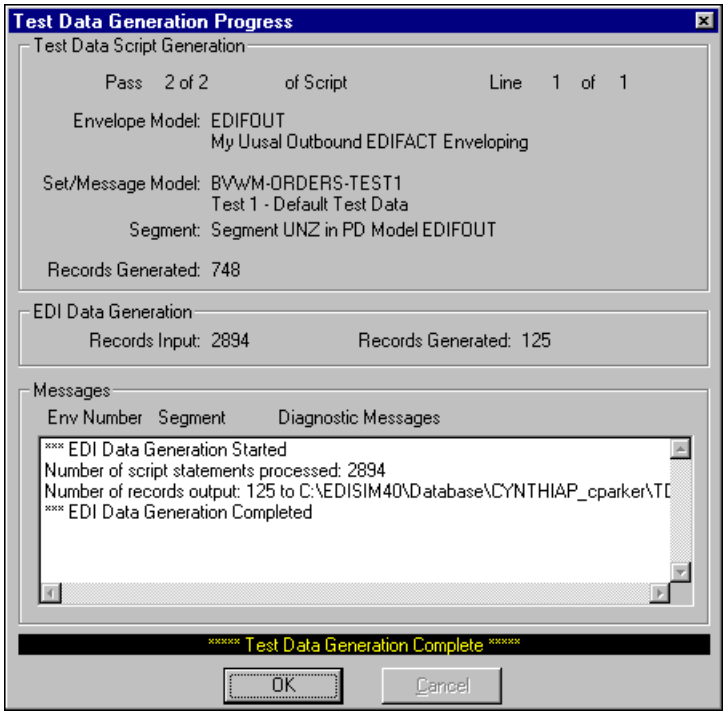
3. Click the **Save** button.

   The Output File field now displays where your file will be saved.



4. Click the **Generate** button.

   Test Data Generator creates the test data. Notice the information in the Messages area. Errors would be reported here. Some errors listed are not data file errors; they are instruction errors to TDG such as a missing delimiter in a function.



5. Click **OK**, and then close the **Script** dialog box.
6. Look at your data file, **TEST1.txt** in the EDISIM \Database\TXT Files folder.

STOP

Comments

- Test Data Generation dialog box:

  1) Number of Repetitions:

     a. cycle the entire script, including interchange header and trailer.

     b. cycle an individual script line.

     c. cycle several script lines.

  2) Generate Test Data Script Only:

     a. generates only a script (TDG) file, not data. A script file is instructions to create data, or used on a PC, mainframe, or AS/400 to make data.

  3) Wrap Data:

     a. places all of the data on one line, with no CR/LFs.

     b. Suppress Padding option is only available after selecting Wrap Data. This option will remove the padding at the end of the file.

- When TDG runs, it creates the EDI data, which you can:

    1) send through a translator.

    2) send to a trading partner.

    3) print as an example, or copy segments to Standards Editor notes to use as examples.

Modifying an Existing Message Set/Model

You can return to a message model to make changes, or save it as another file and then make changes. We're going to save our customized message set model as a new model, then enter an invalid code value that will produce an error in the generated data.

START

1. Return to TDG and choose **File | Open**.

   The Open Model dialog box appears.

2. Make sure the Object Types is set to **Message/Set Models**.

3. Select **TEST1**, and click **Open**.

4. Choose **File | Save As** and name our model **TEST2** with a description of **Test 2 - Invalid Test Data**.

5. Click **OK**.

6. Let's make a change in the data. Open the **Payer NAD** group.

7. Double-click on the **CTA** group, then the CTA segment.

8. Click on **3139 Contact function, coded**.

9. Double-click on the **Code Values** folder in the Data Values Directory at the bottom right.

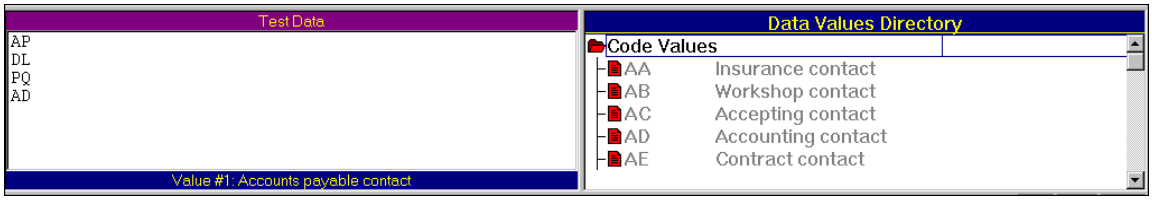
   Notice the local code PQ.

10. If **View | Show Unused** is off, turn it on.

    All codes in the parent standard appear, with the unused ones faded.

11. Double-click on **AD**, an unused code.

    It is now added to the bottom of the Test Data list. The first three values in the list are the code set we set up in Standards Editor. We added the AD code to force an error.



12. We need to rearrange the list so TDG will use AD first. Select **AD** and click the **Move Row Up** button on the toolbar 3 times until AD is at the top of the list.

13. Choose **View | Show Unused** again to turn off display of the unused codes.

14. Save your work.

STOP

Marking Items Not Used and Inserting New Items

Let's continue to add invalid data to our new model. We will delete a mandatory element and insert a segment, which was marked not used in our guideline.

START

1. Open the **FTX** segment at **0070**. Select the mandatory element **4451** at **FTX-01**, and click on the **Use** column to turn it off.

2. Click **Yes** when asked if you are sure you don't want to generate it.

   We are not using this element and it becomes faded. We are intentionally including an error by not including a mandatory element.

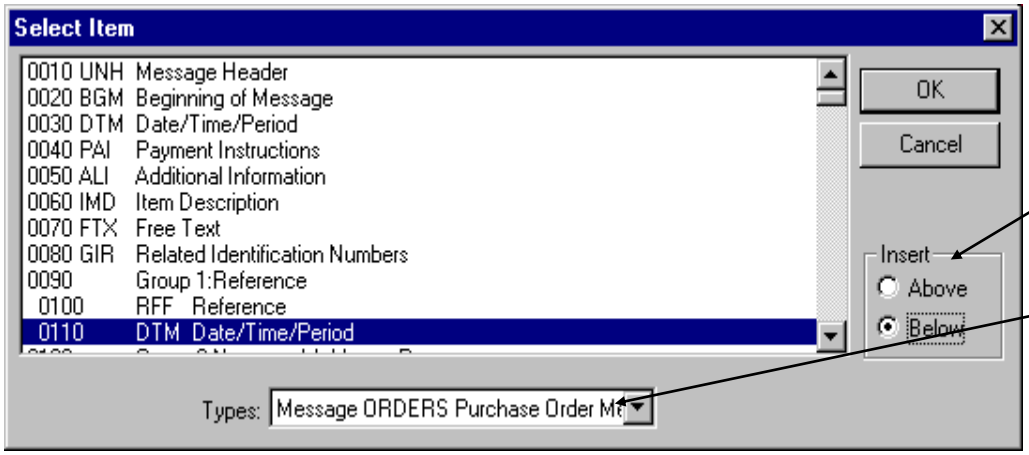
3. Close the **FTX** segment.

4. Click on the **RFF** group at **0090** (DO NOT DISPLAY ITS ELEMENTS).

5. Choose **Edit | Insert**.

   The Select Item dialog box appears.

6. Select the **DTM** segment at **0110**. Click the **Below** button.



7. Click **OK**.

   We have inserted the DTM segment below the RFF segment. This is the segment that we previously marked "not used" in Standards Editor.

8. Save and close Test2.

STOP

Editing an Existing Script

We now want to add the invalid message to our script. This will give us one EDI file containing multiple messages.

START

1. Choose **File | Open**.

   The Open Model dialog box appears.

2. Select **Scripts** for the Object Type.

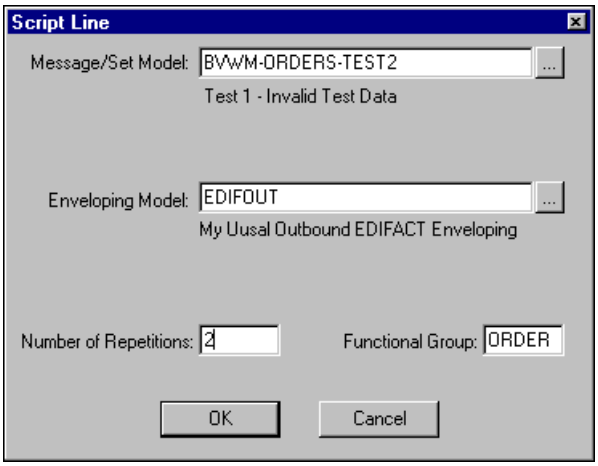
3. Select **TEST1**, and click **Open**.

   Our TEST1 script opens.

4. Choose **Edit | Insert**.

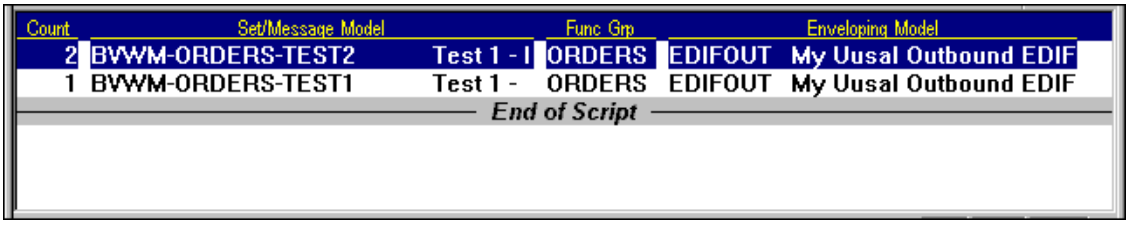
   We are inserting a line into the script.

5. Select **TEST2** from our list of Message/Set Models. Select the Enveloping Model **EDIFOUT**. Change the Number of Repetitions to **2**.



6. Click **OK**.

   We have added the Test2 script to our list. The TEST2 message will appear twice (UNB-UNZ) in the test data file.



7. Choose **File | Save As**, and name the file **NEWORDS**. Replace the description text with **Default and invalid test data with 2 repetitions of invalid test data**.

8. Click **OK**.

9. Choose **File | Generate Test Data**.

10. Select the **EDISIM \User Files\TXT Files** folder, and save the file name as **NewOrd.txt**.

11. Click **Generate**.

12. Click **OK** to close the Test Data Generation Progress dialog box.

13. Close the script.

STOP

Looking at the Test Data

View data in file located at **EDISIM \User Files\TXT Files\Neword.txt**. The following is a condensed test example–some of the data has been removed.

Beginning Enveloping

```
UNB+UNOA:1+78-250-12250001:9:REVERSE ROUT+RECIPIENT:12:ROUTING
ADDR+990630:1516+2+RECEIVER PSWD:AA+APPL REF+A+1+COMM AGREE ID+1'
UNG+ORDERS+APPL SENDER ID:12+APPL RECEIVER
ID:12+990630:1516+2+UN+D:01C:+APPL PSWD'
```

Message with bad data

```
.
UNH+ORDERS2+ORDERS:D:01A:UN:+COMM ACCESS REF+1:F'
BGM+220:12:5:INVOICE+DOC MSG NUMBER:VERSION:RVSN N+2+AC'
DTM+2:990630:102'
.
FTX++1+ABC:16:11+FREE TEXT FIELD:MESSAGE GOES HERE:SENDER MESSAGE:FREE
TEXT FIELD:MESSAGE GOES HERE+ENG'
.
DTM+89:910214:101'
NAD+PR+ACCOUNTS PAYABLE:58:15+BETTY BROWN, SUITE K:PO BOX 543, HOUSTON TX
75321:DAVID PAYNE, M/S 2029:BETTY BROWN, SUITE K:PO BOX 543, HOUSTON TX
75321+KATHY SMITH:ABE LINCOLN:BILL JONES:MARY CARR:KATHY SMITH:1+1234 MAIN
STREET:PO BOX 567:410 N. HIGH STREET:1234 MAIN
STREET+COLUMBUS+ONTARIO+43210+US'
.
CTA+AD+DPRTMNT OR E:PURCHASING'
.
NAD+ST+PURCHASING:58:15+DAVID PAYNE, M/S 2029:BETTY BROWN, SUITE K:PO BOX
543, HOUSTON TX 75321:DAVID PAYNE, M/S 2029:BETTY BROWN, SUITE K+PAT
MEYER:JOHN SMITH:BILL JONES:MARY CARR:KATHY SMITH:1+PO BOX 567:410 N. HIGH
STREET:1234 MAIN STREET:PO BOX 567+SPRINGFIELD+NEW YORK+43756+CA'
.
RFF+AAC:RFF423:2001:REFERENCE VERSION NUMBER'
.
MOA+5:8093:ARP:3:7'
.
UNT+120+ORDERS2'
```

Ending Enveloping

```
UNE+1+2'
UNZ+1+2'
```

Beginning Enveloping

```
UNB+UNOA:1+78-250-12250001:9:REVERSE ROUT+RECIPIENT:12:ROUTING
ADDR+990630:1516+3+RECEIVER PSWD:AA+APPL REF+A+1+COMM AGREE ID+1'
UNG+ORDERS+APPL SENDER ID:12+APPL RECEIVER
ID:12+990630:1516+3+UN+D:01C:+APPL PSWD'
```

Message with bad data

```
UNH+ORDERS3+ORDERS:D:01C:UN:+COMM ACCESS REF+1:F'
BGM+240:12:15:PURCHASE ORDER+DOC MSG NUMBER:VERSION:RVSN N+13+AD'
DTM+89:990630:102'
.
FTX++2+123:16:5+SENDER MESSAGE:FREE TEXT FIELD:MESSAGE GOES HERE:SENDER
MESSAGE:FREE TEXT FIELD+ENG'
.
DTM+115:910214:101'
NAD+PR+ACCOUNTS PAYABLE:100:11+BETTY BROWN, SUITE K:PO BOX 543, HOUSTON TX
75321:DAVID PAYNE, M/S 2029:BETTY BROWN, SUITE K:PO BOX 543, HOUSTON TX
75321+KATHY SMITH:JOHN SMITH:BILL JONES:MARY CARR:KATHY SMITH:1+1234 MAIN
STREET:PO BOX 567:410 N. HIGH STREET:1234 MAIN
STREET+COLUMBUS+ONTARIO+43210+US'
.
CTA+AP+DPRTMNT OR E:MARY WILLIAMS'
.
NAD+ST+PURCHASING:100:11+DAVID PAYNE, M/S 2029:BETTY BROWN, SUITE K:PO BOX
543, HOUSTON TX 75321:DAVID PAYNE, M/S 2029:BETTY BROWN, SUITE K+KATHY
SMITH:Ally Corporation:BILL JONES:MARY CARR:KATHY SMITH:1+PO BOX 567:410
N. HIGH STREET:1234 MAIN STREET:PO BOX 567+SPRINGFIELD+NEW YORK+43756+CA'
```

```
.
MOA+5:6844:USD:7:17'
.
UNT+120+ORDERS3'
UNE+1+3'
UNZ+1+3'
UNB+UNOA:1+78-250-12250001:9:REVERSE ROUT+RECIPIENT:12:ROUTING
ADDR+990630:1516+4+RECEIVER PSWD:AA+APPL REF+A+1+COMM AGREE ID+1'
UNG+ORDERS+APPL SENDER ID:12+APPL RECEIVER
ID:12+990630:1516+4+UN+D:01C:+APPL PSWD'
UNH+ORDERS4+ORDERS:D:01C:UN:+COMM ACCESS REF+1:F'
BGM+320:12:11:DOCUMENT NAME+DOC MSG NUMBER:VERSION:RVSN N+18+AC'
DTM+115:990630:102'
.
RFF+AAB:APJTN5849201194900:K48912:REFERENCE VERSION NUMBER'
NAD+PR+ACCOUNTS PAYABLE:58:5+BETTY BROWN, SUITE K:PO BOX 543, HOUSTON TX
75321:DAVID PAYNE, M/S 2029:BETTY BROWN, SUITE K:PO BOX 543, HOUSTON TX
75321+KATHY SMITH:ABE LINCOLN:BILL JONES:MARY CARR:KATHY SMITH:1+1234 MAIN
STREET:PO BOX 567:410 N. HIGH STREET:1234 MAIN
STREET+COLUMBUS+ONTARIO+43210+US'
.
NAD+ST+PURCHASING:58:5+DAVID PAYNE, M/S 2029:BETTY BROWN, SUITE K:PO BOX
543, HOUSTON TX 75321:DAVID PAYNE, M/S 2029:BETTY BROWN, SUITE K+BILL
JONES:Backs-R-Us:BILL JONES:MARY CARR:KATHY SMITH:1+PO BOX 567:410 N. HIGH
STREET:1234 MAIN STREET:PO BOX 567+SPRINGFIELD+NEW YORK+43756+CA'
LOC+26+PUBLIC SQUARE:16:11:PLACE LOCATION
1A+PLACE:25:15:TORONTO+ENTRANCE:12:5:REDMOND+REL'
.
RFF+AAC:RFF423:2001:REFERENCE VERSION NUMBER'
.
MOA+5:4778:DEM:6:11'
LIN+1+3+ITEMNUMBER1122334455:CL:16:11+1:1+20+I'
IMD+F+13:25:15+76213BB:12:5:SHORT WAVE RADIO::DAN'
.
PAI+1:44:42:16:11:7'
UNT+119+ORDERS4'
UNE+1+4'
UNZ+1+4'
```

Ending Enveloping

Beginning Enveloping

Message with good data

Ending Enveloping

Using Quick Generate

You can quickly generate test data from any message model that you save in Test Data Generator. This is handy if you want to create a single repetition of test data using only one message model.

START

1. Choose **File | Open** and select **Message/Set Models** as the Object Type.
2. Select **TEST1** and click **Open**.
3. Choose **File | Generate Test Data** or click the **Generate Data** button  on the toolbar.
4. Select your **EDIFOUT** Envelope Model.
5. Click the **Output File** Ellipsis button.
6. Select the **EDISIM \User Files\TXT Files** folder, and name the file **TestQG.txt**.
7. Click **Save**.

   The Output File field now displays where your file will be saved.
8. Click **Generate**.
9. Close the file.
10. Open **TestQG.txt** from Windows Explorer and view the test data.

STOP

Exiting Test Data Generator

START

1. Choose **File | Exit**.

STOP

Comments

- Choose **File | New** to create new message/set models, enveloping models, or scripts.
- Choose **File | Open** to use or change the existing message/set model, enveloping model, or script.

Standards Reference

Purpose

The purpose of this exercise is to view the structure of standards or MIGs on-line. Since Standards Reference lets you read but not change EDISIM standards/MIGs, it can be safely used by anyone who needs to examine them.

Opening a Standard

START

1. Choose **Start | Programs | <*TIBCO_HOME*> | EDISIM | Standards Reference**.

   Standards Reference opens.

2. Maximize the Standards Reference window, if not already maximized.

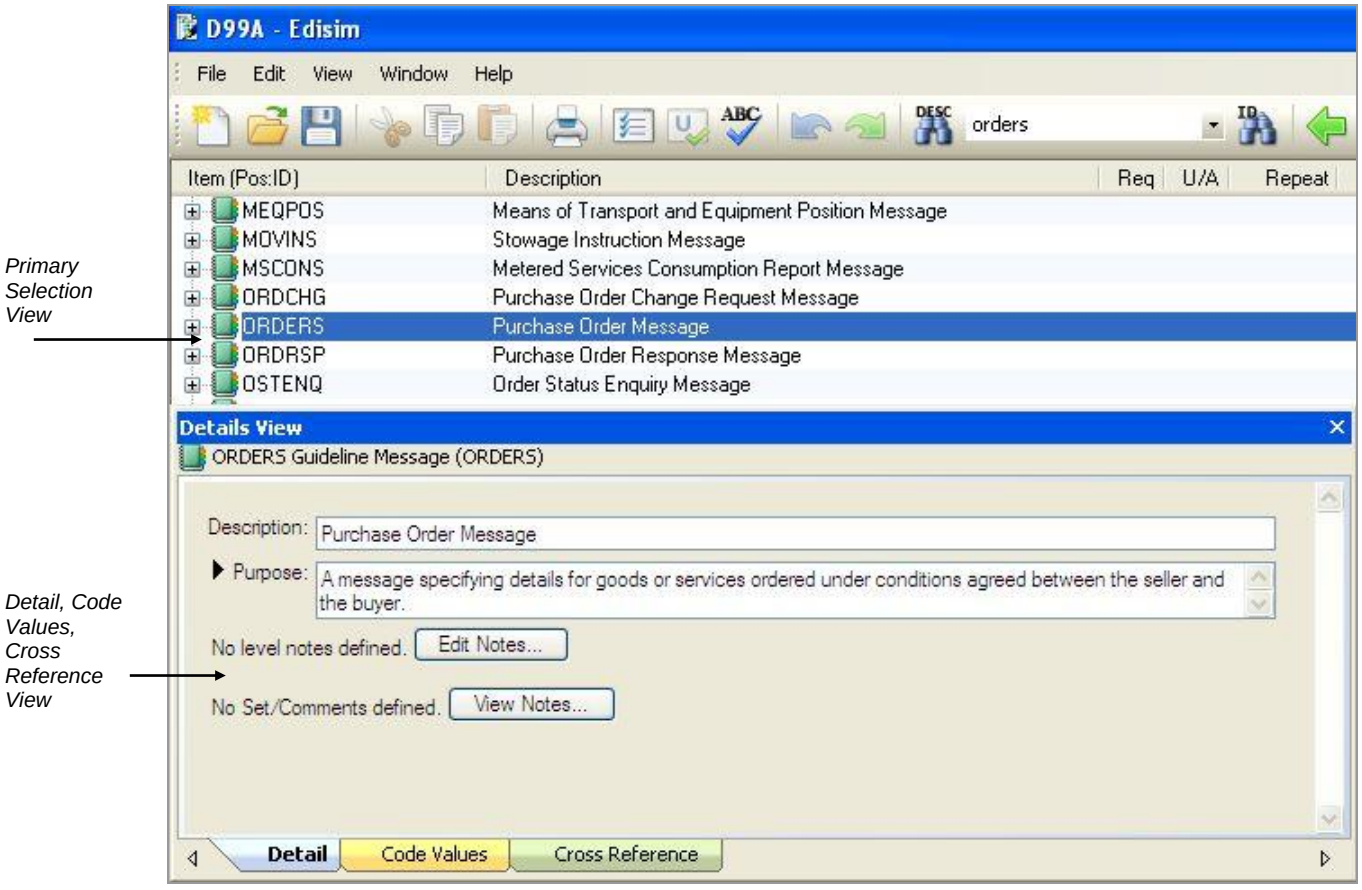
3. Click **File | Open**.

   The Open Standard dialog box appears. It contains all MIGs and TIBCO Foresight-supplied standards.

4. Type **D99A,** hit **Open**.

   All messages in D99A appear.

5. Type **ORDERS** in the speed search box, and hit **Enter**.



STOP

Comments

- Standards Reference lets you examine TIBCO Foresight-supplied standards and also MIGs defined by the Standards Editor user.

- The primary selection pane is the Tree View. The bottom pane displays information about whatever is selected in the Tree View.

Navigating Through Standards Reference

START

1. Expand the **ORDERS Purchase Order Message** in the Tree pane.

   Segments in ORDERS appear in green below it.

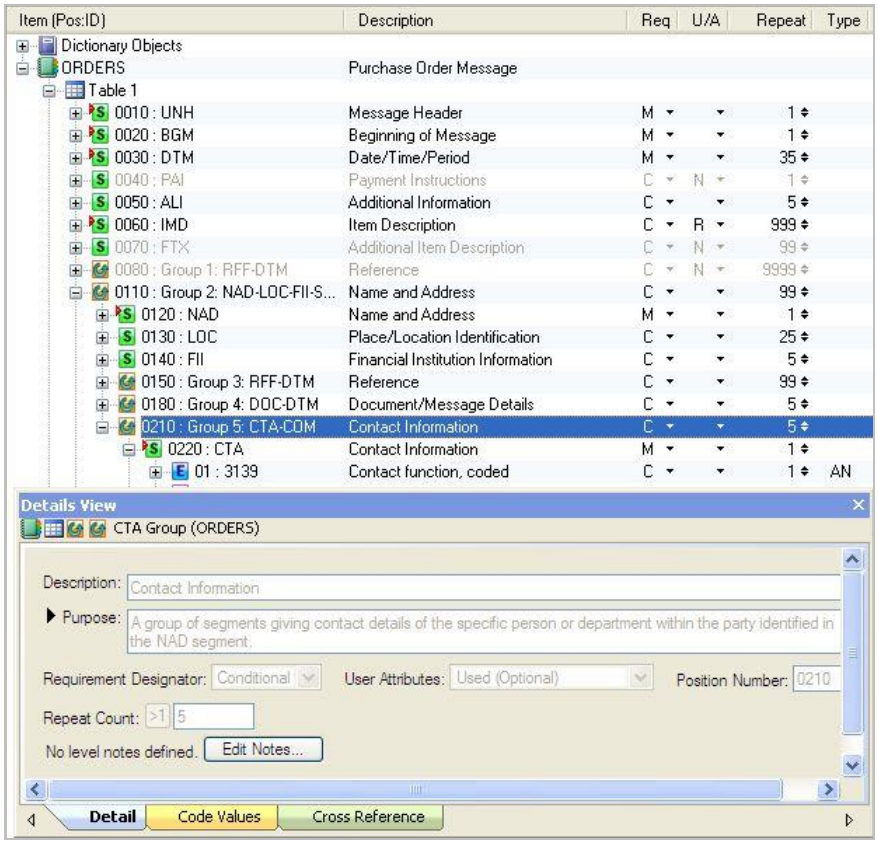
2. Locate and Expand the **Group 2** (NAD group) at **0110**.

   The rest of the screen shows information about the group.

3. Expand the **Group 5** at **0210**.

4. Double-click on the **CTA** segment.

   Its elements appear and the rest of the screen displays information about this contact information group. Your screen should look like the following example.



STOP

Comments

- Clicking on the **+**symbol expands an item and clicking on the **-** symbol collapses the list of subordinates of the selected item in the Tree pane.

- Indicator codes:

SM	Semantic Notes
CM	Comments (X12 Text)
BV	Business Variable
BZ	Business Rule
VR	Application Value List
L1	Level 1 note
L2	Level 2 note
L3	Level 3 note
L+	For all levels of notes above level 3 if defined
CV	Code Value

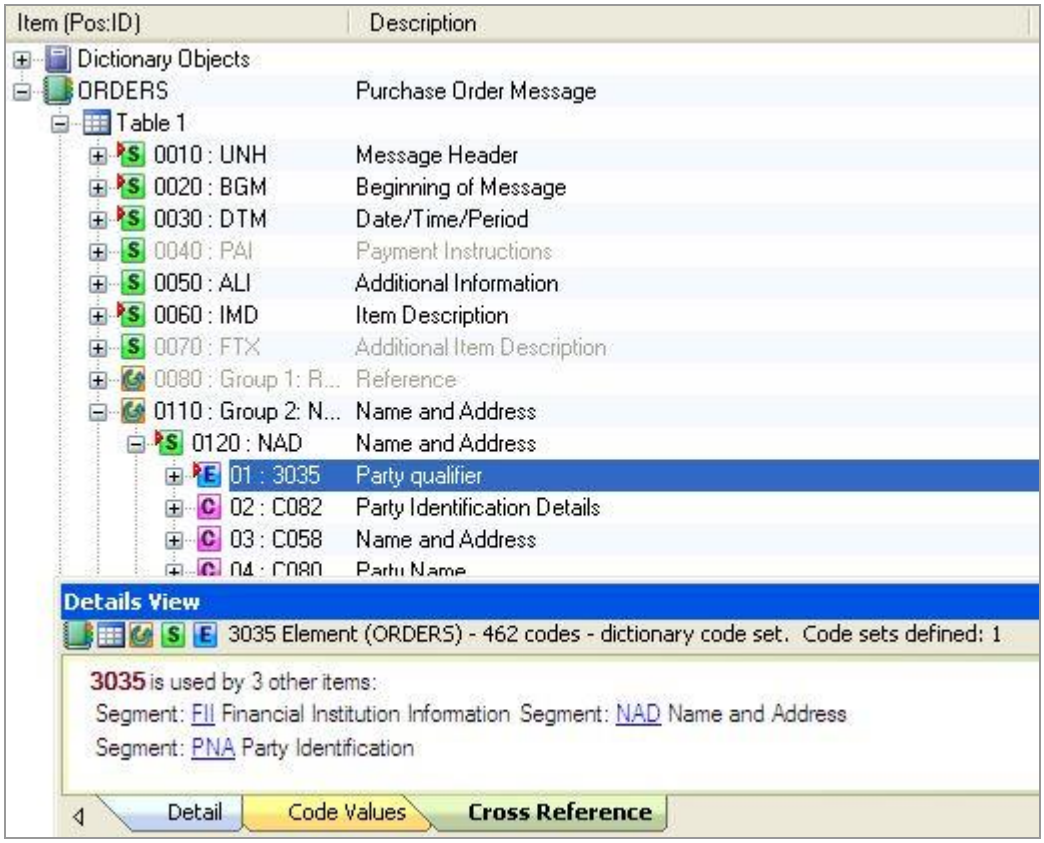
Viewing Cross-References

START

1. In the top pane, expand the **NAD** segment and click the **NAD01** (element **3035**) and click on it.

2. Select the **Cross-Reference** tab at the bottom.

   Element 3035 is used in 3 segments in D99A.



3. Click **File | Close** to close D99A.

STOP

Comments

- A cross-reference answers the question, "Where is this item used?"
- Later, we may explore how to go directly to the cross-referenced item.

Loading Our MIG

START

1. Click the **Open** button (the first one on the toolbar).

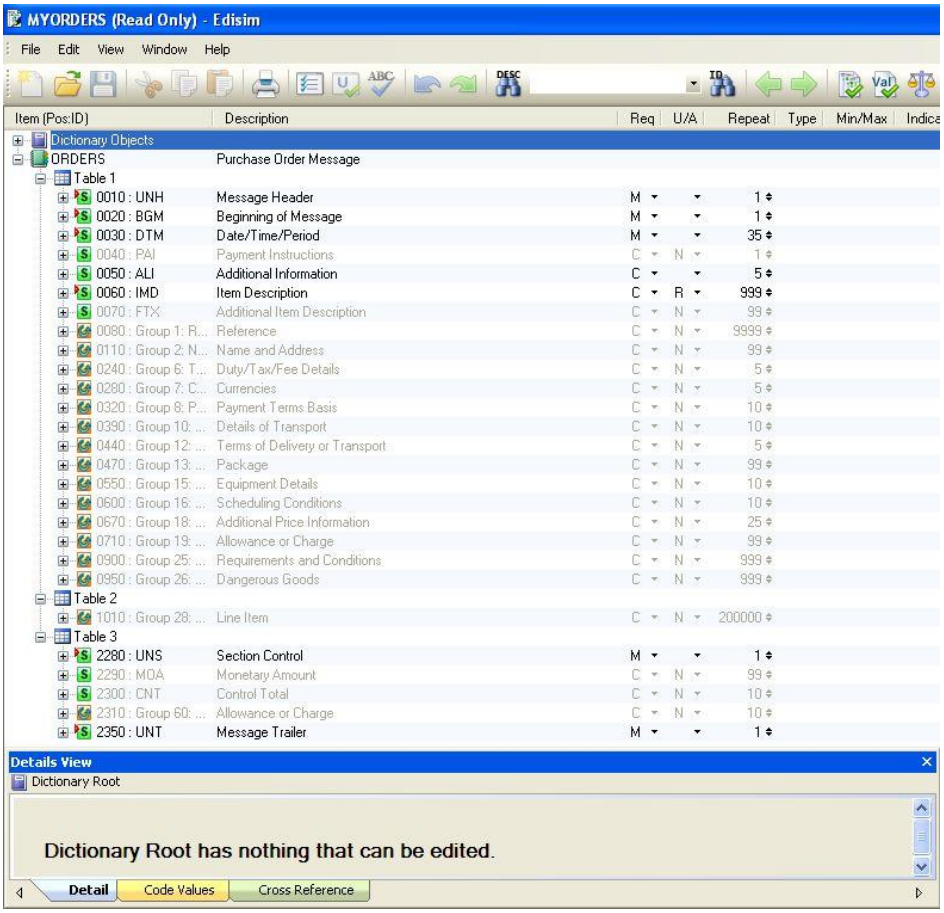
2. Type **MYORDERS**, click **OPEN**.

3. Expand the **ORDERS Purchase Order Message**.

   You will see the segments and groups in MYORDERS.

4. Choose **View | Show Unused Items**. Toggle this setting a few times to see the result, then leave it on.

   You will see all used and unused segments. Unused ones will be faded.



STOP

Comments

- Our MIG should differ from what we saw in the purchase order for the published standard.

Examining Our MIG

START

1. Find **Group 2 (**Payer NAD group) and expand it.

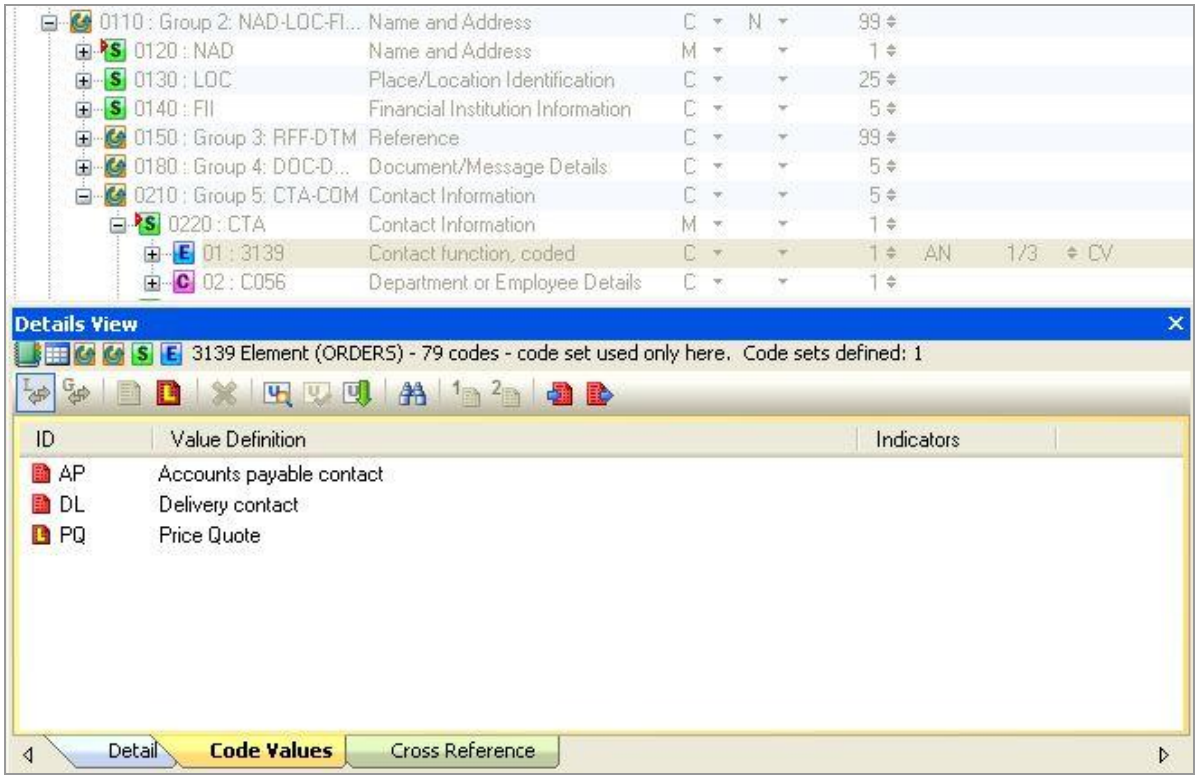
2. Scroll down and expand **Group 5 Contact Information** at position **0210**.

3. Expand the **CTA** segment and click on the **3139** element at **CTA01**. Click on the Code Values tab, if you do not see the unused code values, choose **View | Show Unused Items**.

   The code values in the 3139 appear, with the unused ones faded.

4. Turn off the **Show Unused Items**.

   The unused codes disappear and the screen will look like the following example. Notice the local code icon (the red page with an “L” on it) next to PQ in the Value Definition list.



STOP

Comments

- Code PQ is local because we added it to the message in Standards Editor.
- You can toggle **View | Show Unused** Items to show or conceal unused Tree pane items, or codes in the Code Values tab.

Printing Standards or MIGs

You can print a log of the open standard or MIG. This is handy for searching or for a quick reference. For a formatted document, use Doc Builder.

START

1. Scroll to the top of the Primary Selection pane and click on **ORDERS Purchase Order Message**.

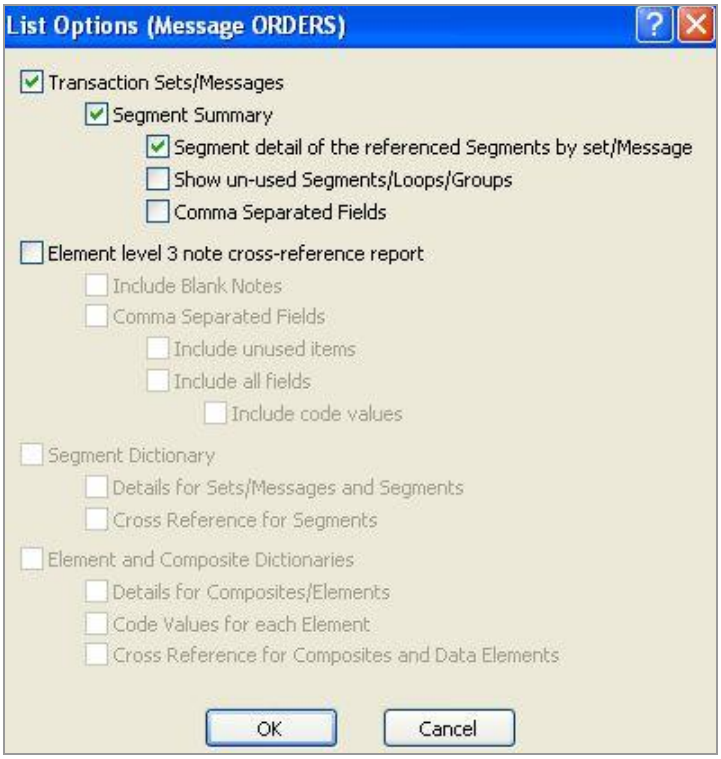
   Note: Selecting a message will produce a log of items within that message.

2. Choose **File | Print**.

   The List Options dialog box appears.

3. Select the check boxes for **Transaction Sets/Messages**, **Segment Summary**, and **Segment detail of the referenced Segments by Set/Message**.

   This will create a log of your MIG with the referenced segments only.



4. Click **OK**.

   The Report of Standard MYORDERS generates.

5. Scroll through the report. Notice the Save As and Print buttons. Click **Close**.

STOP

Comments

- List Options dialog box:

    1) Transaction Sets/Messages

        a. Segment Summary

            (i) Segment detail of the referenced Segments by Set/Message

            (ii) Show un-used Segments/Loops/Groups

    2) Element level 3 note cross-reference report

        a. Include blank notes

        b. Comma separated fields

        c. Include Unused Items

        d. Include all fields

        e. Include code values

    3) Segment Dictionary

        a. Details for Sets/Messages and Segments

        b. Cross Reference for Segments

    4) Element and Composite Dictionaries

        a. Details for Composites/Elements

        b. Code Values for each Element - This is the only way to compile a searchable list of all code values in the standard.

        c. Cross Reference for Composites and Data Elements

- The List Options dialog box enables options according to what you have selected:

    MIG or Standard - Options are enabled for Transaction Sets/Messages, Segment Dictionary, and Element and Composite Dictionaries.

    Transaction Set/Message - Options are enabled for Transaction Sets/Messages and Element level 3 note cross-reference report.

Viewing the Dictionary (Optional Exercise)

START

1. Click on the **+**symbol before the **Dictionary Objects**.
2. Click on the **+**symbol before the **Segments** dictionary.

   You now see the segments in dictionary.
3. Expand the BGM segment.
4. Click on **1225** at **BGM03**.

   Click on the Code Values tab to see the dictionary codes for 1225 in the BGM.
5. Close the **Segment** dictionary, click on the **+**symbol before the **Composite**.

   You now see the composites in the dictionary.
6. Expand the **C002** composite and click on the Cross Reference tab at the bottom.

   It is used in the BMG and DOC segments.
7. Close the **Composite** dictionary, click on the **+**symbol before the **Element** dictionary.

   You now see the elements in the dictionary.
8. Type **3035** and hit **ENTER**.
9. With element **3035** highlighted, look in the cross-reference area.

   The cross-reference list shows which segments contain a 3035 in this standard.
10. Click on the **NAD** segment in the Cross Reference pane.

    You go to the NAD segment in the Segment dictionary.
11. Select the **NAD** segment in the Tree pane.

    The cross-reference area now shows everything that uses NAD in MYORDERS.
12. Click on **ORDERS** in the cross-reference area.

    You now go to NAD in the ORDERS message.
13. Choose **File | Exit**.

STOP

Comments

- You can view two standards or MIGs at once by choosing **Windows | Cascade** (or Tile Horizontal or Tile Vertical).

- You can print the Element Dictionary to a file for future reference, and for easy search capability in Notepad.

- You can also search for words using **Tree | Find**.

TIBCO Documentation and Support Services

How to Access TIBCO Documentation

Documentation for TIBCO products is available on the TIBCO Product Documentation website, mainly in HTML and PDF formats.

The TIBCO Product Documentation website is updated frequently and is more current than any other documentation included with the product. To access the latest documentation, visit https://docs.tibco.com.

Product-Specific Documentation

Documentation for TIBCO® Foresight® EDISIM® is available on the TIBCO Foresight® EDISIM® Documentation page.

The following documents for this product can be found on the TIBCO Documentation site:

- *TIBCO Foresight® EDISIM® Release Notes*
- *TIBCO Foresight® EDISIM® Data Types*
- *TIBCO Foresight® EDISIM® Documentation and Demo Data Index*
- *TIBCO Foresight® EDISIM® Supported File Formats*
- *TIBCO Foresight® EDISIM® Installation Guide*
- *TIBCO Foresight® EDISIM® Introduction to EDISIM®*
- *TIBCO Foresight® EDISIM® DocStarter: Creating a Guideline from EDI Data*
- *TIBCO Foresight® EDISIM® Guideline Merge*
- *TIBCO Foresight® EDISIM® Document Builder User's Guide*
- *TIBCO Foresight® EDISIM® Error Message Numbers, Editing, and Management*
- *TIBCO Foresight® EDISIM® Validator User's Guide*
- *TIBCO Foresight® EDISIM® Using Flat Files*
- *TIBCO Foresight® EDISIM® Library User's Guide*
- *TIBCO Foresight® EDISIM® Validation Profile Files (APF)*
- *TIBCO Foresight® EDISIM® Using XML*
- *TIBCO Foresight® EDISIM® Comparator User's Guide*
- *TIBCO Foresight® EDISIM® Analyzer User's Guide*
- *TIBCO Foresight® EDISIM® Standards and Guidelines Reference Manual*
- *TIBCO Foresight® EDISIM® Test Data Generator User's Guide*
- *TIBCO Foresight® EDISIM® Self-Paced Tutorial: Introduction to EDISIM® (X12 Standards)*

- *TIBCO Foresight® EDISIM® Self-Paced Tutorial: Introduction to EDISIM® EDIFACT D99A Orders*
- *TIBCO Foresight® EDISIM® Standards Editor User's Guide*
- *TIBCO Foresight® EDISIM® Business Rules*

How to Contact TIBCO Support

You can contact TIBCO Support in the following ways:

- For an overview of TIBCO Support, visit http://www.tibco.com/services/support.
- For accessing the Support Knowledge Base and getting personalized content about products you are interested in, visit the TIBCO Support portal at https://support.tibco.com.
- For creating a Support case, you must have a valid maintenance or support contract with TIBCO. You also need a user name and password to log in to https://support.tibco.com. If you do not have a user name, you can request one by clicking Register on the website.

How to Join TIBCO Community

TIBCO Community is the official channel for TIBCO customers, partners, and employee subject matter experts to share and access their collective experience. TIBCO Community offers access to Q&A forums, product wikis, and best practices. It also offers access to extensions, adapters, solution accelerators, and tools that extend and enable customers to gain full value from TIBCO products. In addition, users can submit and vote on feature requests from within the TIBCO Ideas Portal. For a free registration, go to https://community.tibco.com

Legal and Third-Party Notices