TIBCO Foresight® EDISIM®

Self-Paced Tutorial: Introduction to EDISIM® (X12 Standards)

Version 6.20.0

May 2021

Contents

Analyzer 102

Comparator 118

Standards Reference 133

Reference 140

TIBCO Documentation and Support Services 141

Legal and Third-Party Notices 143

Overview of TIBCO Foresight® EDISIM®

The Editor Coordinates Other Tasks

Standards Editor lets you:

- Edit existing EDI standards to create new guidelines that you and your trading partners will use.

- Create new base standards for a group, industry, or committee.

After creating a guideline, you can:

- Use EDISIM® Doc Builder to print formatted documentation from it.

- Use EDISIM Test Data Generator to generate test data from it.

- Use EDISIM Standards Reference to browse through it.

- Use EDISIM Analyzer to check EDI data for adherence to it.

  (Separate TIBCO Foresight validation products: TIBCO Foresight® Instream® for validating during production; TIBCO Foresight® Community Manager® for letting your trading partners validate their own test data).

- Use EDISIM Comparator to compare it to other transaction sets, or to migrate user changes from one guideline to another guideline.
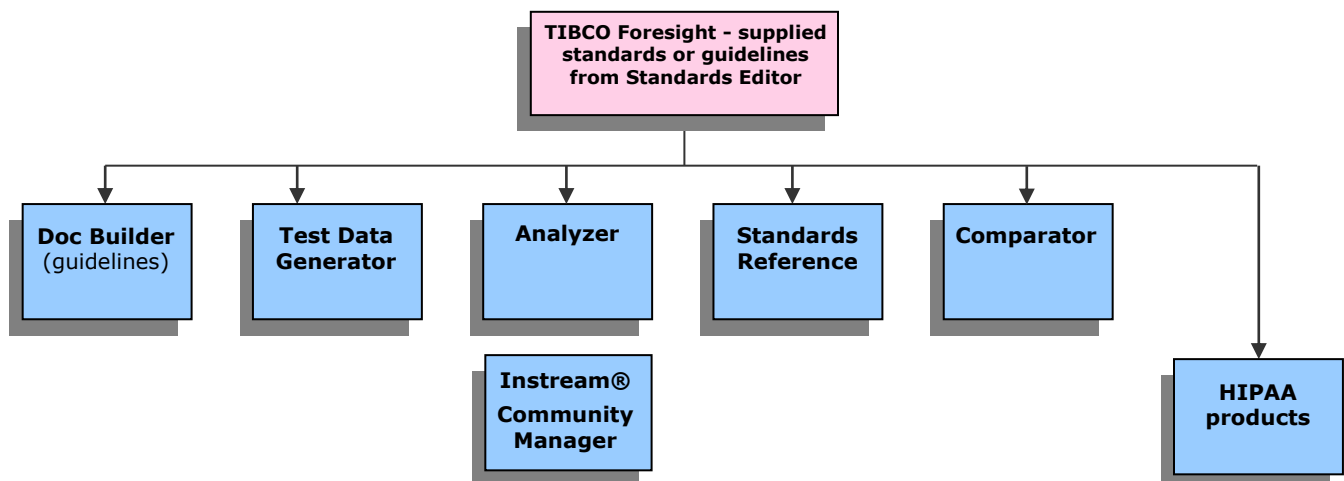
Purchase Order Form

<ST, BEG>

Your Name _<PER02>_   **Phone #** _<PER04>_

SHIP to: Name _<N102 - Ship-to>_

     Department/Institution _<N104>_

     Address _<N2 or N3>_

     City _<N4>_   State _____ Zip Code _____

     Country _____ Phone (____)_____

BILL to: Name _<N102 Bill-to>_

     Department/Institution _<N104>_

     Address _<N2 or N3>_

     City _<N4>_   State _____ Zip Code _____

     Country _____ Phone (____)_____

Items Ordered

Item Number	Item Description	Quantity	Unit Price	Total Price
_<PO107>_	_<PID05>_	_<PO102>_	_<PO104>_	_____
_____	_____	_____	_____	_____
_____	_____	_____	_____	_____

Item Total: Add total price for each item............   Item Total $ _____

Shipping: We ship 1st class mail unless otherwise requested.   _<TD5>_
Shipping by Federal Express available upon request.

Shipping and Handling Charges   Add $5.00 plus 10% of item $ _____
...._____

Total; Add Item Total, Sales Tax, and Shipping and handling Charges   **Total** $ _____

Enclose an Institutional Purchase Order number   or   Account Number _<BEG03>_

Mail to: Kaver Corp. _<ISA, GS>_
                 123 Clarksburg Road
                 Damascus MD 20872

Standards Editor

Customize a Guideline

For more information: See **TIB_fsp_edisim_ _<n.n>_ _fseditor.pdf** in EDISIM's Documentation directory

Purpose

The purpose of this exercise is to customize an inbound 850 Purchase Order for your company. Your guideline will match the exact specifications that your trading partners will use. As you customize the 850, you will select the segments and elements to be used, and will define which specific codes will be acceptable in these elements. Your customized guideline will be used for other exercises throughout this manual.

What standards are guidelines based on?

You can base a guideline on a published standard (X12) or on an industry subset (VICS, UCS, HIPAA, etc.).

Creating a Guideline

Directly From
Published Standard

From
Industry Subset

Guideline

Published
Standard

Guideline

VICS

X12 Standard
Directory
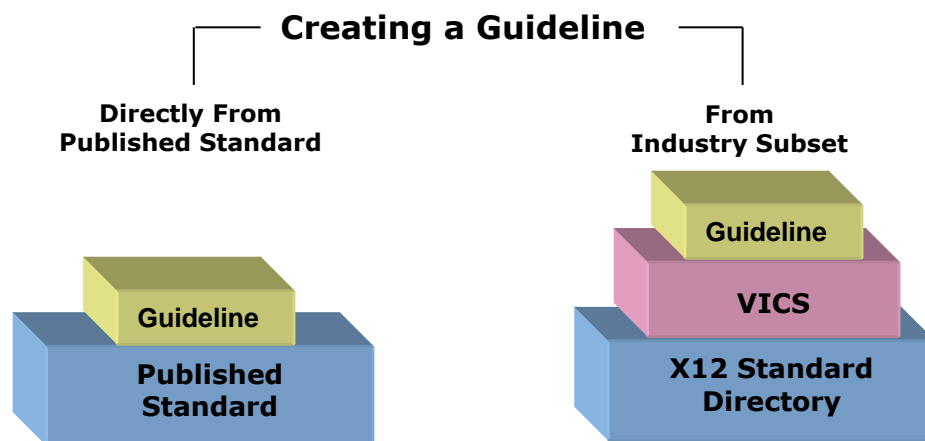
Before Starting Exercises

- Exercises are broken into small units.

- Images are representative but may not always match exactly.

- EDISIM uses standard mouse functions: left button click to select and double-click to process, Shift+click or Ctrl+click to select multiple items.

Flow and Purpose of Course Exercises

- Customize an 850 transaction for specific use (Standards Editor).

- Print a customized guideline for this 850 transaction (Doc Builder).

- Generate 850 test data (Test Data Generator).

- Analyze data to check for compliance (Analyzer).

- Compare two standards or guidelines (Comparator).

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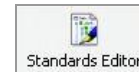
   Otherwise, choose **Start | Programs | <TIBCO_HOME> | EDISIM | Standards Editor**.

   Standards Editor displays a welcome screen.
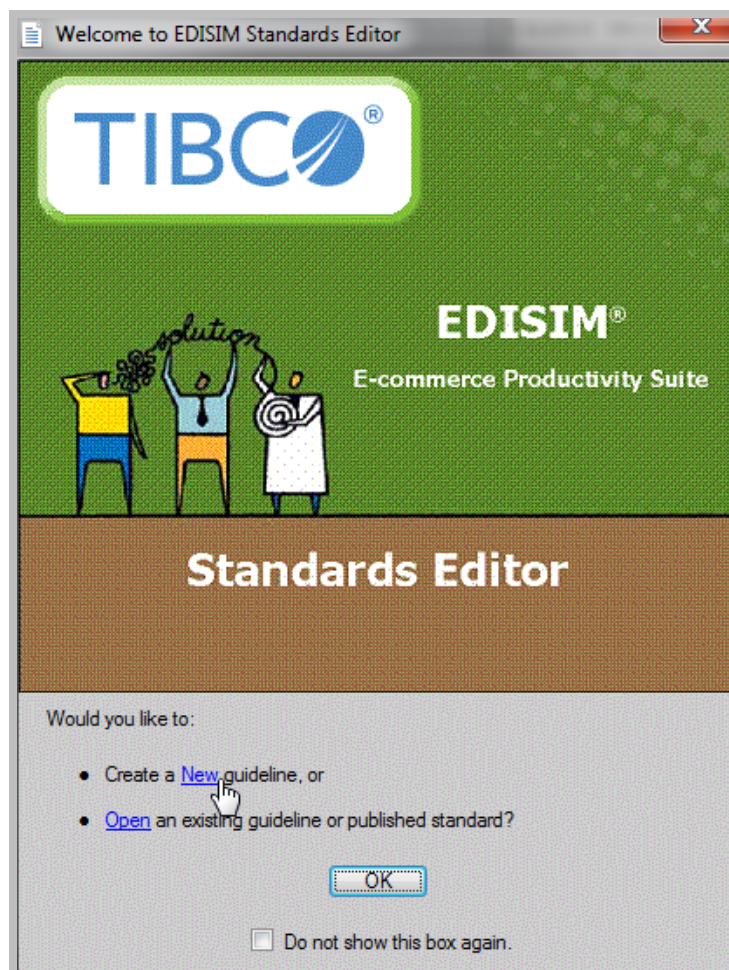
2. Choose the **New** link within the "Create a New guideline" text on the welcome screen.

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

---

3. Select **X12-5010**. (You can type **x** to quickly move to the ones that start with x, or press the *Pg Dn* key.)

4. Click **Next**.

5. Select **850 Purchase Order**.

6. Click **Open**.

When the 850 finishes loading, your screen will look like this:

STOP

Comments

- Use **File | New** or the first toolbar button when starting a new guideline. Use **File | Open** or the second toolbar button when editing it later.

- When moving through lists, you can use the scroll bar, the Arrow keys, or the Page Up and Page Down keys.

Saving Your Work

Save early and frequently!

START

1. Choose **File** | **Save As** and enter **CLASS850** for the Standard Name.

   You may use up to 128 alphanumeric characters, underscores, and hyphens. Do not use spaces.

2. Replace the description text with **X12-5010 Purchase Order for EDISIM Tutorial**.

   Be specific with descriptions. They appear in every guideline list in EDISIM.

3. Choose **Properties | General** tab 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 the Dictionary.

   Analyzer will check a file's enveloping segments against those in CLASS850's dictionary. We will look at these later.

STOP

Comments

- You can also save a file by clicking the toolbar button.

- The "Interchange Envelope Segments" selection tells Analyzer where to find the interchange and group enveloping.

  – Default - use generic enveloping definitions in X12ICS 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 will use this setting if you select Analyzer's "use one standard" option.

Configuring Notes

EDISIM lets you add notes that help explain the guideline. You can have up to 100 "levels" of notes, each one serving a specific purpose.

These are useful in:

- Standards Editor to clarify the guideline

- Document Builder, where you can print different levels for each audience

By default, notes are labeled "Level 1," "Level 2," etc. You can change the labels to be more specific:

START

1. Choose the **Level Notes** tab and notice that you can change the number of note levels at the top of the box. For this exercise, leave the value at **3**.

2. Under Level Note Names, click on <not defined yet> and change the note level names to **Partner**, **Internal**, and **Example**, like this:
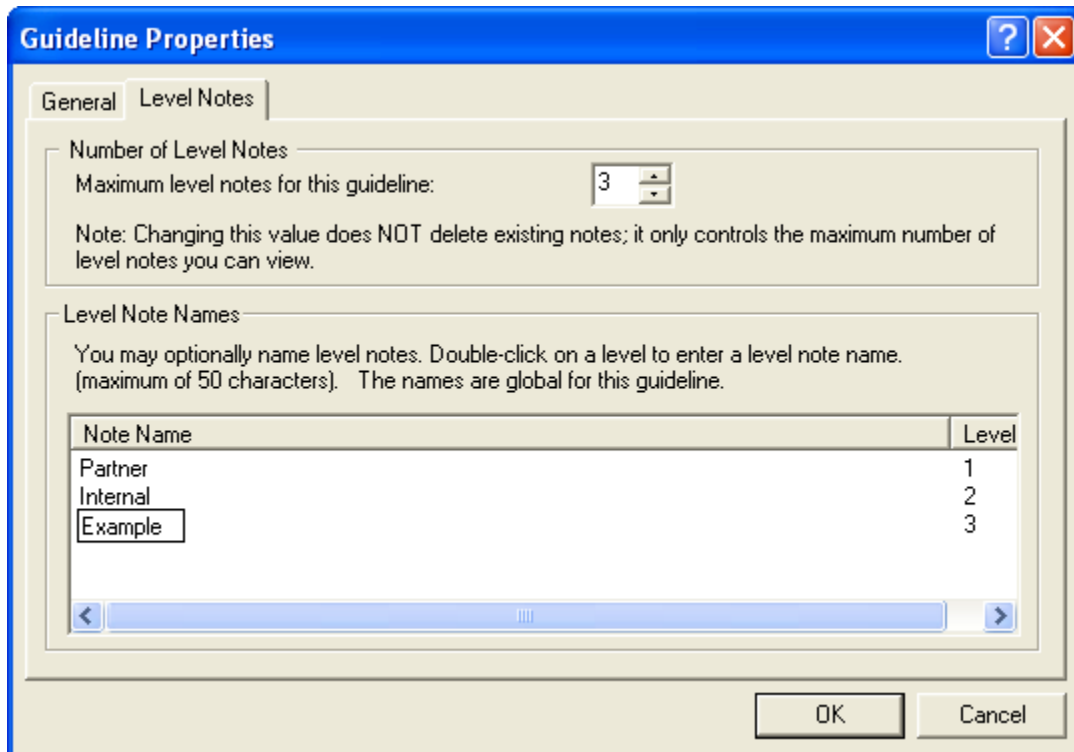
3. Click **OK | Save**.

   The guideline is saved as a .STD file under EDISIM's \User Files\Public Guidelines directory.
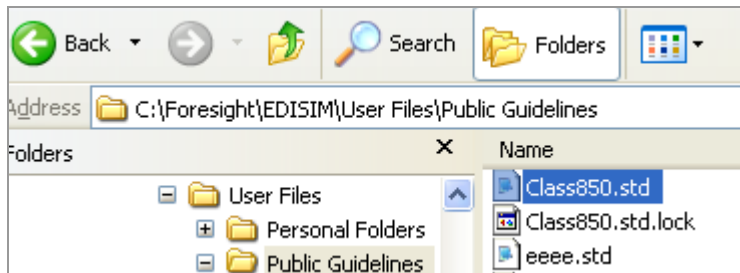
STOP

Getting Help

START

1. Click **Help | Contents**.
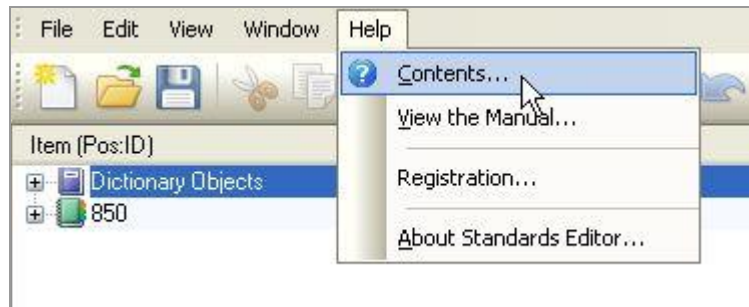
2. Look at each of the tabs and browse through a topic or two, then close help.

3. Choose **Help | View the Manual** and browse through a few pages, then close it.

4. Choose **Help | About Standards Editor** and notice the website and technical support phone number.

    Click **OK** to close the About box.

5. Choose **Help | Registration** and notice the field where you can update your registration key at renewal time.

STOP

Marking Segments as Not Used

A not used segment will not print in Doc Builder. It will generate an error message in Analyzer if it is in the data.

The Req (requirement) column in the top pane shows the X12 requirement. You can change the usage of items that are optional according to X12.

The U/A (user attributes) column in the top pane shows your company's usage.

START

1. Expand the 850 Transaction Set by clicking the **+** symbol that is before the **850.**

2. Expand **Table 1**.

   In the Req column in the top pane, you can see whether X12 considers the segment mandatory or optional.

3. Click the **ST** segment at position **0100** in Table 1, and using the scroll bar, scroll to the end of Table 1. Holding the Shift Key, click the **ADV** loop. This selects all segments between the ST and the end of Table 1.

4. Select **Edit | Mark Used/Not used**.

   Click in the highlighted area and notice that all of the optional segments are faded (not used), and the mandatory segments are not.

   The optional segments now have N (for not used) in the U/A column.
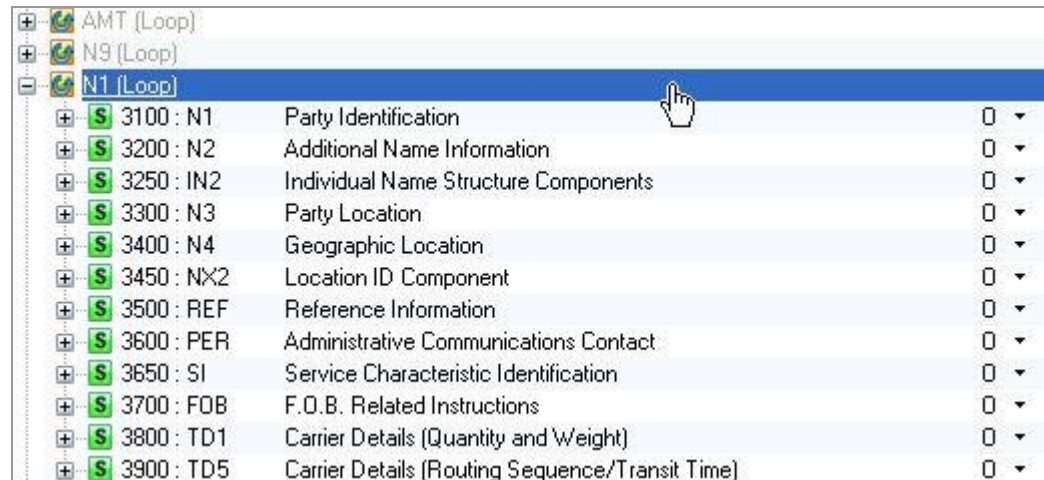
5. Right-click on the **REF** segment at position 0500 in Table 1 and choose **Mark Used/Not Used**.

   Notice that the **REF** segment returns to normal text from faded and is again marked as **O** (optional), as opposed to **N** (not used). The others remain marked not used.

Dragging the horizontal splitter bars resizes the panes.

---
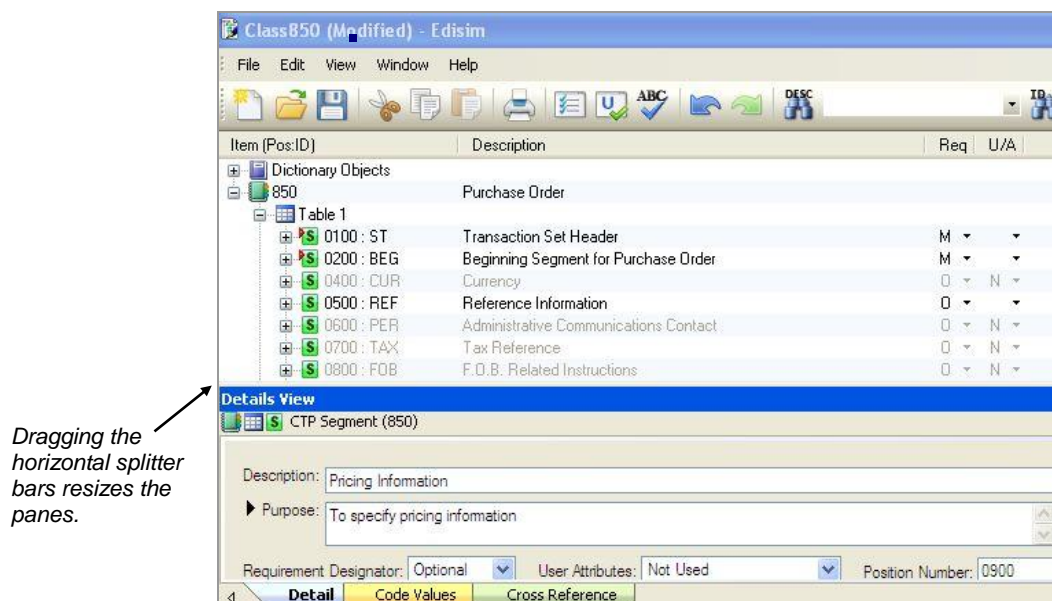
6. Go to **Help | Contents | Index** tab and look for information on **top pane**. This describes each column in the top pane.

7. Select the **PER** segment at position 0600. Use the *Alt+M* keys on your keyboard to mark it as Used.

8. Click the **Save** toolbar button.

9. Expand the **N1 Loop** just below the N9 Loop in Table 1 and be sure it is the one at position 3100. Highlight the **N1 Loop** line and mark it as Used.

    This marks all of the segments in the N1 Loop as Used.

⊞ 🔵 AMT (Loop)			
⊞ 🔵 N9 (Loop)			
⊟ 🔵 N1 (Loop)			
⊞ Ⓢ 3100 : N1	Party Identification	0 ▾	
⊞ Ⓢ 3200 : N2	Additional Name Information	0 ▾	
⊞ Ⓢ 3250 : IN2	Individual Name Structure Components	0 ▾	
⊞ Ⓢ 3300 : N3	Party Location	0 ▾	
⊞ Ⓢ 3400 : N4	Geographic Location	0 ▾	
⊞ Ⓢ 3450 : NX2	Location ID Component	0 ▾	
⊞ Ⓢ 3500 : REF	Reference Information	0 ▾	
⊞ Ⓢ 3600 : PER	Administrative Communications Contact	0 ▾	
⊞ Ⓢ 3650 : SI	Service Characteristic Identification	0 ▾	
⊞ Ⓢ 3700 : FOB	F.O.B. Related Instructions	0 ▾	
⊞ Ⓢ 3800 : TD1	Carrier Details (Quantity and Weight)	0 ▾	
⊞ Ⓢ 3900 : TD5	Carrier Details (Routing Sequence/Transit Time)	0 ▾	

10. Open **Table 3** and the **CTT** loop and mark the **AMT** segment as Not Used.

STOP

Comments

▪ Normally, use Mark Not Used rather than Delete. Delete is for correcting copy/paste mistakes.

▪ Leave **View | Show Unused** on when editing.

▪ Menus operate according to what item is selected.

Requiring a Segment

If X12 has designated a segment as optional, you can change its requirement.

The toolbar's speed search box searches the ID column.

START

1. Click on the **ST** segment at the top of Table 1.

2. From the toolbar, type **REF** in the speed search box and click the binoculars button to its right or press Enter:

3. Click on the arrow for the **Req** (Requirement Designator) field in the top pane.
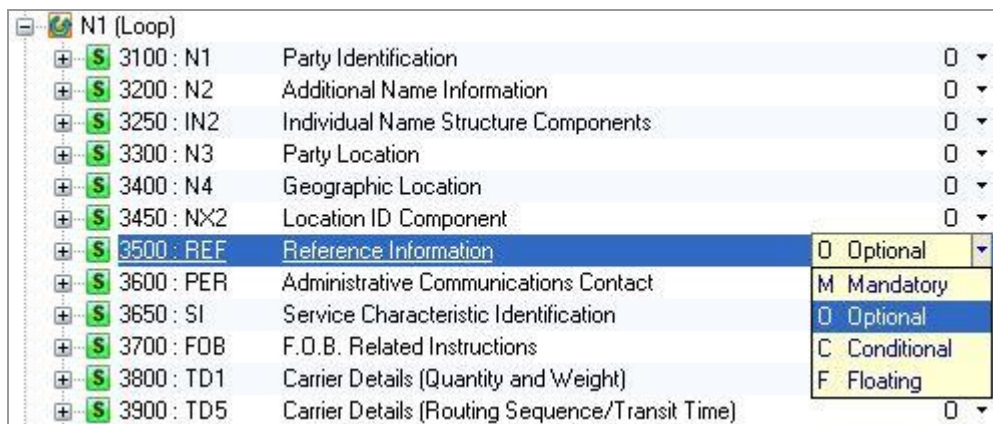
All X12 requirements (M, O, C, and F) are listed. These are Mandatory, Optional, Conditional, and Floating. Do not change the Req column settings.

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

5. In the top pane, the REF segment's U/A column now shows **MU** (Must Use). This is the column for your company's setting.
6. **Save**.

STOP

Marking Elements as Unused and Must be Used

START

1. Open Table 2, the **PO1** loop, and the **PO1** segment.

   You will see the PO1's elements, preceded with blue icons.

2. Select the elements at position **08** through **25**, and press *Alt+M* to mark these elements as not used.

3. Select element **639** at **PO105** (this is the 5th element in the PO1).

4. In the lower pane, click the User Attribute drop down arrow, and select **Must be Used (Required)**.
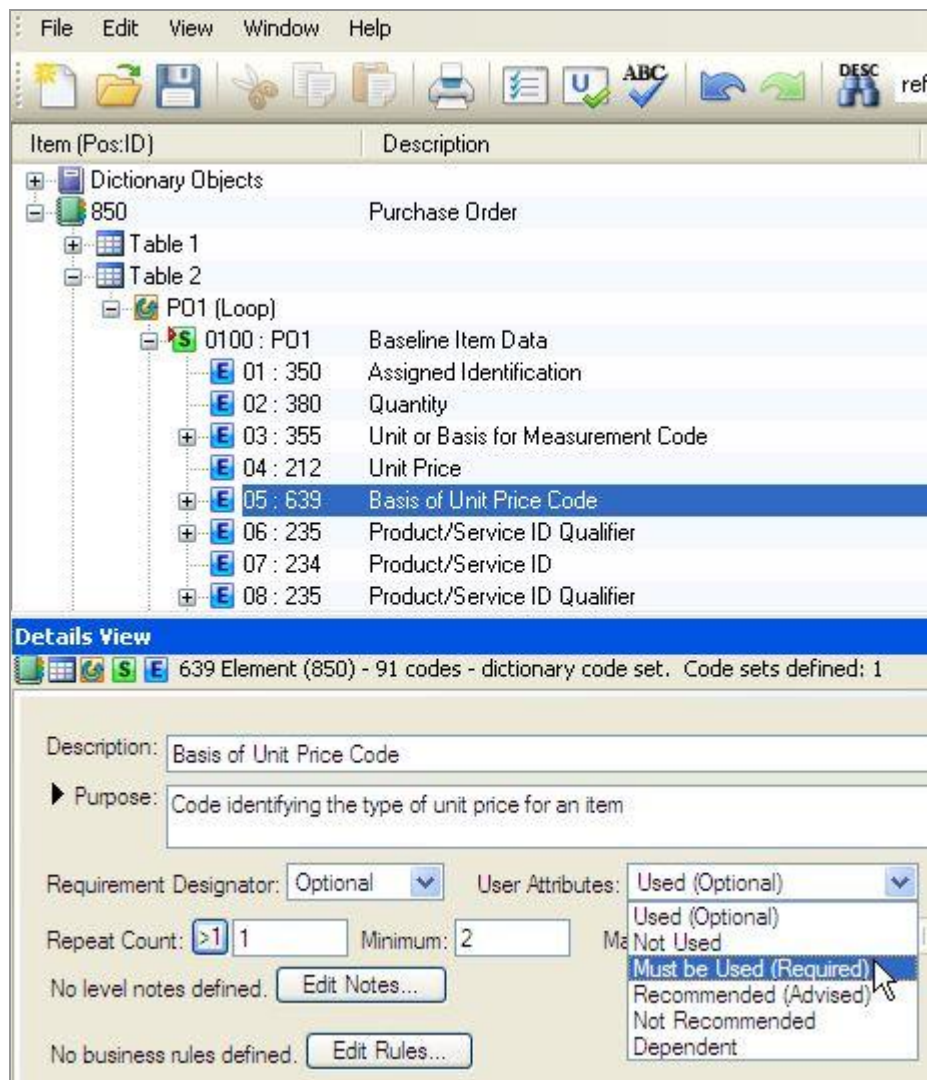
You can also use the bottom pane to change the Properties of elements or segments.

The U/A column in the top pane now shows **MU**.

5. Close the **PO1** segment.

6. **Save**.

STOP

Comments

- Selecting and marking user attributes en masse works for elements or segments.

- A gray field in the bottom pane is read-only information from the dictionary, but it can be copied. A white field can be changed.
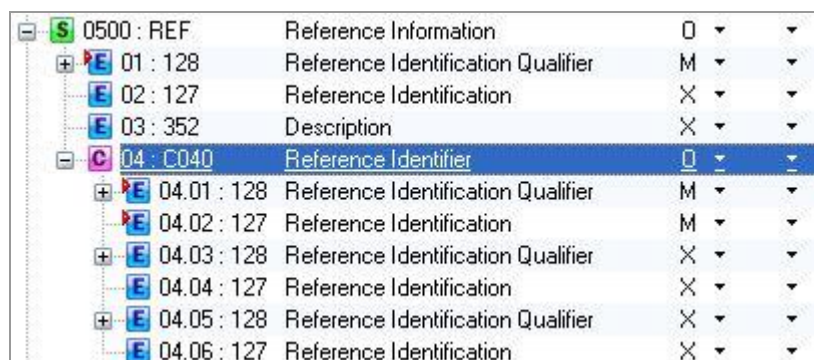
Working with Composites

A composite is a group of elements that are used together. A composite ID starts with a C in X12. An element within a composite is called a "sub-element."

START

1. Open the **REF** segment at **0500** in Table 1.

2. Open the composite element **C040** at **REF04**.

   Its sub-elements are displayed.

   The sub-element position numbers have two parts: two numbers representing the position of the composite within the segment, followed by two numbers representing the position of the sub-elements within the composite.

   The C040 composite has a Requirement and User Attributes, but has no Type, Min, or Max. That is because the composite itself does not contain data; its sub-elements do.

3. Select the sub-elements from **128** at **REF04.03** through **127** at **REF04.06** and choose **Edit | User Attributes | Not Recommended | OK**.

---

4. NR appears in the U/A column. We will not be using these sub-elements.

5. **Save**.

STOP

Comments

- In the EDI file, sub-element delimiters differ from element delimiters.

  Example (bold text is in the composite):

- `REF*AB*392039*NAILS*`**`BT:123456789:CM:500:AB:3920394930203!`**

Changing Element Length

START

1. Open the **PO1** segment at **0100** in Table 2.

2. Highlight element **350** at **P0101** and look at its Min/Max column in the top pane.

   The Min length is 1 and the Max length is 20, but we want exactly 12 characters.

3. Type 12 in the Minimum and Maximum fields in the bottom pane.

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

4. Save.

STOP

Editing Code Values

X12 provides code values for some elements. You can change this list.

START

1. Click on element **639** at **PO105**.

2. Select the **Code Values** tab at the bottom of the screen.

   The code values for 639 appear in the bottom pane.

3. Look at the Values title bar for code information.

   It reports 91 codes, but we will not use all of them.

4. Choose the 'Mark all of the code values not used' button on the toolbar below the title bar ().
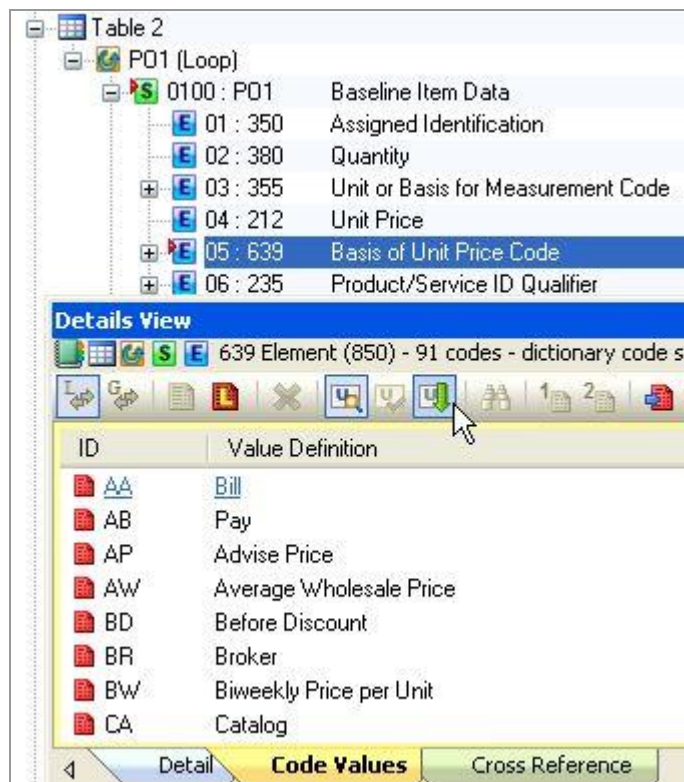
If the codes disappear, click in the bottom pane to activate the valid bottom pane toolbar buttons and then click the **Show or hide unused items** button. The codes reappear in faded text.
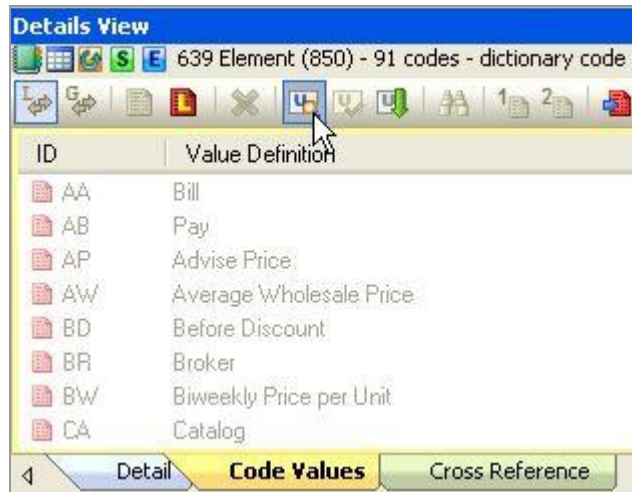
5. In the list of codes, click on **AP**. Use the scroll bar to find the **CP** code and Ctrl+click on it. Ctrl+click on the **RS** code also.

6. From the toolbar, select **Mark the selected code value(s) used or not used (Alt+M)**.
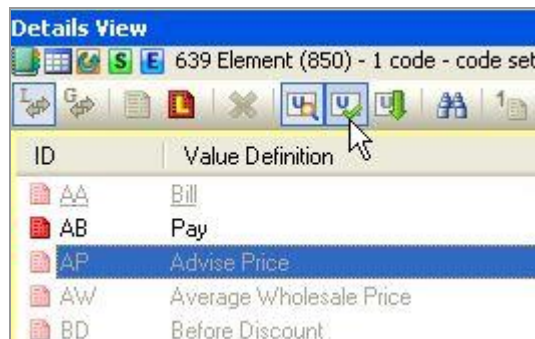
7. Use the button to toggle off display of the unused codes.

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

   This is another method of marking code values.

9. Save.

STOP

Comments

▪ When you change the list of codes, you are making a "Code Set" which will print in Doc Builder.

Adding a Local Code

You can add codes to the X12 list.

1. Click on one of the codes for the PO105.

2. Show the unused codes.

3. Click the Local Code button:
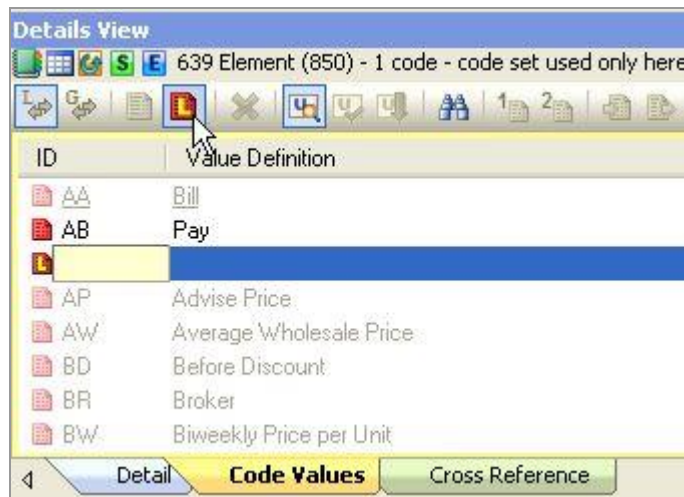
4. In the **Local Code** field, type **PC,** press the *Tab* key and type the definition **Price Cap**.
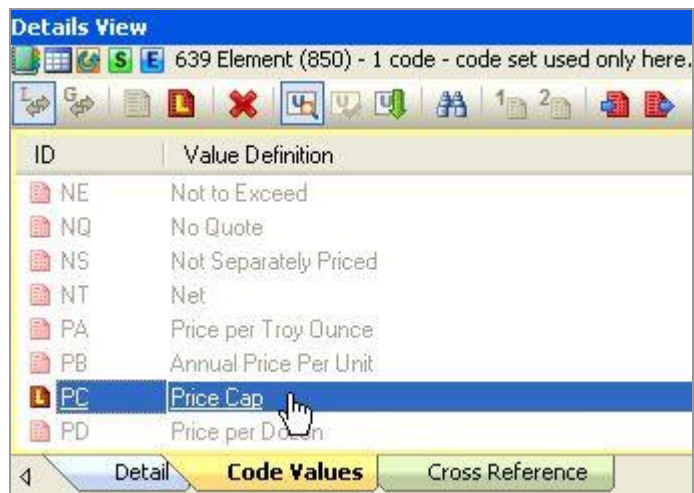
The title bar shows the number of codes in the Code Set.

5. Save.

Comments

- Normally, try to conform to the published standard by using only dictionary code values. For elements without codes, put local values into an application values list.

- Local codes can be added in the dictionary or in the transaction set.

Adding Level Notes

We specified in Properties (see page 10, Configuring Notes) that we wanted to have three levels of notes in this guideline:

- Level 1 Note = Partner

- Level 2 Note = Internal

- Level 3 Note = Example

Any time we change the guideline, we should add a level note clarifying the change and drawing attention to it.

Notes can go on any item (transaction set, segment, composite, element, or code).

START

1. To add a note to our Price Cap code, right-click on **PC Price Cap** and select **Level Notes/Properties**.

2. Add a Partner note (this is a Level 1 note) to the new code value.

   In the Notes area at the bottom, type **This is not an ASC X12 code, but we prefer it since orders process faster if it is used**.

Notes text goes here →
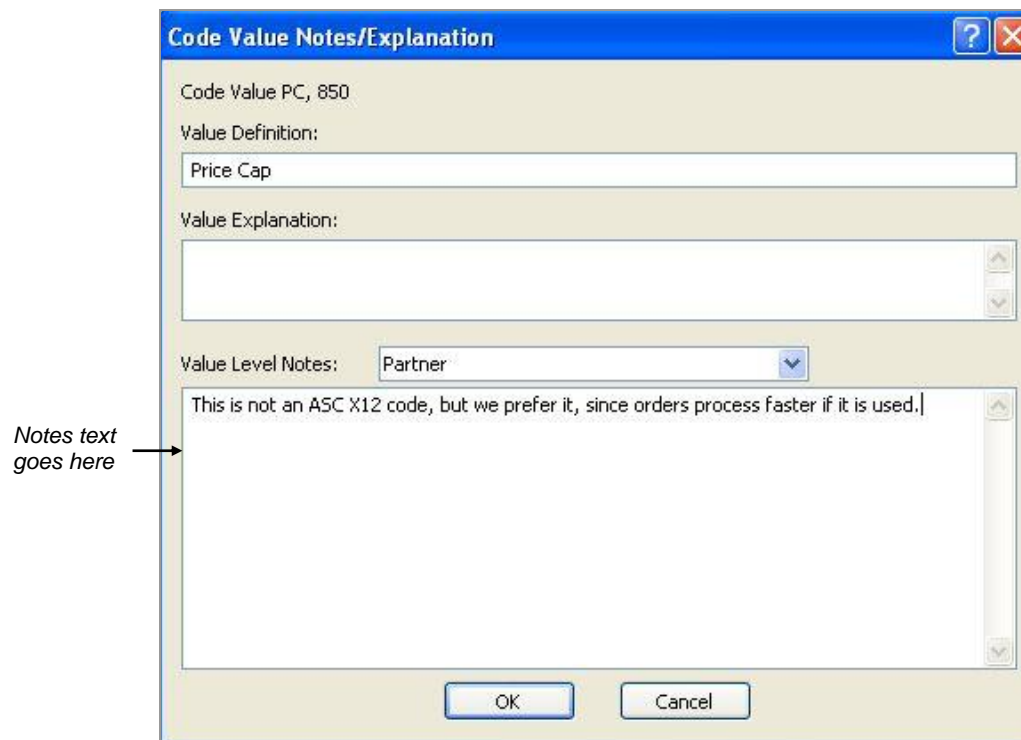
3. Click **OK**.

   The Indicators column now shows **L1** for a Level 1 note.
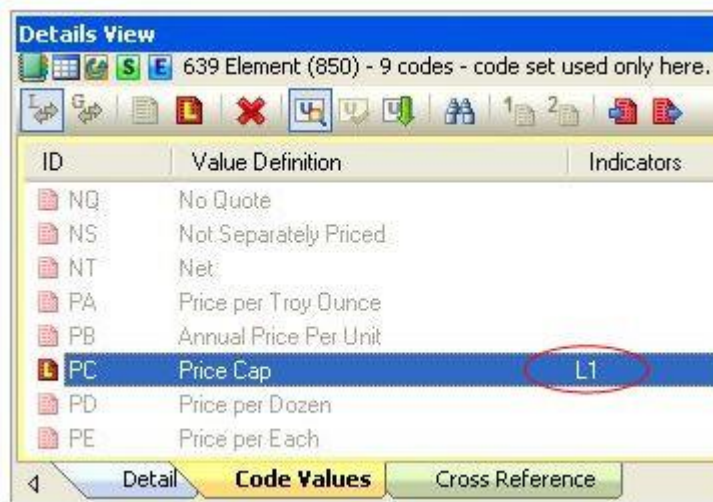
4. Right-click on element **639** at **PO105** in the top pane and select **Level Notes...**.
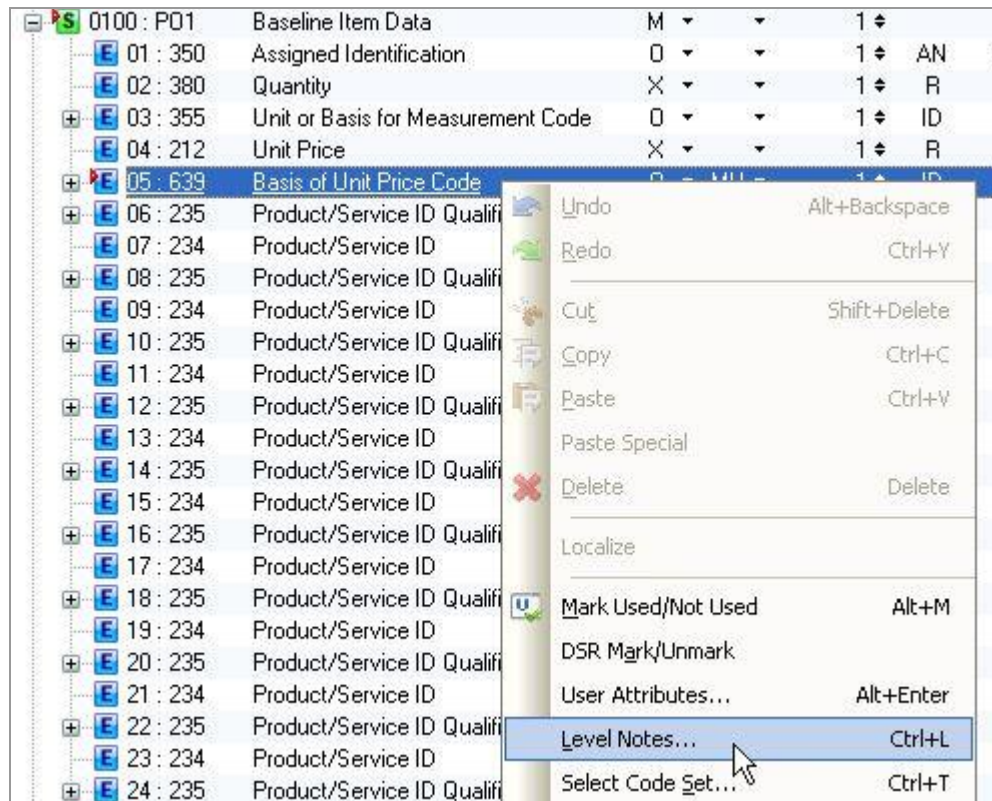
5. Select Note to Partner from the drop box (this is a Level 1 note).

6. Click in the Notes area and type **This element is optional for ASC X12 but mandatory for Kaver Corp.**:

7. Select Internal Note from the drop box (this is a Level 2 note). 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, if necessary.

8. Click **OK**.

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

9. Add these notes to the **PO1** segment:

    Partner: **Use a PO1 loop for each item being ordered.**
    Internal: **Is there a limit to the number of PO1 loops that we can send?**

10. Add these notes to the **850 Purchase Order**:

    Partner: **This purchase order goes into effect May 1, 2007.**
    Internal: **Will the Eastern Division be using this too?**

11. **Save**.

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.

- Where are notes used?

    – You can print or suppress any note level in Doc Builder.

    – Notes show in Standards Reference.

    – Analyzer cannot understand notes.

Adding a Note from an Outside Source

In this exercise, we are going to use a level 3 note for a segment example.

START

1. From Explorer, go to EDISIM's **Samples** folder and double-click on **testpo-5010-A.txt**.
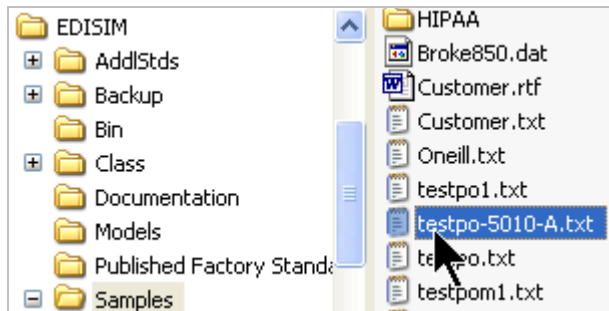
2. Copy the entire **PO1** segment.

3. Close textpo.txt and return to Standards Editor.

4. Right-click on the **PO1** segment and choose Level Notes.

5. Select the **Level 3 note** from the drop-down box.

6. Type **Example:** and press *Enter* twice. Press ***Ctrl+V*** to paste the example.

7. Click **OK**, close the **PO1** segment, and save

STOP

Comments

- You can copy notes from other places within EDISIM, or from another application, such as a word processor, and past them into the guideline.

- If you copy text into notes, you can make their line breaks look better in DocBuilder by turning on **File | Preferences | Appearance | Wrap Soft Returns in Pasted Text**. This causes new lines in pasted text to become single spaces. Two or more consecutive CR/LF result in hard-returns.

Copying and Pasting Loops within a Guideline

You can copy and paste segments and loops. Customizations that have been done for those items (user attributes, notes, code list changes, etc.) will copy also. This exercise will copy and paste an N1 loop, so each can be customized for the guideline.

START

1. Make sure **View | Show Unused** is on for the Segments pane. It must be on before pasting.

2. Click on **Loop N1** at position 3100 in Table 1.

3. If it is open, close it.

4. Choose **Edit | Copy**.

   This will copy the entire loop.

5. Leave the Loop N1 line selected, and choose **Edit | Paste**.

   We pasted a second N1 loop just after the first one.

   Pastes always go BELOW whatever is selected when you perform the paste.

6. Turn off **View | Show Unused** and re-open Table 1.

7. Save.

STOP

Comments

- To MOVE segments or loops, use **Edit | Cut**.

- The position numbers of pasted items are reused to keep in sync with X12.

- Be careful not to nest a pasted loop within another loop, unless that is what you mean to do.
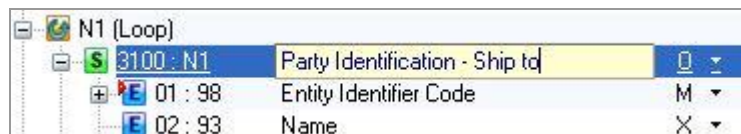
Customizing the Two N1 Loops

We customize each of the two N1 loops so that our printed guideline will clearly show that we have two different types of N1 loops.

We will change the name of each N1 slightly so that we can easily tell them apart, and then set the requirements specifically for each loop.

START

1. Open the first **N1** loop and change the description of the N1 *segment* to **Party Identification - Ship to**.

2. We must have this loop, so in the **U/A** (User Attributes) column, choose **Must be Used (Required)**.

   The requirement for the first segment in a loop sets the requirement for the loop itself.
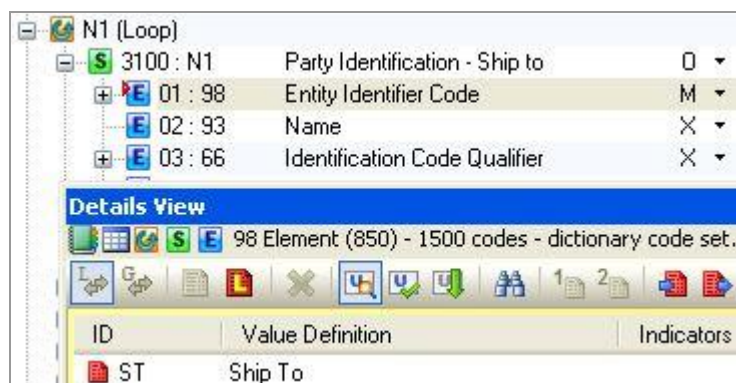
3. The Ship to N1 loop can be used only once. In the **Repeat** column, on the N1 *Loop* line (not the segment line), change the repeat count from 200 to **1**, and click **OK**.

4. Open the N1 *segment* and click on data element **98** at **N101**.

   We're going to edit the first code value in the Ship to N1 loop's first segment so that Analyzer can tell the loops apart.

   On the **Code Values** tab at the bottom of the screen, mark all codes unused with the 🔲 button.

   10. If all of the code values disappear, choose the Show unused button 🔲.

   Right-click on the ST code and press **Alt+M** to activate it.

---

5. Change the code set for element **66** at **N103** to include only **9** and **92**.
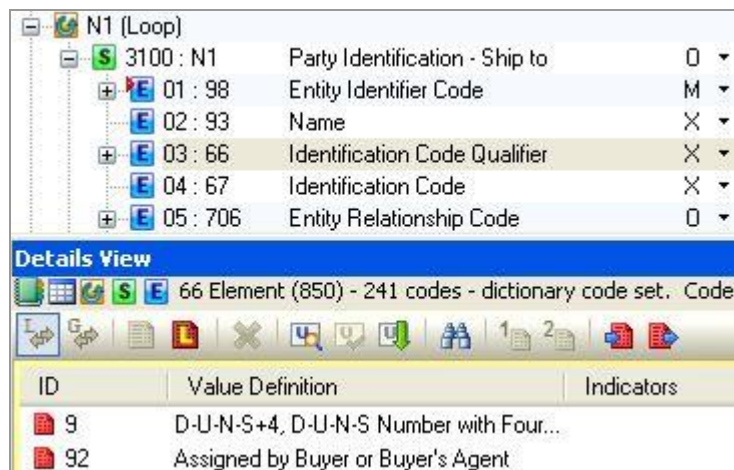
6. Close the Ship-to N1 loop.

7. Save.

   We have just required one N1 Ship to loop.

Now, for the second N1 loop:

1. Open the second **N1** loop and make sure the position number is **3100** for the N1 segment.

2. Change the description of the N1 *segment* to **Party Identification - Bill to**.

3. Change the Repeat from 200 to **1** for the N1 *loop* (not the segment).

4. Change the code set for the **N101** to just have code **BT**.

   The ST from the first loop and the BT from this loop will let Analyzer to tell the two loops apart.

5. Right-click on element **66** at **N103**, choose Select Code Set, and choose the one that includes 9 and 92:
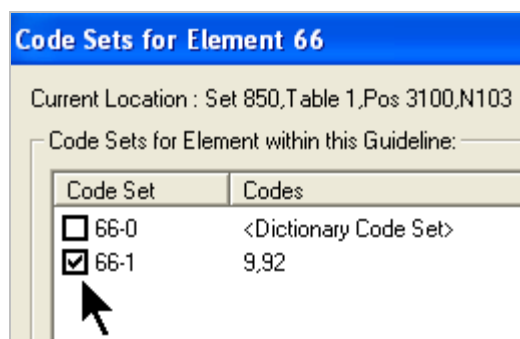
6. Click **OK**. You have re-used the codes previously defined in the Ship to N103.

7. **Save**.

   The heading above the lower level toolbar will now indicate that the code set is used by 1 other Element.

8. In the N1 Bill to loop, Mark the segments from **SI** through **PKG** as unused.

9. **Save**.

STOP

Comments

- Analyzer can match an N1 loop in the data to the correct one in the guideline by looking at the first code in the first segment of each N1 loop. If necessary, Analyzer will look at the next code in each loop to try to tell them apart.

- If you edit the code values in a code set that is used in more than one place, Standards Editor will ask if you want the changes to be local or global.

Attaching Application Values

For elements without code values, you can require specific values. These are called "application values."

START

1. Open the **N3** segment in the **N1 Name-Ship to** loop.

2. Right-click on data element **166** at **N301** and choose **Application Values**.

3. In the Value List field, type **RECVDOC**, the name we are giving to our list.

4. *Tab* to the Description field, type **Receiving Dock**, and press *Enter.* The cursor moves to the Value field at the bottom.

5. In the Value field, type **LOADING DOCK 437**, and press *Enter*.

6. Type **LOADING DOCK 3**, and press *Enter*.

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

    In the top pane, the Indicator column for the N301 now displays **VR** for Values Reference.

STOP

Comments

- Application Values and regular expressions are case sensitive.

Requiring Values with Specific Formats

For elements without code values, you can require patterns of values called **Regular Expressions**. These are a variation of application values.

START

1. Open the **REF** at 0500 in Table 1.

2. Right-click on the **REF02** and choose **Application Values**.

3. In the Value List field, type **REFID**, the name of this list.

4. For Description, type **Reference ID** and press *Enter*.

   The cursor moves to the Value field.

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

   This expression requires a value that starts with **ABC** followed by 2 capitalized alpha characters and then 3 numeric characters such as **ABCAB123**.

   The caret symbol (^) means nothing can appear before the regular expression.

   The dollar sign ($) means no characters can appear after it.

6. Select **Regular Expression** at the bottom.
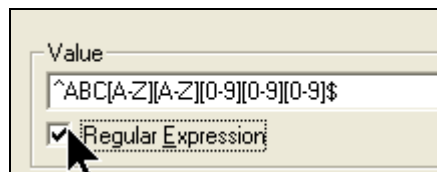
7. Click **Add** and **Attach**.

   Again, the Indicator column in the top pane shows VR for Value Reference.

8. Add a level 1 note for your trading partners that explains the regular expression:

   The required value for this element starts with ABC followed by 2 alpha characters and 3 numeric characters. Example: ABCAB123

9. Click **OK**.

10. Save.

STOP

Comments

- Analyzer will require a value that matches the regular expression pattern.

Creating a Business Rule to Set Usage

You can create a business rule to reflect a particular requirement for your guideline. These are usually "if-then" conditions.

We will create this rule:

If   the code value for element 353 at BEG01 = 04
Then the PER is required

START

Define a variable on the first element in the rule:

1. Open the **BEG** at **0200**.

2. Right-click the **BEG01** and select **Business Rules**.

3. Click in the **Local Variable** text box and type **BEG01TransCode**.

   We recommended that you use the segment and the element position as part of the name so that it will be easier to locate. You can use up to 16 alphanumeric characters.
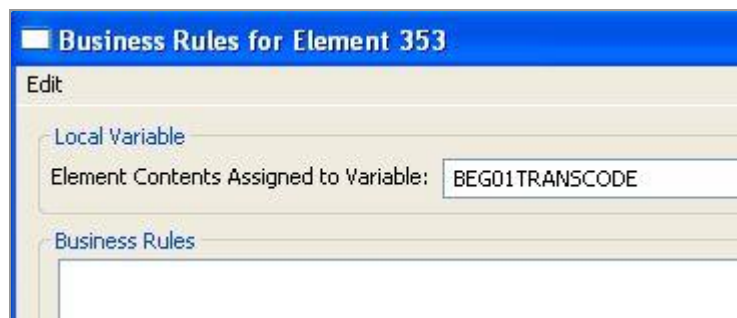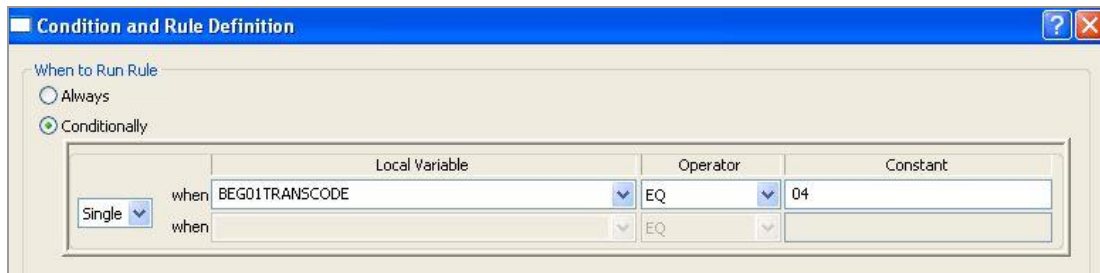
4. Click **OK**.

Now, create the business rule on the PER segment, that depends on the value in the variable:

1. Right-click on the **PER** at position **0600** and choose **Business Rules**.

2. Click **New**.

3. Click **Conditionally.**

4. Select **Single** to activate one "if" condition.

5. Click the arrow for the first field and select our variable **BEG01TransCode**.

6. In the second field, select **EQ** (equals), and then type **04** in the third field.

This line now shows the "if" condition.
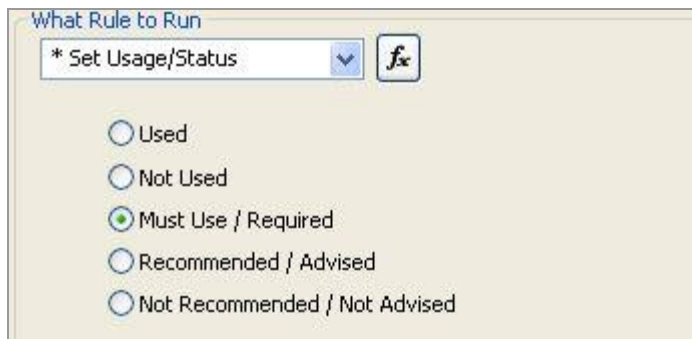
7. Start setting up the "then" condition by ensuring that **Set Usage/Status** is selected.

8. Click **Must Use/Required**.
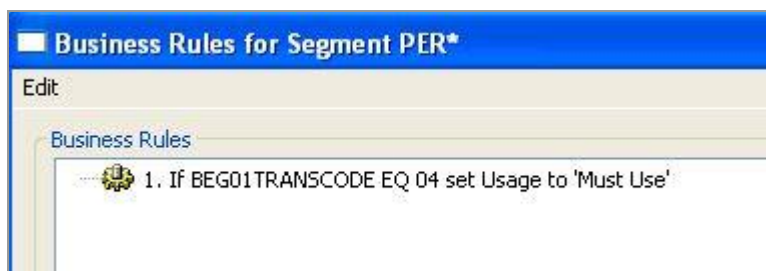
   The new rule now says the current item (PER segment) must be used if the BEG01 contains 04.

9. Click **OK**.

   The new rule is added and Analyzer will check for it. The rule is displayed in the Business Rules box:
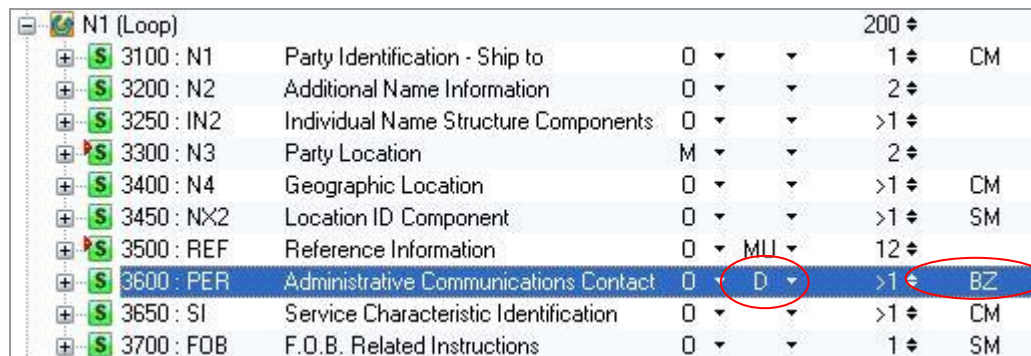
   You could click **New** again and set up another rule on the same PER that says if the BEG01 is not 04, usage is Not Used.

10. When you are done with rule creation, click **OK**.

In the top pane, we see that the segment is **D**ependent and its Indicators column contains **BZ**.

11. Save.

Comments

- A variable must be located before the rule where it is used. In this exercise, the BEG01 is first so the variable is attached to it, and the rule itself is attached to the PER segment.

Creating a Business Rule to Use Application Values

You can use an application value list under "if-then" conditions.

To do this:

1. Create an application values list but do not attach it anywhere.

2. Create a business rule that uses the list under certain conditions.
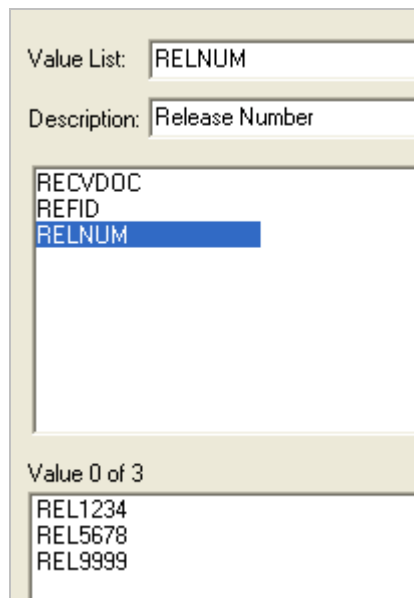
Create a business rule that states: 'If the code value for element 353 at BEG01 = 18 then use the Application Value List PONUM at BEG04'.

START

First create an Application Value list:

1. Right-click anywhere in the top pane and choose **Application Values**.

2. In the Value List field, type **RELNUM**.

3. In the Description field, type **Release Number** and press *Enter*.

4. In the Value field, type **REL1234** and press *Enter*.

   Add two more values: **REL5678** and **REL9999**.

   Attention: Do NOT click Attach.

5. Click **Close**.

We now have an application value list that can be attached under certain conditions. To set up these conditions:

1. Right-click on the **BEG04** and select **Business Rules**.

2. Click **New**.

3. Set up the "if" condition in the When to Run Rule area:

   Select **Single**.

   Select the variable **BEG01TransCode** in the first field, select **EQ** in the second field, and type **18** in the third field.

4. Set up the "then" condition in the What Rule to Run area:

   Select *** Use Application Value List**.

   Select the RELNUM application value list.

5. Click **OK** twice.

6. Save.

Analyzer will enforce this condition.

STOP

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 the entire transaction set.

START

1. Click the **ST** segment and then click the spell check toolbar button ( ).

2. Choose these options:
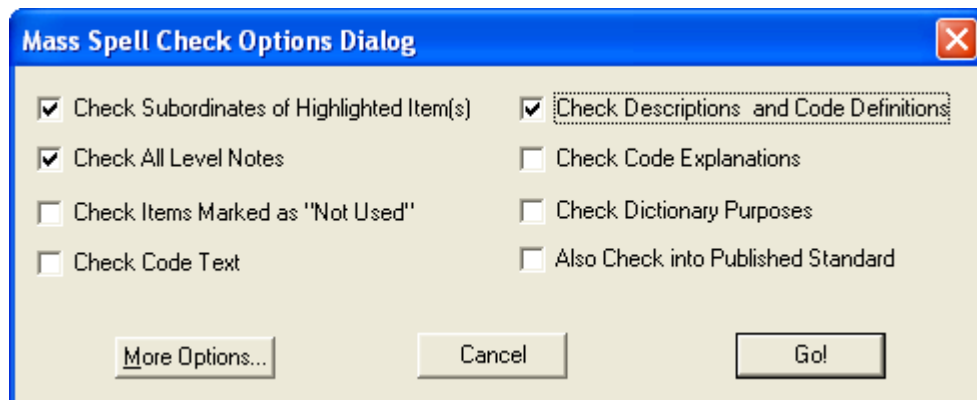
3. Click **More Options** and look at the options.

4. Click **OK**.

5. Click **Go!**.

   If no misspelled words are found, the "Spell Check Done" message appears.

   You have the opportunity to fix misspelled words or add words to the dictionary.

6. Click **Close**.

STOP

Using the Dictionary

The dictionary is a list of all segments, composites, elements, and codes in the underlying standard (X12-5010 in our case).

START

1. Open **Dictionary Objects** at the top of the top pane.

2. Open the **Segments** dictionary.
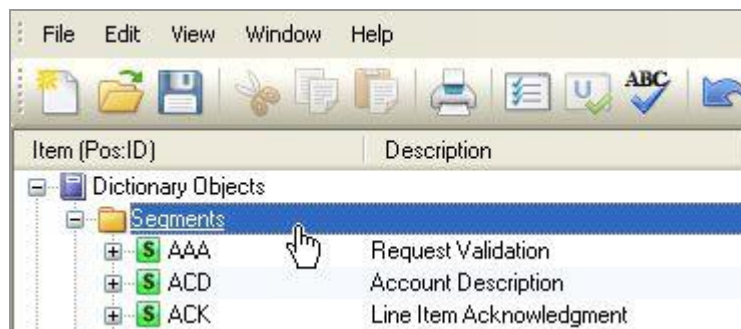
3. Search for the **ISA** segment
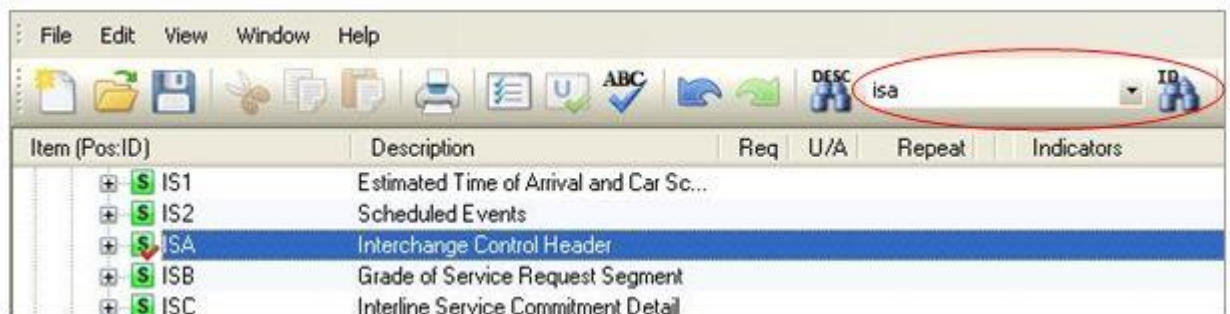
4. Open the ISA segment and notice the bottom pane:

    The **Detail** tab has some choices not available in the transaction set.

    The **Code Values** tab shows the complete list of X12 dictionary codes.

    Dictionary segments can be edited, including codes, comments, and syntax rules. These changes carry through to the transaction set.

5. Close the Segment dictionary.

6. Open the **Composites** dictionary.
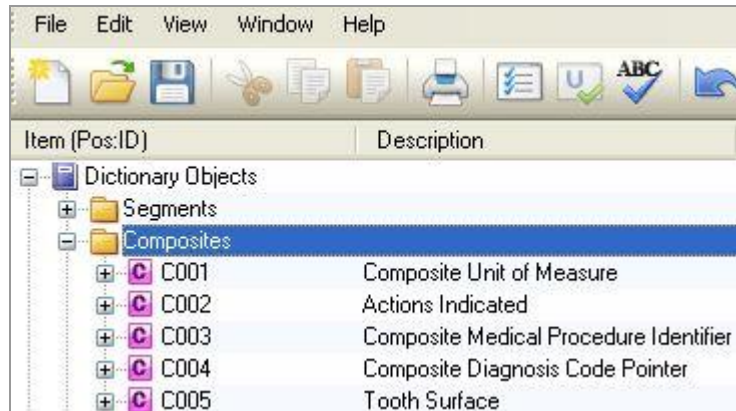
7. Open a composite and view its sub-elements.

8. Close the Composites dictionary.

9. Open the **Elements** dictionary to see all elements in X12-5010.

10. Close the Elements dictionary.

11. Open **Code Collections**. This is a list of all the code sets found in this guideline and the standard on which it is based.

12. Open element **66** and notice the dictionary codes and the 9, 92 code set we created on the N103.

13. Close Dictionary Objects.

14. Click on **850 Purchase Order** in the Transactions pane to return to the transaction set.

STOP

Comments

▪ Enveloping segments can be customized in the dictionary.

▪ Changes in the dictionary carry through to the transaction set.

▪ **Caution**: Creating a code set in the Element Dictionary will permanently delete the unused codes.

Exporting a Guideline

You can export your guideline to a .SEF (Standard exchange format) file, which you can:

- Share with other EDISIM users

- Use as a backup

- Use to populate a SEF-ready translator

START

1. Save.

   You must save before you can export.

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

3. Go to EDISIM5's **User Export** folder and save the file as **CLASS850.SEF**.

   You are creating a SEF file, which will serve as a backup.
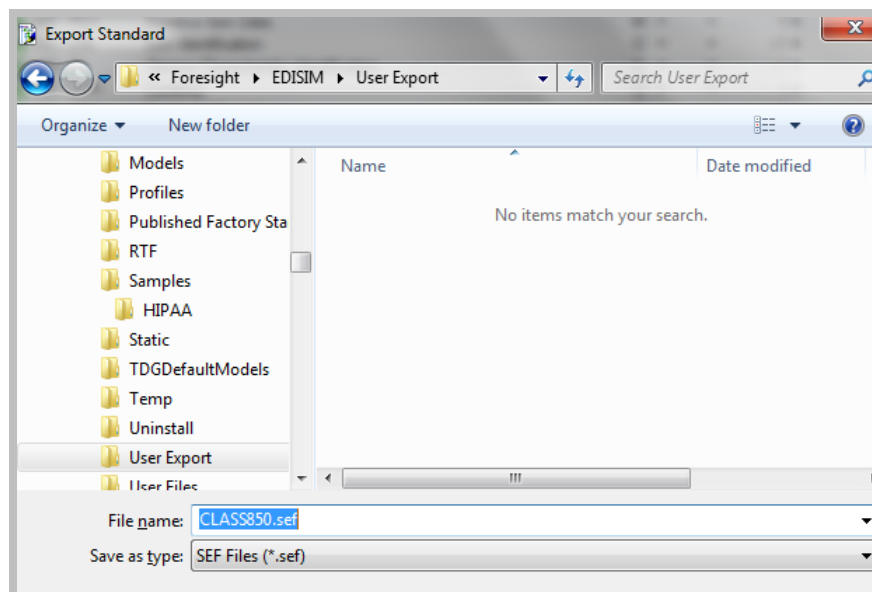
4. Exit Standards Editor.

STOP

Comments

- Be sure to zip a SEF file before sending it electronically since it contains some long lines, which can be broken during e-mail transmissions.

- From within Standards Editor, use **File | Import** to import a SEF file or DocStarter file into the EDISIM database.

Re-Opening a Guideline

START

1. Enter Standards Editor again.

2. If the Welcome to EDISIM Standards Editor box appears, choose **Open | User Guidelines** and select the desired guideline.

   If the Welcome to EDISIM Standards Editor box does *not* appear, choose **File** and look at the menu.
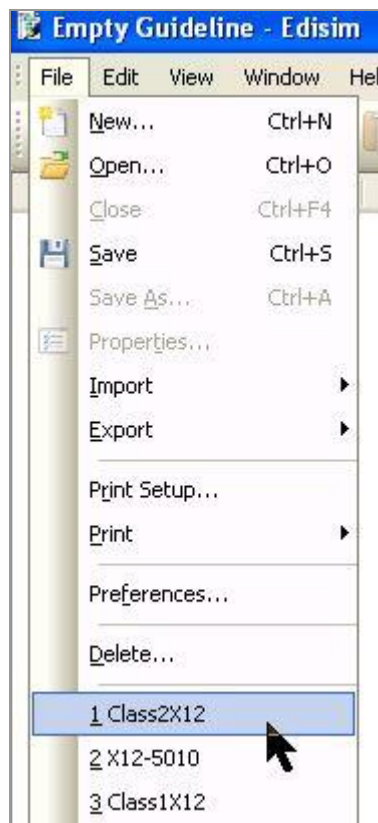
Choose guideline near the bottom of the menu.

Note: Do not use **File | New** on a guideline you've already created unless you want to create a separate guideline based on it.

STOP

Exiting without Saving

START

1. Choose **File | Exit**.

2. DO NOT SAVE. Click **No** if asked whether 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 EDISIM's User Files\Public Guidelines directory. By default, this is:

  C:\<*TIBCO_HOME*>\EDISIM\User Files\Public Guidelines

- Export an individual guideline or multiple guidelines 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, and 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 trust all backups to the same device.

Doc Builder

Create Formatted Guideline Documentation

For more information: See **TIB_fsp_edisim_*<n.n>*_fsdocbld.pdf** in EDISIM's Documentation directory

Purpose

The purpose of this exercise is to generate documentation from the guideline that we customized in the last exercise.

Why use Doc Builder?

With DocBuilder, you can create professional-quality documents from any guideline in Standards Editor. You can format it to look the way you want.

Output can be print, RTF (word processor), or HTML.

Selecting a Guideline, Transaction Set, and Profile

START

1. Choose **Start | Programs | <*TIBCO_HOME*> | EDISIM | Document Builder**, or, from within other EDISIM modules except Standards Editor, use this toolbar button:

    Within Standards Editor, use this toolbar button:

2. Maximize your screen.

3. Choose **File | Open Standard**.

    Only guidelines (not published standards) are available in Doc Builder.

4. Select **CLASS850**, the guideline you just created in Standards Editor, and click **OK**.

5. In the Transaction Set/Message Selection dialog box, click on **850 Purchase Order**, so our generated document will use this transaction set.
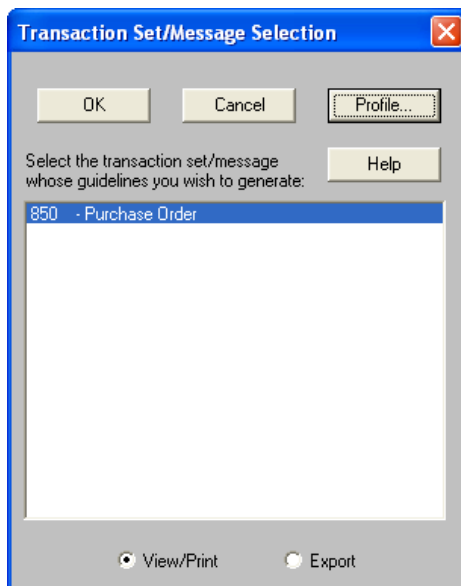
6. Click **OK**.

STOP

Viewing the Document on the Screen

The first page is the beginning of the Segments Table, which lists all segments in your document.

START

1. Look at the Partner and Internal notes that you put on the transaction set just under the Introduction.

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

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

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

4. Click the **Next Page** button on the toolbar to advance one page at a time through the document until you get to the **ST** and **BEG** pages.

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.

Changing Doc Builder Options

We are going to customize the appearance of our documentation.

We want some changes to the segment table:

Bold level 1 notes

Requirements next to each other with headings

Page numbers

No unused segments showing

Page No.	Pos. No.	Seg. ID	Name	X12 Req.	Kaver Req.
6	0100	ST	Transaction Set Headersp	M	M
7	0200	BEG	Beginning Segment for Purchase Order	M	M
9	0500	REF	Reference Information	O	M/U
11	0600	PER	Administrative Communications	O	D

START
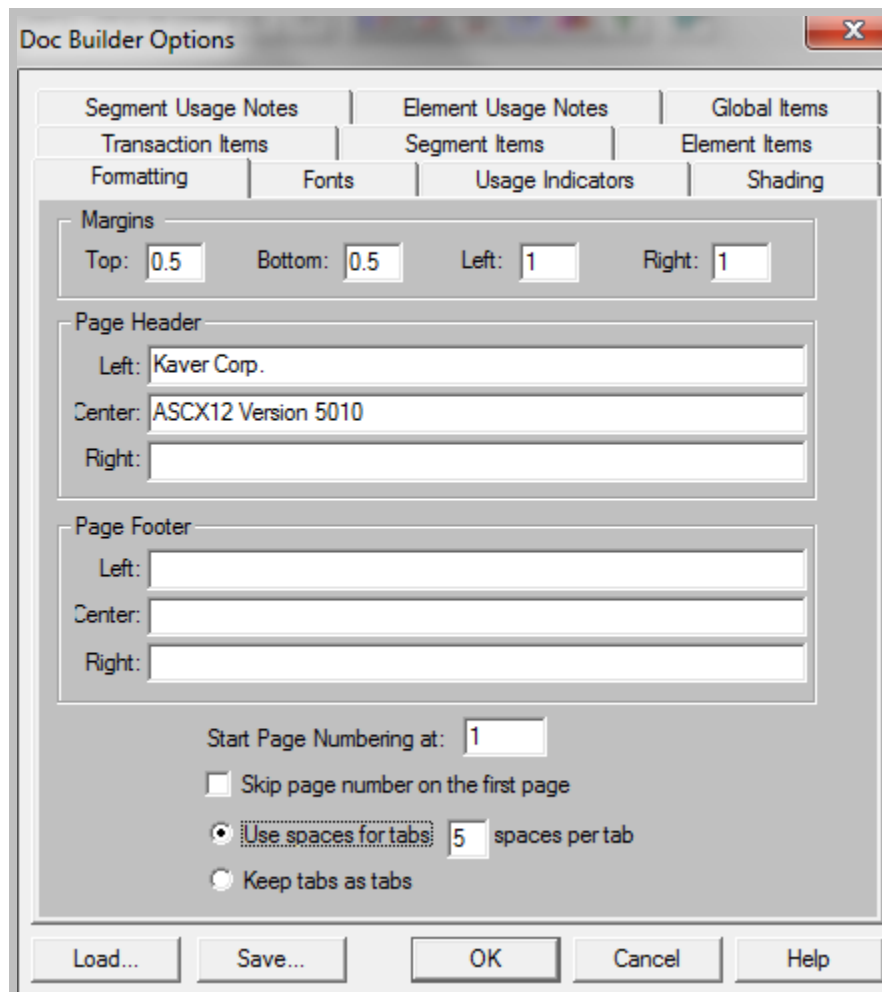
1. Choose **Profile | Options**.

2. In the Page Header field, type **Kaver Corp**.

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

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

3. For Left Footer, type **ASC X12 Version 5010**.
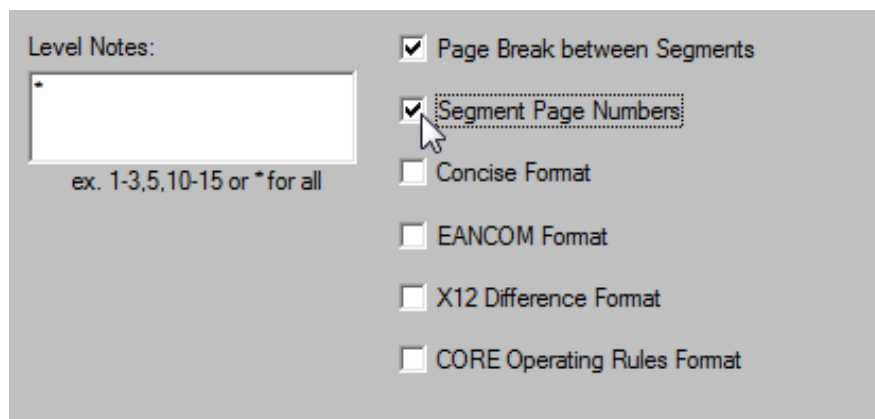
    This will appear at the bottom left of each page.

4. Click the **Global Items** tab select **Segment Page Numbers**.

    This will display a column of page numbers on the left side of the Segments Table, making it like a table of contents.

5. Click the **Transaction Items** tab and select **User Status**.

    This puts our usage right next to the X12 usage column.

    Type the heading text for each column as shown below:

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

    This will eliminate unused segments from the Segments Table.

7. Click the **Element Items** tab and select **Placeholders Only** and **Regular Expressions**.

    This will display minimal information about unused elements will include the regular expressions (value patterns) that you created in Standards Editor.

8. Click the **Fonts** tab and select **Bold** for **Level 1 Notes**.

   The bold font will make your Partner notes more noticeable.

9. Click the **Usage Indicators** tab.

   This is where you specify the text for usage indicators.

   For each indicator, you can select from the drop-down list, type your own text, or leave it blank.

   Choose N/U for Not Used segments and elements.

   Choose M/U for Must Use segments and elements.

10. Click the **Segments Usage Notes** tab, and in the **Must Use** field, type this:

11. Do the same in the **Element Usage Notes** tab.

STOP

Comments

- For more information, choose **Help | Index | Contents tab | Doc Builder Details | Customizing your Document with a Profile**.

Saving Doc Builder Options

Now that we have customized our Doc Builder profile, we want to save it for future use. This saves time and ensures that our documents are consistent.

START

1. Click the **Save** button at the bottom.

2. Go to **EDISIM's User Files** folder and make a new folder called Profiles.
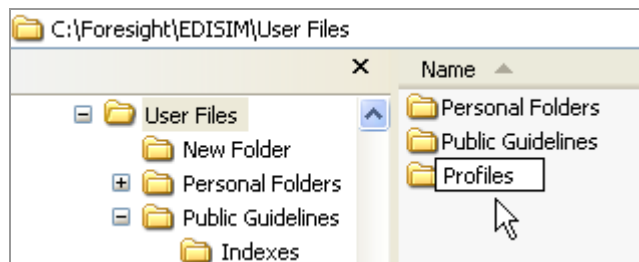
3. Type **Usual_internal.DPF** in the File name field.

We chose this name to remind us that this profile should be used for internal company use. It displays all levels of notes.

We will also need a profile for external use. It should not display Level 2 (Internal) notes.

Notice the option at the bottom to save the 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.

---
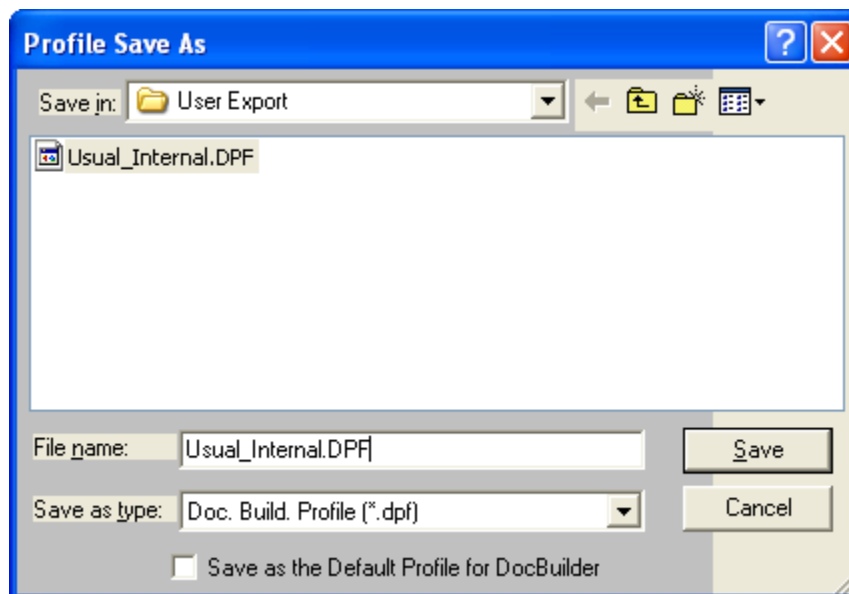
4. Click **Save**.

   This profile stays in effect until you close Doc Builder. The next time you open Doc Builder and want to use the saved profile, choose **Profile | Open** and select Usual_Internal.DPF.

5. Click **OK** and then **Yes** to regenerate with the new options.

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 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.

- Company-wide profiles give consistency and prevent errors. You may want to save your profiles to a network drive so others can use them.

Using Print Preview

Before choosing an output method, it's a good idea to check your document with Print Preview.

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

START

1. Click the **Print Preview** toolbar button.
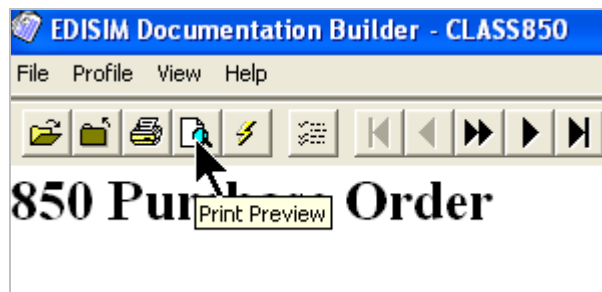
    The full page is viewable and the cursor is now a magnifying glass.

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

    DocBuilder tries to zoom in on the area where you clicked.

3. Click once more to return to full page 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

Checking the Display Created by your Profile

START

1. Click the First Page button ◄ to go to page 1.

2. Notice the page numbers at the left margin.

3. Look up the page number of **Name - Ship To N1**.

4. Use the **Go To** button ►► to go to the **Name - Ship to N1** page.

   Review the changes that you made in Standards Editor: the ST code value in the N101, the 9 and 92 code values in the N103, and the shaded usage notes.

5. Return to the Segments Table and find the page number for **PO1** at **010**. Use the **Go To** button to go to it.

6. Look for your notes on the **PO1** itself, on the **639**, and on the **PC** code.

STOP

Exporting a Guideline to a Word Processor Format

You can export your guideline as an RTF (Rich Text Format) file.

This is a generic word processor file that you can open with Word or any other word processor.

This gives you the power to make any changes that you'd like.

It gives you a word processor document for those who do not have EDISIM.

START

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

2. Click on 850 Purchase Order.

3. Click the Export button at the bottom.
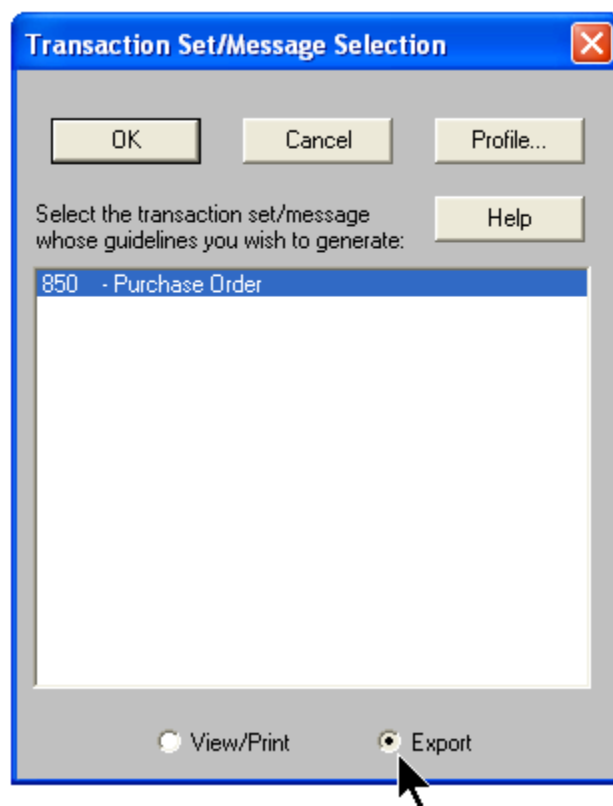
4. Click OK.

5. Go to EDISIM's User Files folder and make a new folder called RTF.

6. Save the file in that directory and call it CLASS850.rtf.
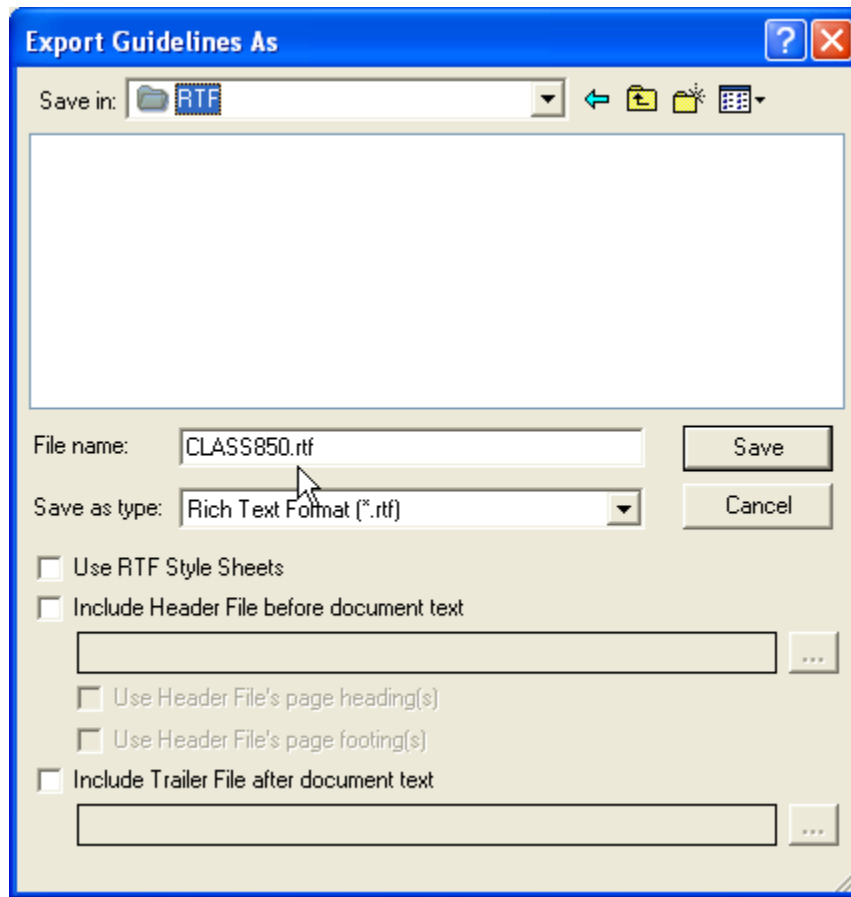
Notice that you can choose to have the output use special TIBCO Foresight styles that start with DB.

You can also include other RTF files before or after the transaction set.

7. Click **Save**.

You now have an RTF file saved in the User Export directory.

STOP

Comments

- You can edit the RTF document with a word processor.

- Do not make structural changes in the RTF file; make them in Standards Editor and then regenerate the document.

Creating HTML Files

You can quickly create HTML documentation from within Doc Builder.

START

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

2. Go to EDISIM's **User Export** directory.

3. Click the **Create New Folder** button, type **HTML850** for the new folder name, and press *Enter* twice so that 'HTML850' appears in the Save in field.

4. Name the file **CLASS850.htm**, and click **Save**.

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

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

   The Segments Table will be in one file and each segment detail page will be in its own separate file.
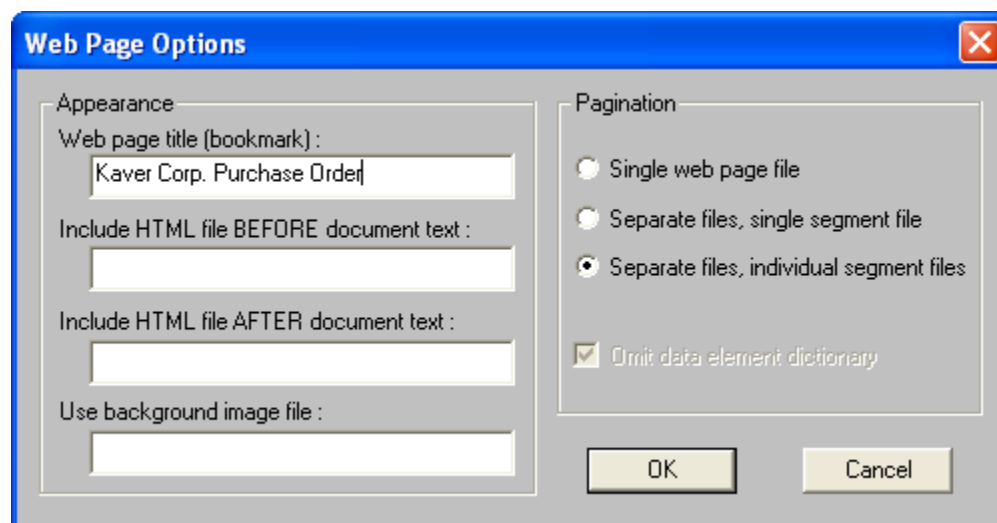
7. Click **OK**.

8. Open Explorer and double-click **CLASS850.htm** in EDISIM's **User Export\HTML850** folder.

   The entire Segments Table is in one file. There are no page numbers, but there are links to each segment's page.

9. Click the link for the **N1 Ship To** and view your customized changes.

10. Scroll down to the bottom of the page and view the links.

> **Previous** goes the previous page.
> **Next** goes to the next page.
> **Up** goes to the Segments Table.

11. Close your browser.

STOP

Comments

Review the Web Page Options dialog box.

- Appearance

  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.

  Include HTML file BEFORE document text:
  > Text or graphics, such as a logo or instructions that 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 Export\HTML Files\before.htm

  Include HTML file AFTER document text:
  > Same as #2 above, except the text or graphics will print at the bottom of each page.

  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 Export\HTML Files\background.gif.

- Pagination

  Single web page file:
  > Creates one single web page. This method creates a very large file that may open slowly in a browser. Good for printing.

  Separate files, single segment file:
  > Segment Table in one file with details of all segments in a second file.

  Separate files, individual segment files:
  > Segment Table menu file with each segment detail in its own separate file. Good for browsing.

  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**.

Test Data Generator

Create test EDI data

For more information: See **TIB_fsp_edisim_*<n.n>*_fstdg.pdf** in EDISIM's Documentation directory

Purpose

We define our enveloping and then create a streamlined "base model" of our transaction. From that, we "save as" to create individual test cases. We then create a script that makes test data files with multiple transaction sets.

This diagram shows the architecture of TDG:
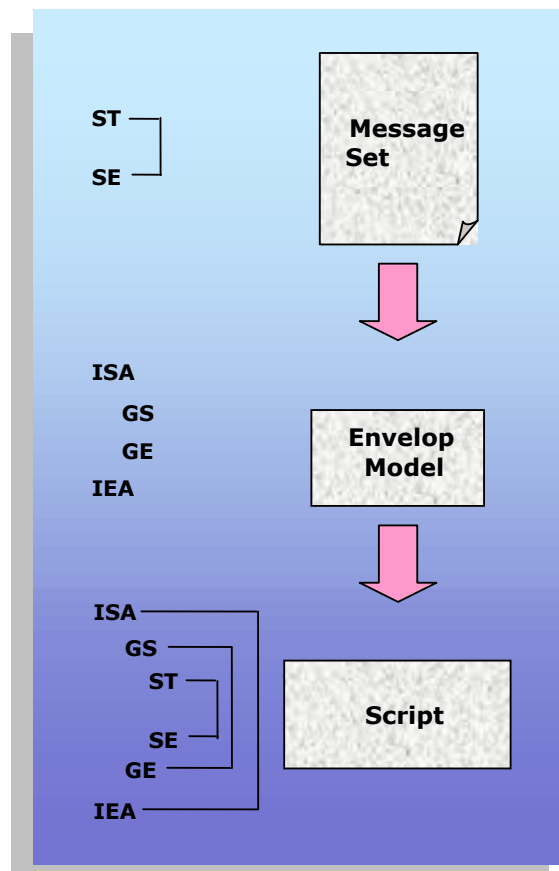
Use Test Data Generator:

- When your translator or trading partner cannot provide test data

- When systems are not yet ready

- To simulate error conditions and verify they'll be handled appropriately

- To perform high-volume (stress) testing and performance testing

- To create data that does not contain identifying information about trading partners

Features:

- Each element is preloaded with data

- Sample models are included

- Uses guidelines and standards

Setting Up Enveloping

We need to customize the enveloping so that it is appropriate for our company's inbound documents.

START

1. Open TDG with **Start | Programs | *<TIBCO_HOME>* | EDISIM | Test Data Generator**.

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

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

3. For the description, type **My Usual Inbound 4020 and later Enveloping**.

4. For Envelope Type, select:

   ```
   X12 (ASC X12 Draft Standards - ISA, IEA)
   X12 (Y2K) (ASC X12 Standards 3072 and later - ISA, IEA)
   X12 (Rep Elm) (ASC X12 Standards 4020 and later - ISA, IEA)
   EDIFACT (UN/EDIFACT - UNB, UNZ)
   EDIFACT4 (ISO 9735 Version 4 - UNB, UNZ)
   UCS (UCS/WINS - BG, EG)
   TRADACOMS (TRADACOMS 93 - STX, END)
   ```
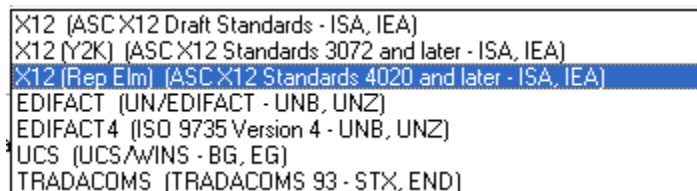
5. Choose **File | Save As**, type **X12IN** for the name, and choose **OK**.

   You now have your own copy of X12 enveloping and can use it over and over.

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

7. Click the **Ellipsis** button (three dots) at the end of the **ISA - Interchange Control Header** field:
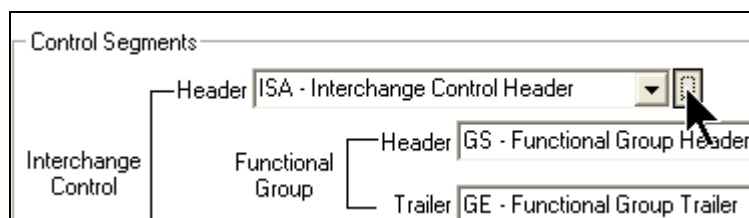
   You will see the ISA elements and can edit them.

8. Click on **I05 Interchange ID Qualifier** at position **ISA07** and look at the Test Data pane at the bottom left. This 01 is what TDG is going to put in the test data file for the ISA07.

---

9. What is 01? To find out, double-click on the Code Values folder in the bottom right pane:
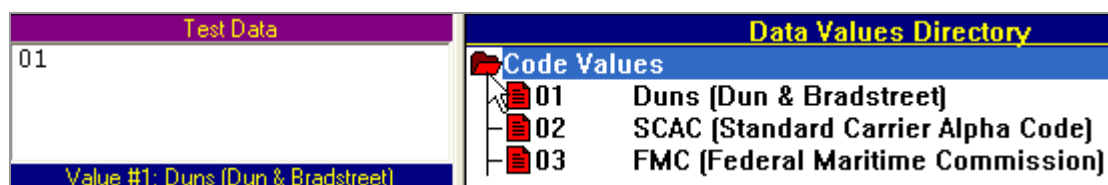
We want to use 16 - Duns Number With 4-Character Suffix.

10. Double-click on the **01** code in the Test Data pane and change it to **16**. Click **OK**.

11. Click on **I07 Interchange Sender ID** at position **ISA08**. Use your delete key to delete all the values in the Test Data pane.

12 Click the **Insert Row** toolbar button:
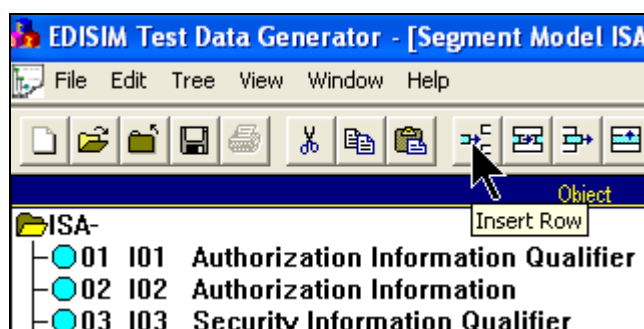
Type our DUNS+ 4 number: **1234567890123<space><space>**. Verify that you have 15 characters:
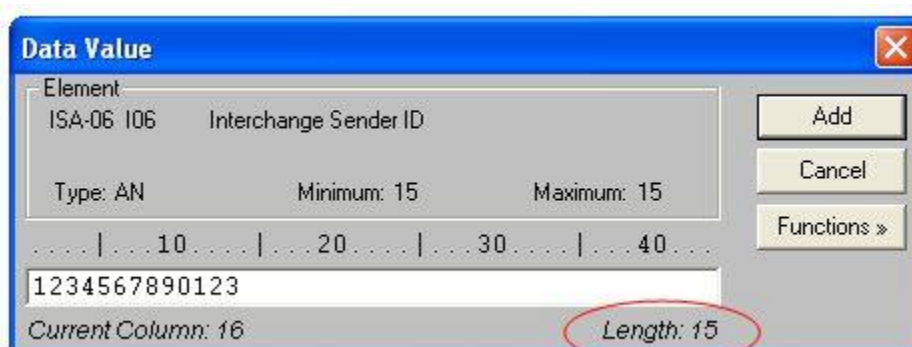
… and click **Add** and then **Cancel**.

13. Save and close the ISA.

14. Open the **GS** and delete the T code for the GS07. We always want to use X, for X12. Save and close the **GS**.

15. Save and close the envelope model.

STOP

Comments

- This enveloping model will create these segments:

```
ISA*00*          *00*          *01*9088877320000  *16*1234567890123  *070501*1114*+*00200*000000001*0*T*:!
GS*HC*908887732000*901234572000*20070501*1615*1*X*004010X098A1!
GE*1*1!
IEA*1*000000001!
```

- You will probably want another enveloping model for outbound documents.

- You probably will not have to change your enveloping models very often.

- You can change the delimiters.

Creating a Base Transaction Set Model

We will create a basic transaction set model based on our company guideline, CLASS850. We'll use it as the basis for our test case models.

START
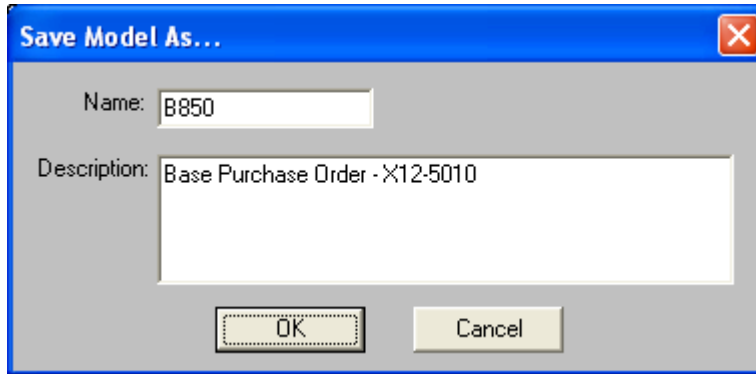
1. Choose **File | New | New Message/Set Model |** select **CLASS850** and click **OK**.

2. Maximize the Message/Set Model window. Drag the splitter bars until the screen is similar to the following example.
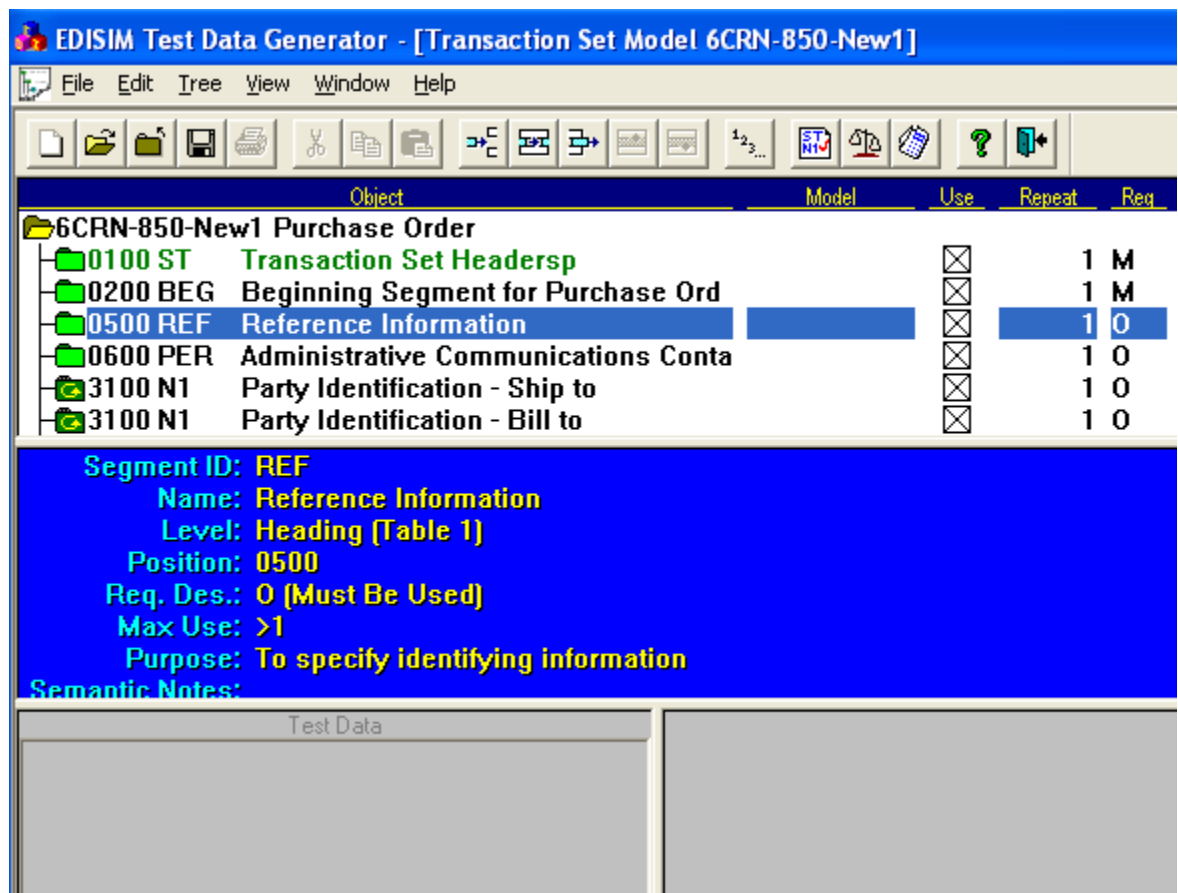
The **CLASS850** guideline items appear in the top pane. The blue pane in the middle displays information about what is highlighted at the top (in the example above, this is the REF segment at position 0500).

TDG assigns four characters to the front of a new Message/Set model's name in the title bar. These are only for identification purposes.

3. Choose **File | Save As** and fill out the box as follows.

4. Click **OK**.

STOP

Generating Data

We can create data right now. Every element already contains default values.

START

1. Click in the top pane and click the Generate Data button toolbar button [image] .

2. Select our **X12IN** Envelope Type.

3. For Output File, change the filename and path, if you'd like.
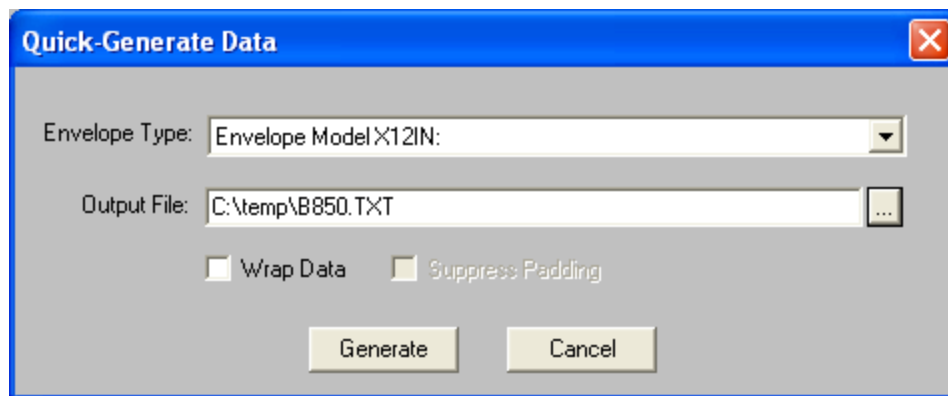
4. Click **Generate**.

   Tip: This dialog box contains the path and filename of the output file. Leave it open until you find the file.

5. Open **CLASS850.txt** from Windows Explorer and view the test data.
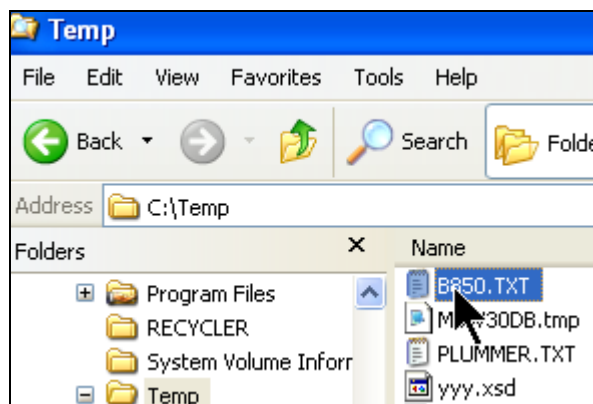
   You will notice many changes that you'd like to make to the file. We edit our model to do that.

6. Close Explorer, return to TDG, and close the generation box.

STOP

Comments

- Wrap Data:

  – Places all of the data on one line, with no CR/LFs.

  – Suppress padding.

Viewing the Data

Let's look at the data that TDG is using in our transaction set model.

START

1. Loops are symbolized by dark green folders with a looping arrow. Double-click on the **N1 Name - Ship To** loop in the top pane.

   The segments in our customized N1 loop display.

2. Segments are symbolized by bright green folders. Double-click on the **N1 Party Identification - Ship to** segment at 3100 under the loop.

3. Elements are symbolized by light blue circles. Click on **N101**, element 98.

   Look at the Test Data pane; it is using the code set of ST, which you created in Standards Editor.

4. Click on **N102**, element 93 and look at the Test Data pane.

   Default values have been provided by Test Data Generator, since the guideline had no code sets or application values in those elements.

5. Click on **N103**, element 66.

   TDG uses the 9 and 92 code values from your code set.

6. Open the **N3** segment at position 3300, and click on N301, element 166.

   This data came from your RECVDOC application value that you created in Standards Editor.

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.

- 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 or business rules.

Adding New Values

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

START

1. Open the **BEG** segment at **0200**.

2. Click on the **BEG01**, element 353, and look at the values in the Test Data pane:

3. Let's add another code value. Double-click on the red **Code Values** folder in the bottom right pane to open it.

4. Double-click on code **00 Original** to add it to the test data list.

   Note: You have to double-click directly on the code or the description. You can also drag a code into the list.

5. Click on **00** in the Test Data pane and click the Move Row Up toolbar button until 00 is at the top of the list.

6. Close the **BEG** segment by double-clicking it.

7. Click on **Name - Ship To N102**, element 93.
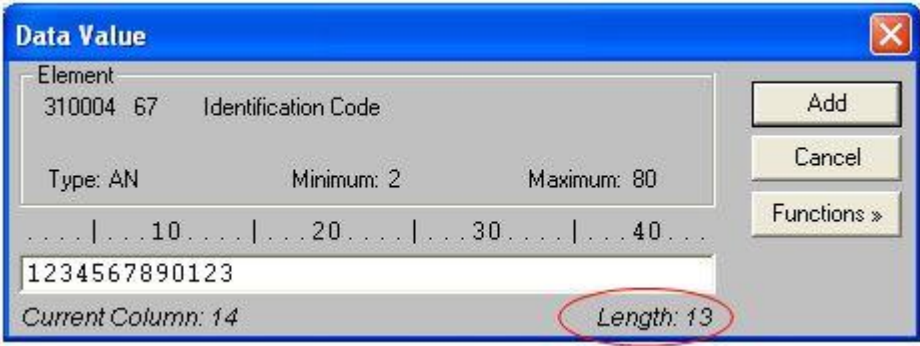
   We want to change the **Ship To** customer list.

8. Select the list of names in the Test Data pane and press the *Delete* key.

9. Click in the Test Data pane and click the **Insert Row** toolbar button .

10. In the Data Value box, type **ALLY CORPORATION** and click **Add**.

    This adds the name to the Test Data pane. The box remains open so that you can add another value.

11. Add **DIAMOND PROCESSORS** and click **Add**.

12. Click **Cancel** to close the Data Value box.

13. Click on **N104**.

14. Click on the top value in the Test Data pane, click the Insert Row toolbar button and type our DUNS + 4 number **1234567890123**. Verify that you have 13 characters.

15. Click **Add** and then **Cancel**.

16. Close the N1 loop.

17. Save.

STOP

Comments

▪ The toolbars and menus operate according to where your cursor is placed.

▪ You can copy a list of values from elsewhere and past them into the Data Values pane with *Ctrl*-**v**.

Using Functions to Provide Values

A function lets you automatically supply a value or part of a value. The Functions folder in the bottom right pane contains every function in TDG, in alphabetic order.

Let's set up two functions:

- One to create a date every time we generate test data from this model.

- One to count the number of times the PO1 occurs in this transaction set.

START

We will use the Date function to create a shipping date:

1. Open the **BEG** segment at position 0200 and click on element 373 at **BEG05**.

2. Look at the Test Data pane.

   The {DATE} function inserts today's date each time you generate data with this model.

3. Add 10 days to today's date by double-clicking on {DATE}.

   Click after the **E** in DATE, and type **+10** so that it looks like this:
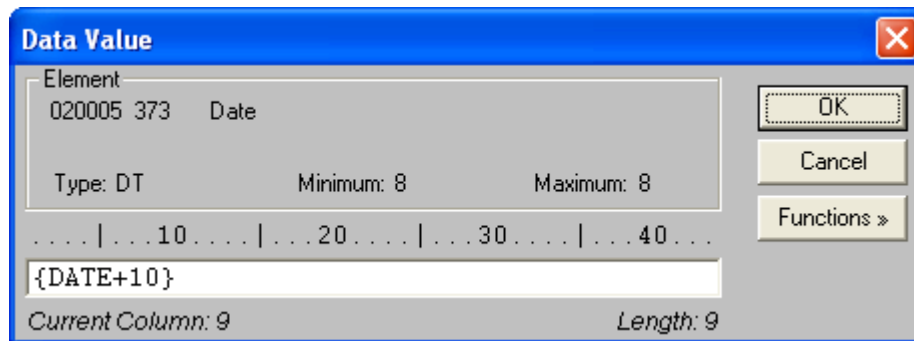
4. Click **OK**. Click **OK** again at the message that says "Date required" since we do not want to use a literal date here.

5. Close the **BEG** segment.
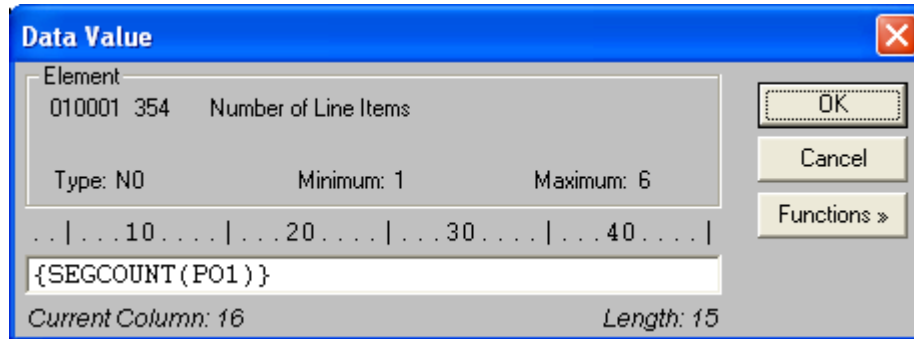
   Now we will create a function to count the number of PO1 segments in the data.

1. At the bottom of the transaction, open the **CTT** loop and the CTT segment and click on the CTT01.

2. Delete the values. We're going to use a function instead.

3. Open the red **Functions** folder at the bottom right.

4. Double-click on **SEGCOUNT(seg)}** to add it to the Test Data pane.

   Lowercase text in a function is a placeholder for text that you need provide.

5. Double-click on {SEGCOUNT(seg)} in the Test Data pane and replace (seg) with (PO1) so the function looks like {SEGCOUNT(PO1)}. Be sure to use the alpha character "O" and not zero.

6. Click **OK**.

7. Close the CTT loop.

8. Save.

STOP

Trimming down the Output
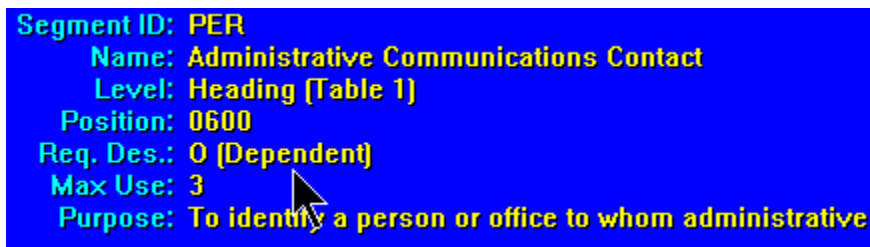
You will want to make most test cases as short as possible, so we need to create a base model that is compact. We will then do "save-as" from this base and then modify to create the test cases.

"**Generating**" an item means putting it in the test data. By default, TDG generates items that are used in the guideline.

"**Ungenerating**" an item means not putting it in the test data. By default, TDG ungenerates items that are marked not used in the guideline.

The Use column in TDG's top pane shows whether an item will be generated. You can override these settings for an item by clicking on its Use box. TDG allows you to ungenerate mandatory and must use items, but it warns you.

Always look at the **Req. Des**. line in the blue pane before ungenerating something. The first letter is the X12 requirement (O in the example below) and it may be followed by your company's setting in parentheses:

Since this is our slimmed-down base model, we want to mark everything unused that we don't need.

START

1. Open the **PO1 loop** and the **PO1 segment** and look at the faded text for elements at PO108 through PO125.

   Since they were marked unused in Standards Editor, Test Data Generator will not generate them.

2. Close the PO1 segment (not the loop).

3. Open the **Ship to N1 loop** and ungenerate all segments from the **N2** through the **PKG** at 4200 by clicking in their Use boxes:

Data for these segments will not be generated.

Shortcut: the space bar toggles the Use column on and off. By using the keyboard down-arrow and the space bar, you can quickly mark items unused.

4. Do the same with the segments in the **Bill to N1 loop**.

5. In the rest of the transaction, only generate these segments:

   PO1
   PID
   CTT
   SE

6. Open the **REF** segment at 0500 in Table 1 and click REF02. In the center blue pane, the Syntax note shows that you must use the REF02 or the REF03.

   Ungenerate the **REF03**.

The 'X' requirement means there is a syntax rule attached.

7. Double-click on composite **C040** at REF04.

   A composite is symbolized by a blue-green folder with a 'C' on it.

   Notice that its subelements are numbered within the composite.

8. Ungenerate the **C040**. You do not have to ungenerate the subelements separately.

9. Save.

STOP

Changing the Repeat

So far, our test data file will have one of each segment and loop. We can change the repeat in TDG.

START

1. Click on the Repeat column for the **PO1 loop**.

2. Type **3** and press *Enter*.

   The data will now include three PO1 loops.

3. To see which values will be included, open the **PO1 loop** and the **PO1 segment**, click on **PO105**, and look at the Test Data pane.

   The first repetition of the PO1 segment will AP, the second repetition will use CP, and the third will use PC. If we had a repeat of 4, the fourth repetition would cycle back to the top of the list and again use the AP code value.

STOP

Copying Values from an External Source

In this exercise, we will copy company names from a word processor file. This is an easy way to add multiple names to our value list. Since we may want to use this list elsewhere, we will save an external model of the element that uses the company names.

START

1. Open the **Ship to N102**.

2. Using Windows Explorer, open **Customer.txt** in EDISIM's Samples directory.

3. Copy all of the names.

4. Close Customer.txt and Explorer.

5. Return to TDG and click in the Test Data pane.

6. Choose **Edit | Paste**.

    Our complete list of customer names is added to the values already in the Test Data pane.

STOP

Creating an External Model

Optional Exercise

You can customize a loop, group, segment, or element, and then save it as an *external model* that can be attached to any item with the same ID. You can make changes to the model that will automatically update wherever it is used. This is a time saver and it keeps your data consistent.

We will create an external model using our list of Ship To names.

START

1. Click the Ship to N102 and choose **File | Save Externally (Name).**

   We are going to make this element a model to use again.

2. Type SHIPTO for the model name and replace the description text with Element 93 Ship To Name List, then click OK.
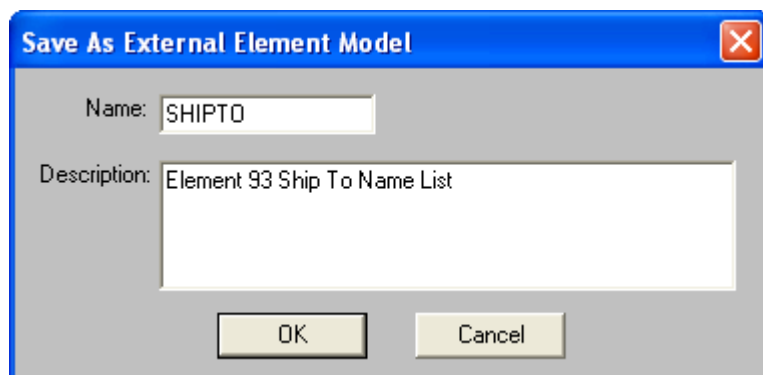
3. We want to actually attach the model here, even though we already have the values that we want to use. The advantage of doing this is that the list of names will be automatically updated if the model is changed somewhere else.

   Choose **Edit | Replace** and choose 93-SHIPTO and press OK.
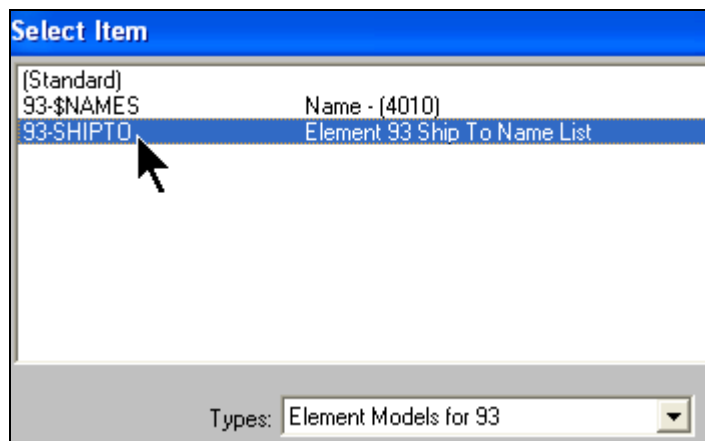
The model replaces the original data list.

SHIPTO appears under the Model column.

This element and the bottom two panes now have a shaded background, indicating that this is a model.
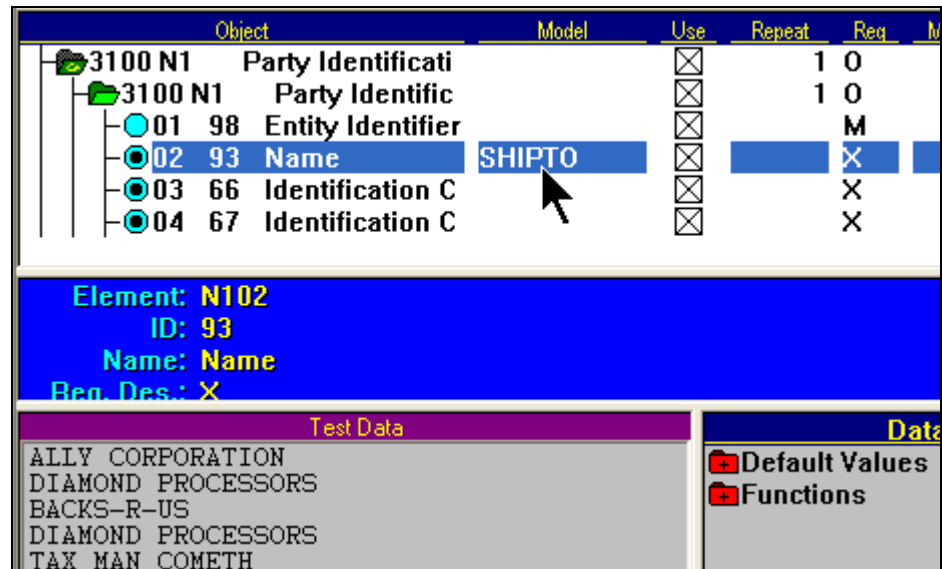
4. We want to re-use this same list of values in the Bill To loop. Go to the Bill to N102 and choose **Edit | Replace** | **93-SHIPTO | OK.**

   You can see SHIPTO in the Model column, and the shaded bottom panes.

5. Save.

STOP

Comments

- We saved a specific definition of element 93. The SHIPTO model is available to any element 93 in any message/set model for any guideline.

- Good practice: any time you customize a loop, segment, or element and think you may need it again somewhere else, save it as an external model.

- You can export external models and give them to other EDISIM users.

- Reasons to create external models:

  – Saves effort

  – Assures consistency

  – Improves database efficiency

- If you want to edit the values in a model, you will have to either:

  – Double-click on the SHIPTO model name in the Model column (to edit the model itself - this is a global change affecting all places where the model is attached).

  Or …

  – Click on the model and then choose **File | Save Locally** to detach the model so that you can edit as usual.

Changing an External Model

Optional Exercise

Let's imagine that we just received notice of a new customer and need to add a name to the model.

START

1. With **93** still selected, click in the **Test Data** pane, and choose **Edit | Insert**.

   Nothing happens! As a precaution, TDG will not allow you to add a new name to the model this way. You must edit the model, so your changes will be made wherever the model is attached.

2. Double-click on **SHIPTO** in the model column.

   The model opens. Notice the title bar shows the model name.

3. Click the top name in the Test Data pane, and choose **Edit | Insert**. In the Data Value dialog box, type **LIGHTHOUSE GIFTS**. Click **Add** and then **Cancel**.

4. Click the **Save** button to save the change to the SHIPTO model.

5. Choose **File | Close** to close the SHIPTO model.

   Look in the Test Data pane to see the added name.

6. The change affects all places where the model is used.

7. Go to the **Ship to N102** and look at the values.

8. Lighthouse Gifts is there too, since it uses the same model.

STOP

Detaching a Model

Optional Exercise

Now, that our Ship To and Bill To customer lists are updated, we have decided to customize only the Bill To list. We do not want it to affect the Ship To list.

For this reason, we want to detach the Bill To N102 element from its model. Once the model is detached, any updates to the model will not affect it. Likewise, any updates to the Bill To N102 will not affect other places that still use the model.

START

1. Click the **Bill To N102** and choose **File | Save Locally (Embed)**.

   This detaches the model from this element. The shading is gone. From now on, any changes here will not affect other elements, 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 **SOFTWARE SCENE**, and then close the box.

4. Look at the values for the **Ship To N102**.

   SOFTWARE SCENE was not added here.

5. Save.

STOP

Correcting Problems

Use Analyzer to find errors, and then correct the TDG enveloping and transaction models that created the errors.

When satisfied with the data being created, you can use it to create test cases.

START

1. Click in TDG's top pane and click the [1²₃] toolbar button.

    Use X12IN for enveloping and specify an output file for test data.
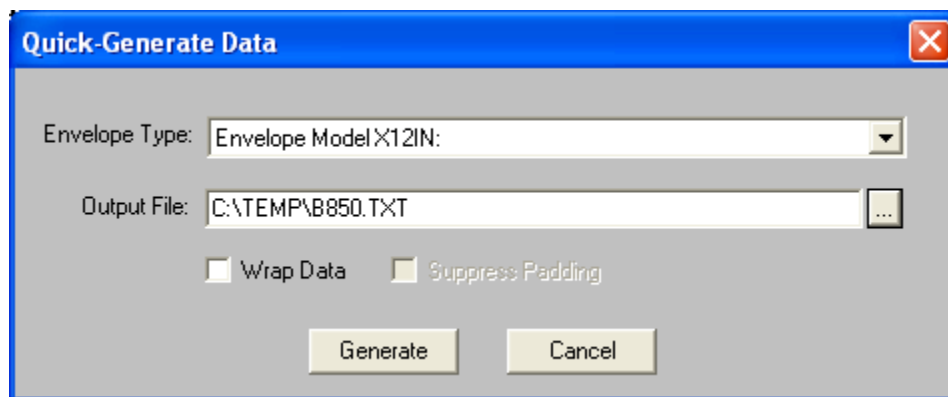
    Choose **Generate**.

    Click **OK** when generation completes.

2. Open Analyzer with the [STD] toolbar button.

    Choose **File | Open** and open the EDI file that you just created.

    Choose **CLASS850** for the standard.

3. When the analysis completes, you will see this error:

---
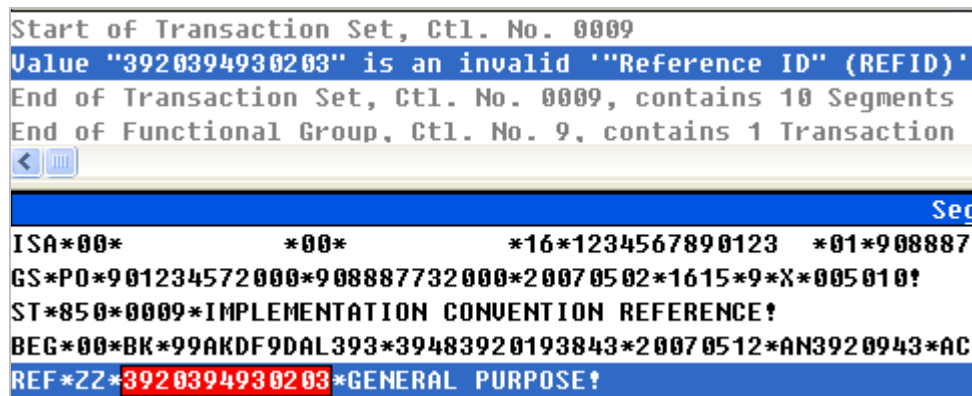
4. What value is expected there? If we used good notes, we will be able to find out by looking in TDG.

Since this is a transaction set error, return to your B850 model in TDG and open the **REF** right after the ST and BEG segments.

Click on **REF02** and read the note in the blue box.


```
6CRN-850-B850 Base Purchas
  0100 ST    Transaction Set        ☒    1 M    1
  0200 BEG   Beginning Segm         ☒    1 M    1
  0500 REF   Reference Inform       ☒    1 O    >1
    01  128  Reference Identifi     ☒      M        ID
    02  127  Reference Identifi     ☒      X        AN
    03  352  Description            ☒      X        AN
```

```
    Syntax:  R0203: At least one of REF02 or REF03 is required.
Description:  Reference information as defined for a particular Transaction Set or
              the Reference Identification Qualifier
Level 1 Notes:  The required value for this element starts with ABC followed by 2 al
              and 3 numeric characters.

Example: ABCAB123
```

5. Replace the bad values in the Test Data pane with **ABCAB999**.

6. Correct any other problems found by Analyzer.

If there were errors in the ISA or GS, open the enveloping model X12IN and correct them.

7. Save, regenerate, and analyze the data to be sure it is correct now.

STOP

Creating a Test Case

With our clean base model open, we can now save it to a new name and create data to match a specific test case or a different base model.

In this exercise, we are creating a test case that omits a mandatory element and inserts an out-of-place segment.

START

1. With B850 open and all changes saved, use **File | Save As**.

2. Name the new model **T850-1** with description X12-5010 850 - **Mandatory BEG03 missing, misplaced CUR**.

   At Kaver Corp., our convention is to use T in front of the name to indicate that it is a test case rather than a base model.

3. Click **OK**.

4. Ungenerate the **BEG03**.

   At the warning message, click **Yes**.

5. Click on the closed **REF** segment at 0500 (DO NOT DISPLAY ITS ELEMENTS.)

   We are going to insert an out of place segment below the REF.

6. Choose **Edit | Insert** and select the **CUR** at 0400.

   Click the **Below** button.
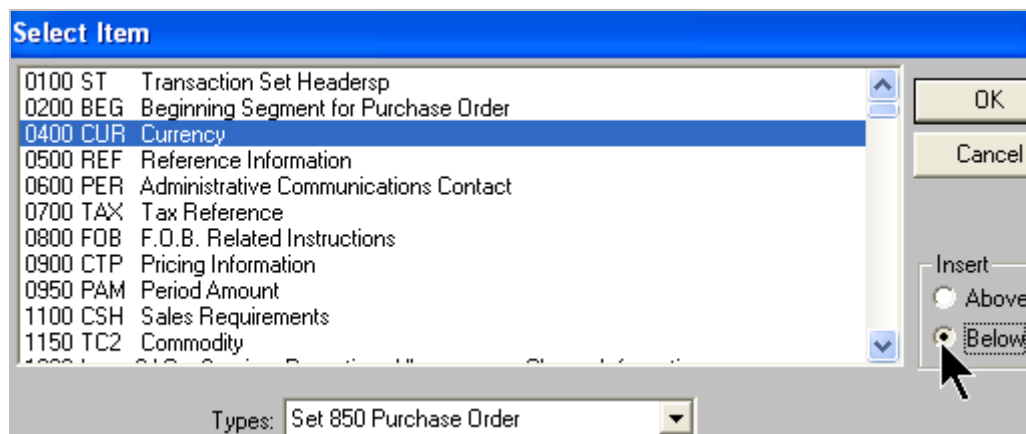
7. Click **OK**.

   Notice the CUR below the REF. It belongs above the REF, if it is used at all.

8. Save.

---

9. Generate data and check it in Analyzer.

   You should see a fatal error on the BEG03 and errors around the misplaced CUR.

   You now have data for your first test case.

10. Close **T850-1** test case.

STOP

Creating a Second Test Case

We will now create our second test case from our base model B850.

START
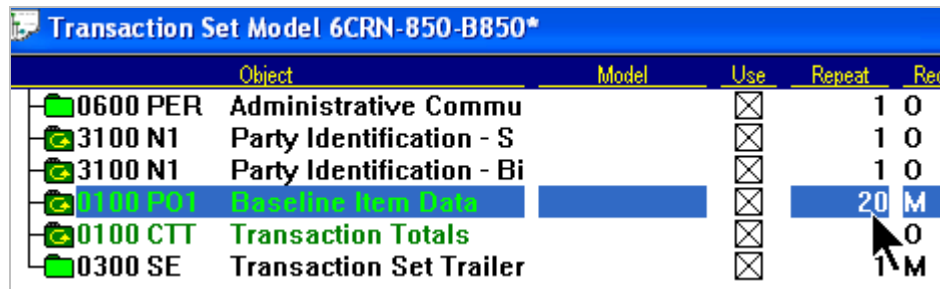
1. Choose **File | Open**.

2. At the bottom, choose Object Types **Message/Set Models**.

3. From the list of transaction models, open **B850**:
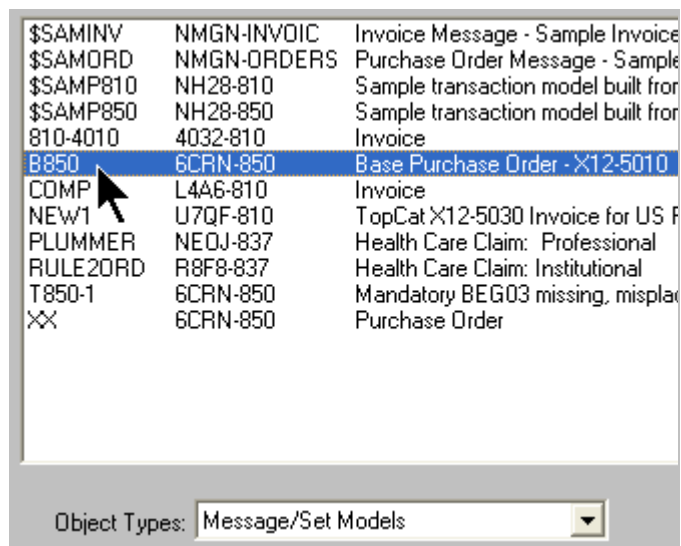
4. **Save as** with this information.
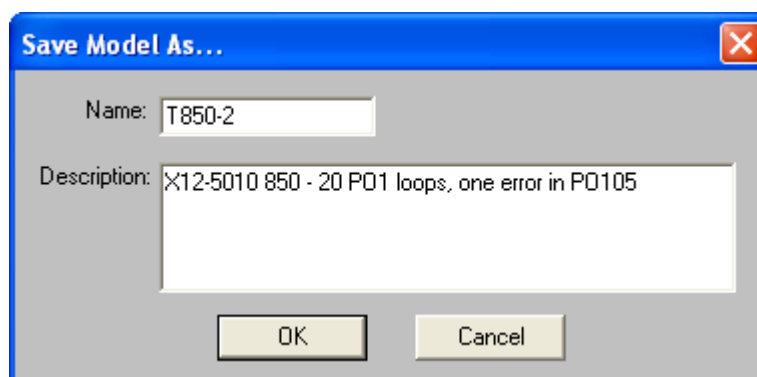
5. Change the Repeat for the **PO1** loop to **20**:

6. Open the PO1 loop and PO1 segment and click on **PO105**.

7. Double-click on the **Code Values** folder in the Data Values Directory.

   Notice the local code **PC**. A local code is symbolized by a red page with an 'L' on it.

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

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

9. Let's add a code to force an error. Double-click on **CA**, an unused code to add it to the bottom of the Test Data list.

   Code CA will be in the 4th repetition of the PO1 loop.

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

11. Save and close T850-2.

STOP

An Introduction to EDISIM (X12 Standards)   Test Data Generator • 96

Creating a File with Multiple Transaction Sets
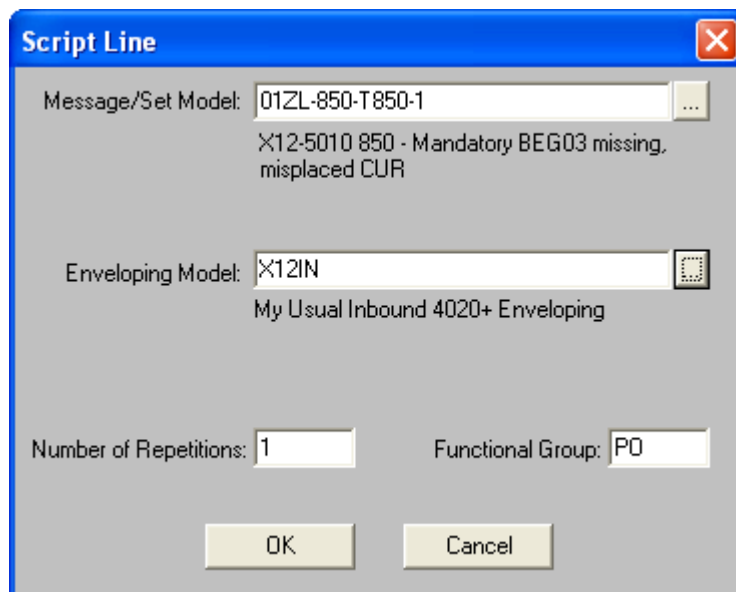
When you want to have more than one transaction in a file, create a "script." By using a script, you can have any number and combinations of transaction sets in one file.

START

To create a script that uses one transaction model:

1. Choose **File | New | New Script**. Maximize the script.

2. Insert a line into the script with **Edit | Insert**.

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

4. Select **T850-1**, your transaction set test case.

5. Click the **Ellipsis** button after the **Enveloping Model** field and select **X12IN**, your customized enveloping model.

The Script Line dialog box now looks like this:

6. Click **OK**.

   The Script dialog box now lists one line.

New script line →
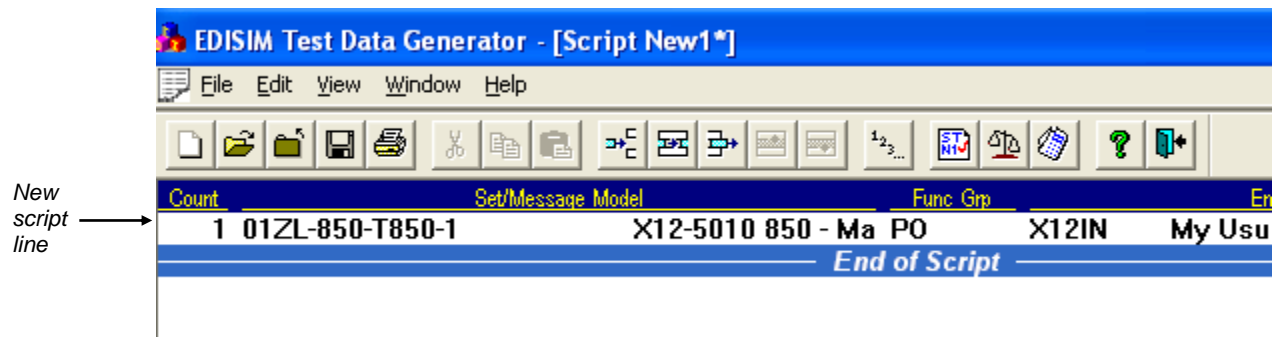
7. Click on the **1** in the count column at the top left and change it to **3**.
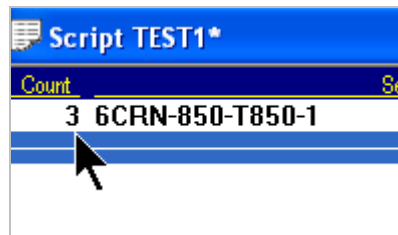
8. Choose **File | Save** and fill out the box like this:
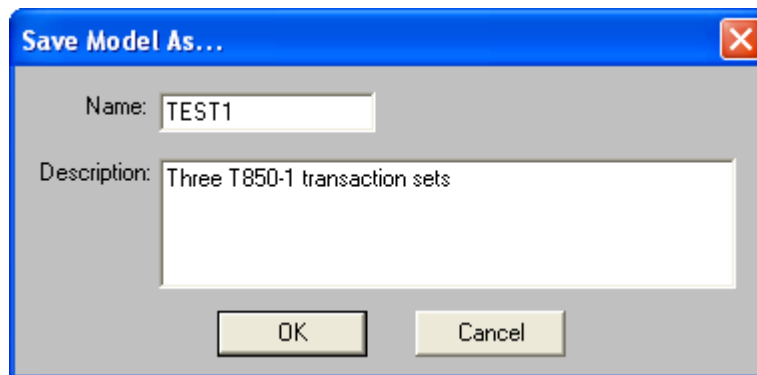
9. Click **OK**.

10. Time to generate the file. Click .

11. Insert a path and filename for the output and click **Generate**.
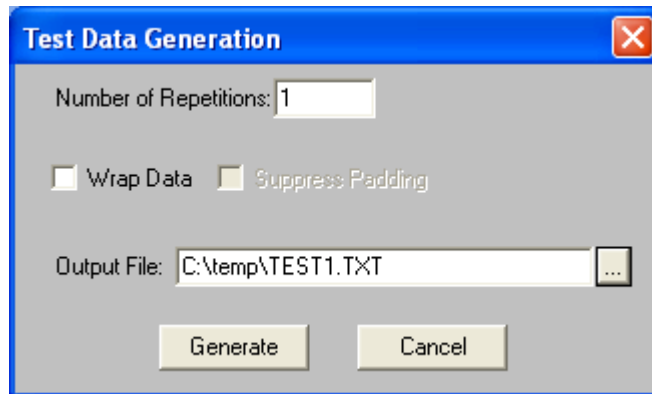
12 Go to Windows Explorer, open the file, and search for **ST***, the beginning of each transaction set.

13. Back in TDG, close the generate box.

To add another transaction model to the script:

1. Save as this:
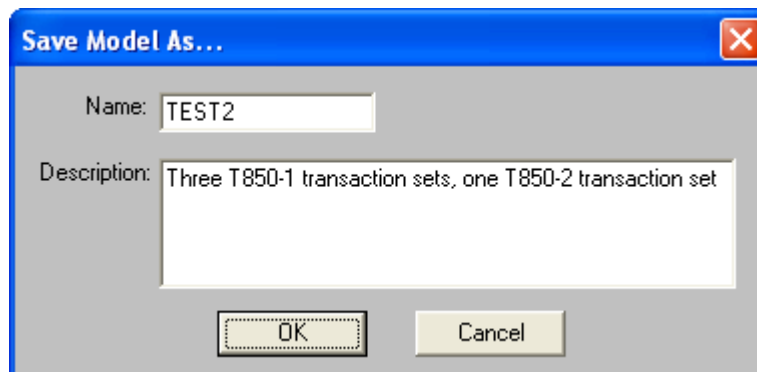
2. Click the **End of Script** line, choose **Edit | Insert** and add **T850-2**.

   Use **X12IN** enveloping and choose **OK**.

4. Save and generate data.

   It will contain 3 transaction sets from T850-1 followed by 1 transaction set from T850-2:

---

5. Close TDG.

6. Look at the new data file.

STOP

Comments

- Test Data Generation dialog box:

    – Number of Repetitions:

        Cycle the entire script, including interchange header and trailer.

        Cycle an individual script line.

        Cycle several script lines.

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

Viewing the Test Data

Data created by the script TEST1 will have the following structure.

```
ISA*00*          *00*          *16*1234567890123  *01*9088877320000  *070502*1142*+*00200*000000015*0*T*:!
GS*PO*901234572000*908887732000*20070502*1615*15*X*005010!
ST*850*0015*IMPLEMENTATION CONVENTION REFERENCE!
BEG*00*BK**39483920193843*20070512*AN3920943*AC*IBM*02*AE*02*BA!
.
.
.
SE*11*0015!
GE*1*15!
IEA*1*000000015!
ISA*00*          *00*          *16*1234567890123  *01*9028312210000  *070502*1142*+*00200*000000016*0*T*:!
GS*PO*908473457000*902831221000*20070502*0718*16*X*005010!
ST*850*0016*IMPLEMENTATION CONVENTION REFERENCE!
.
.
.
95*50*HB*400*QD*GENERAL PURPOSE!
SE*11*0016!
GE*1*16!
IEA*1*000000016!
ISA*00*          *00*          *16*1234567890123  *01*9083733330000  *070502*1142*+*00200*000000017*0*T*:!
GS*PO*902837382000*908373333000*20070502*1952*17*X*005010!
ST*850*0017*IMPLEMENTATION CONVENTION REFERENCE!
BEG*10*DS**394829041*20070512*9392039*AC*INR*09*AE*09*PP!
.
.
.
SE*11*0017!
GE*1*17!
IEA*1*000000017!
ISA*00*          *00*          *16*1234567890123  *01*9088877320000  *070502*1142*+*00200*000000018*0*T*:!
GS*PO*901234572000*908887732000*20070502*1615*18*X*005010!
ST*850*0018*IMPLEMENTATION CONVENTION REFERENCE!
.
.
.
<20 PO1 - PID segments>
.
.
.
SE*48*0018!
GE*1*18!
IEA*1*000000018!
```

Analyzer

See if EDI has errors

For more information: See **TIB_fsp_edisim_ *<n.n>*_fsanalyze.pdf** and **BusinessRules.pdf** in EDISIM's Documentation directory.

Purpose

The purpose of this exercise is to check EDI data to see if it complies with X12 requirements and our own requirements.

Choosing a File and a Standard

First we check the EDI file for compliance to a published standard.

START

1. Choose **Start | Programs | *<TIBCO_HOME>* | EDISIM | Analyzer**.

2. Choose **File | Open**.

3. Go to the **EDISIM's Samples** folder.

4. Click on **testpo-5010-A.txt**, and click **Open**.

   Analyzer prompts you to select a standard (or guideline). The list is sorted by the Standard column.

5. Single click on **X12-5010**.

6. Click **OK** to start the analysis and click **OK** again when analysis completes.

STOP

Comments

- Sources of data for Analyzer:

  - A trading partner test or rejected transmission inbound.

  - A test from the translator for an outbound transaction.

- You can drag an EDI file from Explorer and drop it into the Analyzer window.

Checking the Diagnostics

1. Maximize the Analyzer window.

2. Press the *Up Arrow* to go to the top of the top pane. These are diagnostic messages and errors.

3. Press the *Down Arrow* through the diagnostics, while noticing how Analyzer highlights the corresponding segment in the bottom pane.

4. When you reach a blue error message, notice how the bottom pane displays the erroneous data in red.

5. Choose **File | Close**.

Comments

- Each message in this exercise is from the X12-5010 perspective.

- Diagnostic messages are explained in the Help file under the Contents tab.

- You can adjust the splitter bar between the panes.

Analyzing Against Our Guideline CLASS850

START

1. Choose **File | Open**.

2. Select **testpo-5010-A.txt** again, and click **Open**.

3. Select **CLASS850** and click **OK**.

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

5. Press the *Up Arrow* to go to the top of the Diagnostics pane.

6. Press the *Down Arrow* through the diagnostics and review the errors that Analyzer found (in blue text).

7. In the Diagnostics pane, right-click on the message "**Conditional Element Relation violated: If either PER07 or PER08 is present, then the other is required.**" Note that it is a #10621 diagnostic message.
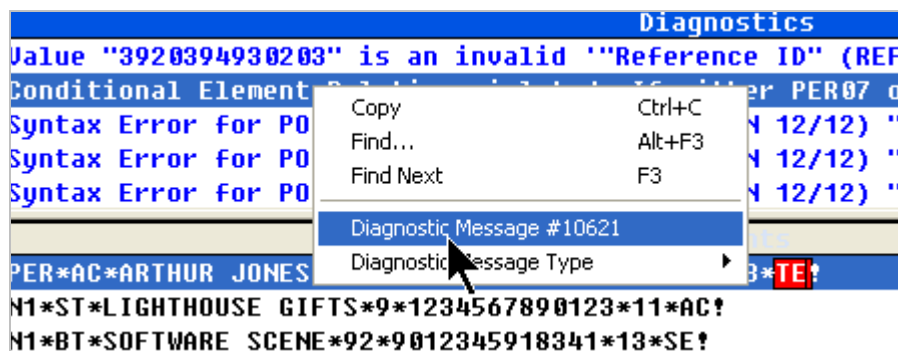
8. Determine what caused this error message. Press **F1** to open Help, click the Search tab, type **10621** and click **Display**.

9. Close Help.

STOP

Comments

- Diagnostic message explanations are also located in the Help file under the Contents tab and in anamsgs.rtf in EDISIM's Bin directory.

- Each message in this exercise is from the CLASS850 perspective.

Viewing Analysis Options

1. Choose **Options | Analyzer Profile**. The Analyzer Profile dialog box appears. Leave the defaults selected.
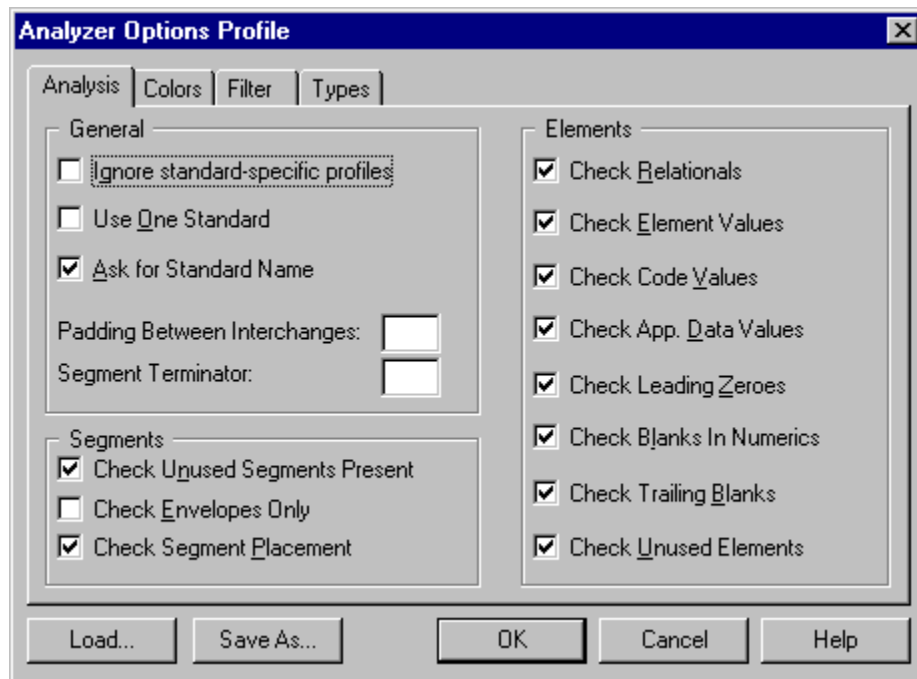
2. Press **F1** or click the **Help** button at the bottom right of the dialog box. Click on the **Index** tab, type **Analysis Options**, and click **Display**.

   All of the options are described in Help.

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

4. Click **OK**.

STOP

Changing Diagnostic Message Colors

Optional Exercise

Analyzer's diagnostic messages have pre-assigned colors that represent their type. You can change the colors.

START

1. Before changing the display of a diagnostic message, let's determine its type.

   Right-click any dark gray message in the top pane and choose **Diagnostic Message Type**.

   The submenu shows that it is an Informational message.
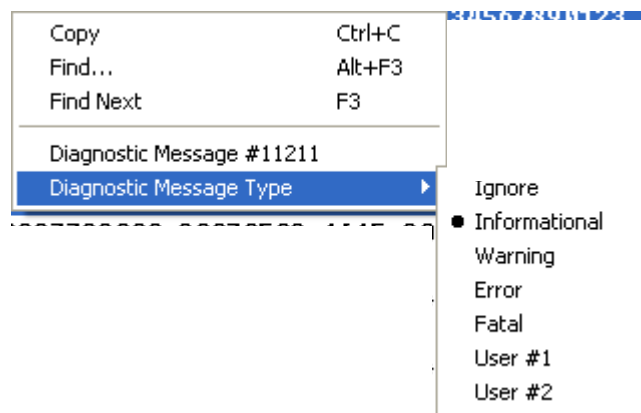
2. Now change the color of the Informational type messages. Choose **Options | Analyzer Profile | Colors** tab.

Types assigned by Analyzer

Types you can assign

---
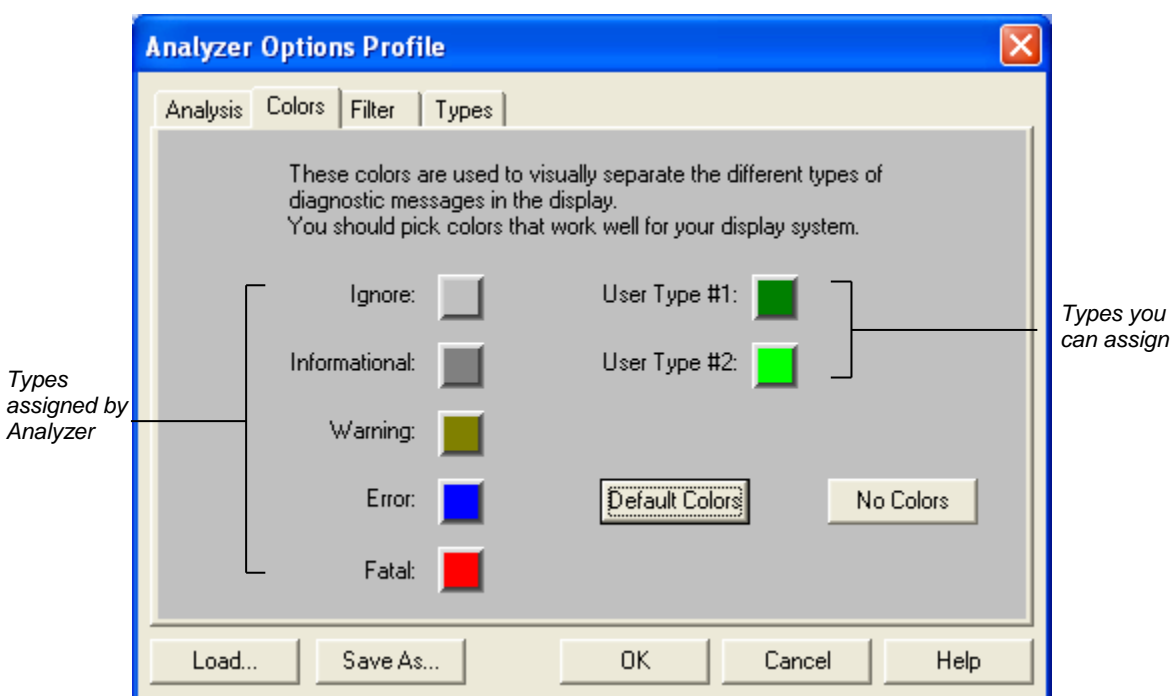
3. Click on the dark gray square beside Informational.

4. In the Color selection box, choose purple and click **OK** twice.

   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 diagnostics 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 value length violations and want them to appear in their own color.

START

1. Right click on **Syntax Error for PO101 (D.E. 350) at col. 5 (AN 12/12) "10051": Value too short**, and note that its number is #10626.

2. Select **Diagnostic Message Type** and change it to **User #1**.

3. The message is now green, the color of User 1 messages. All other #10626 errors are also User #1 now.

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

5. Page down until you see **#10626** (the message we just changed). Notice that the message shows a green bar and User #1.

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

   A popup menu displays the types. This menu is another way of changing a diagnostic message type. Leave this message number as User #1.
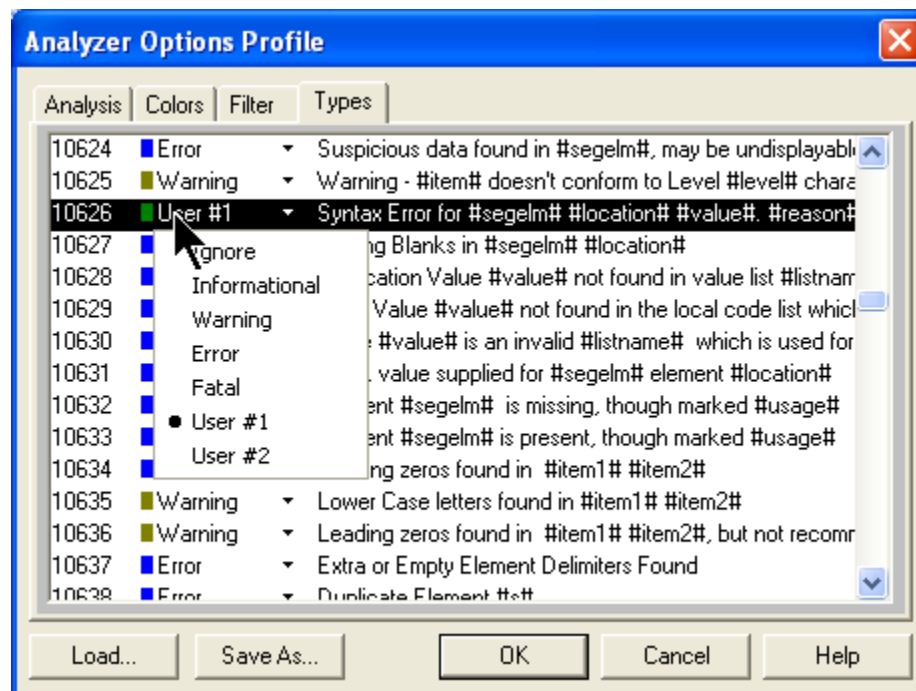
STOP

Comments

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

Type	Default Color	Default Meaning
Ignore	Light gray	For your use.
Informational	Dark gray	Non-error messages such as the start of a transaction set.
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.

Filtering Diagnostic Messages

Optional Exercise

You can hide or "filter" diagnostic message types. Filtering lets you view or print only the message types that you choose.

In this exercise, we stop displaying:

- Messages about unused segments or elements being present (like the CUR segment in our EDI file). We ignore these items and don't want to flag them as errors.

- Informational messages.

START

1. Choose **Options | Analyzer Profile** and unselect **Check Unused Segments Present** and **Check Unused Elements**.
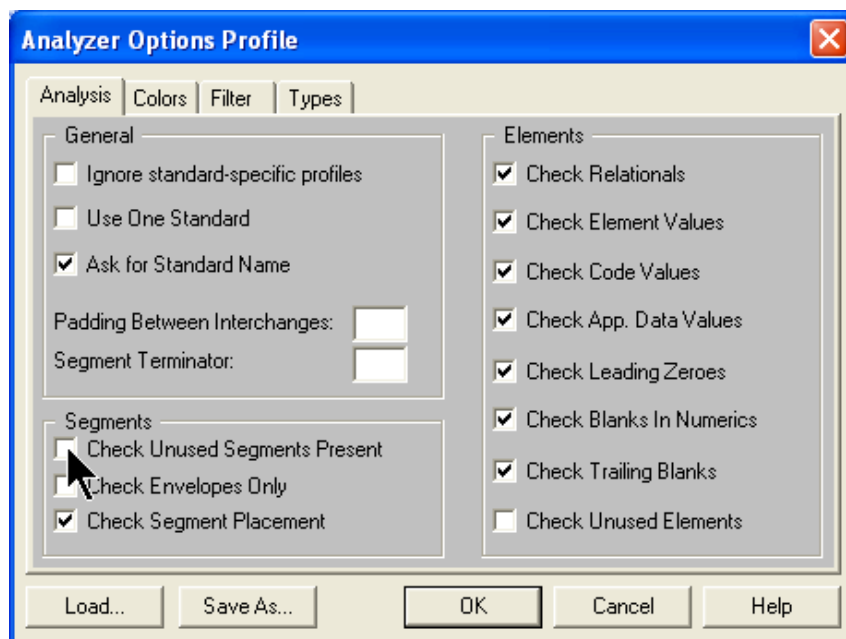
2. Go to the Filter tab and clear the **Informational** check box.

3. Click **OK**.

4. Reanalyze.

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.

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. Choose **Options | Save Profile**.

2. Save it as **usual** in EDISIM's **User Files\Profiles** directory.

   The file type will be .apf.

   If you used this profile for most of your files, you could select the check box at the bottom to save your profile as the default.

   This profile will remain in effect until you close Analyzer. The next time you open Analyzer and want to use the saved profile, choose **Options | Load Profile** and select usual.apf.

3. 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.

Printing Analyzer Reports

You can print the diagnostics and segments to a file or to a printer.

Note: Your computer must have a printer defined.

START

1. Choose **File | Print to File**.

   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.
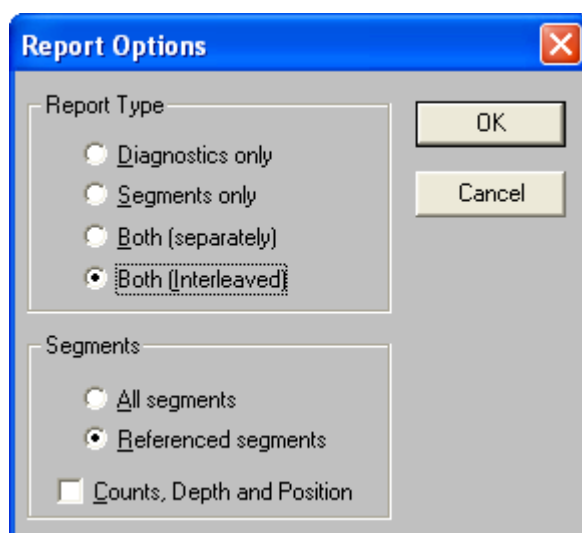
2. Click **OK**.

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

3. Click **OK**.

   The Print Text File dialog box appears.

4. Go to EDISIM's **User Files** directory and create a new directory **Analyzer_Reports**.

5. Save the report as **Diagmsg.txt** in **Analyzer_Reports**.

6. Open Windows Explorer and open **Diagmsg.txt**. View the diagnostic messages, then close the file.

STOP

Comments

- For details on report options, Go to **Help | Index** and search for **Report Options box**.

Using DocStarter

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

Creating a DocStarter Collection File

Let's create a DocStarter Collection file that contains two files.

START

1. From Analyzer, open **testpo.txt in** EDISIM's Samples directory.

2. In the Select Standard dialog box, select **X12-4010**, choose **Use for all occurrences of 004010**, and click **OK**.

   When creating a new guideline with DocStarter, always check against a published standard.
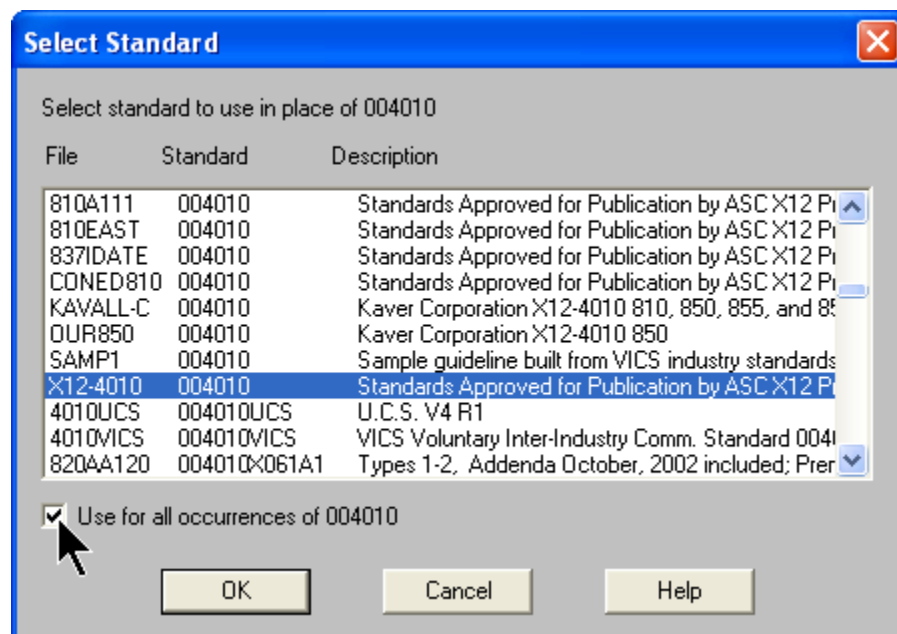
3. View the diagnostics to see if you want to base the guideline on this data.

---

4. Choose **File | DocStarter Collection**.
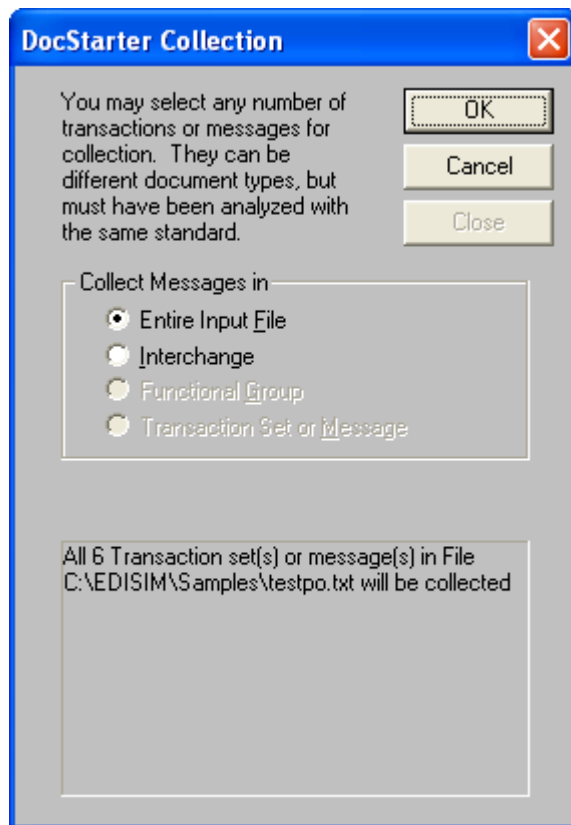
5. Leave the default setting of **Entire Input File** selected.

6. Click **OK**.

7. Create a new folder named **COL Files** in EDISIM's User Files folder.

8. Save the file **testpo.COL** and click **Save**.

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

---

9. When **Collection Complete** displays, click **Close**.

10. To add more data to the same DocStarter collection that we just created, repeat steps above, except in step 1 select the file **testpo1.txt**. Save to the same COL filename and answer Yes when asked about replacing testpo.COL.

Save to the same collection file
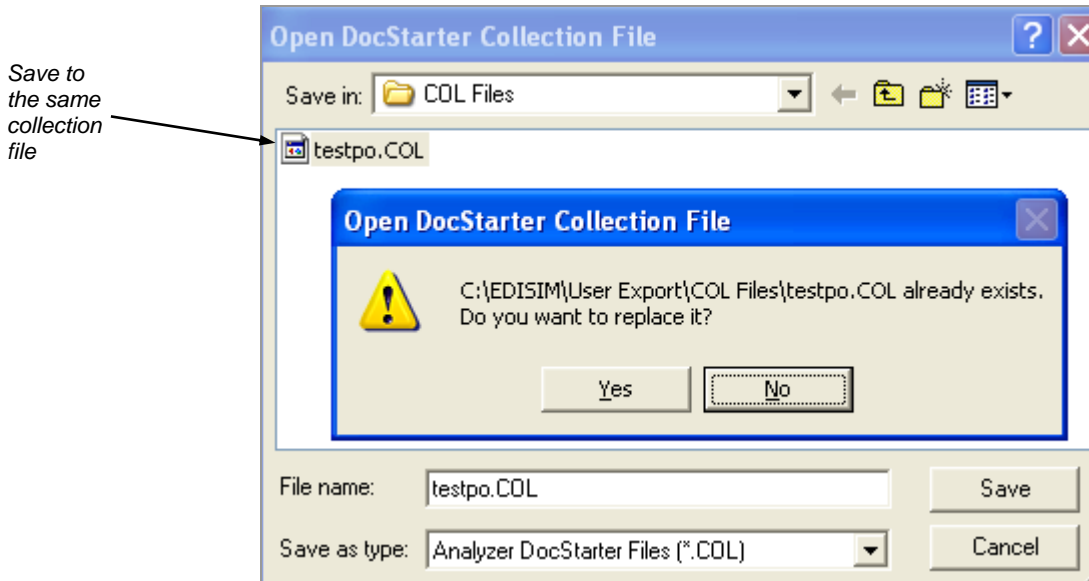
11. Click **Save**.

All of the data from testpo1.txt is added to the collection.
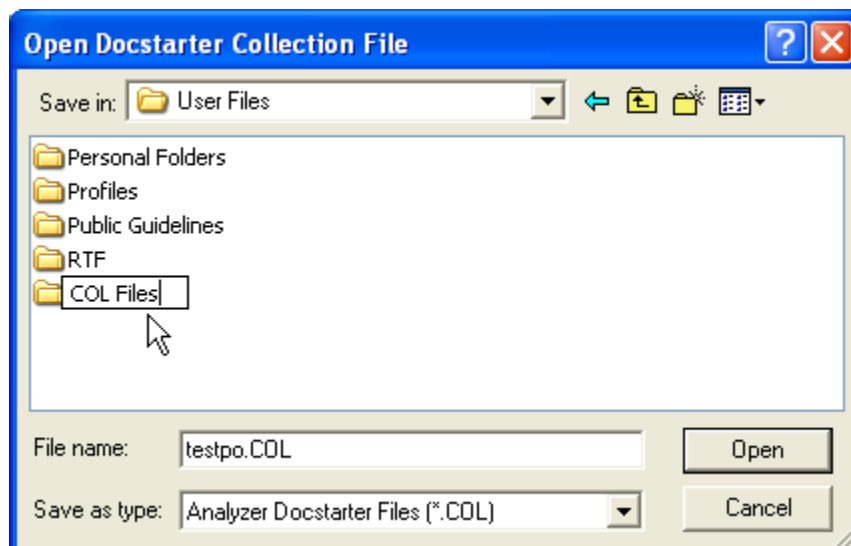
Note: You could continue repeating this process, analyzing other EDI files that can help you define the guideline, and saving the results to the collection file. More collected files yield better guidelines.

12. Close the DocStarter collection dialog box.

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

STOP

Importing a DocStarter Collection File

We can now use Standards Editor to import the collection file we just created.

START

1. Open Standards Editor.

2. Choose **File | Import | Import Analyzer Docstarter File**.

3. Go to EDISIM's **User Files\COL Files** directory and open **testpo.COL**.

4. Leave **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 X12 compliant.

5. Click **OK**.

   Standards Editor imports the DocStarter collection.

6. Look at the codes in the **BEG** segment. These codes came from the EDI data files.

7. Exit Standards Editor.

STOP

Comments

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

  Important: If you customize the imported DocStarter guideline 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 guidelines and migrate the changes to the customized guideline.

Comparator

Compare guidelines and standards

For more information: See **TIB_fsp_edisim_<*n.n*>_fscompar.pdf** in EDISIM's
Documentation directory

Purpose

The purpose of this exercise is to compare our guideline CLASS850, which is based on X12-5010, with X12_5040. We then migrate our user-created changes to a new guideline.

What can you compare?

- Published standard to another published standard

- Guideline to a published standard

- Guideline to guideline

Preparing an X12-5040 Guideline for Migration

Let's assume that we are now preparing to move our 850 to version X12-5040. We do not want to have to start developing the new guideline from scratch.

Instead, we move our local edits to version 5040 by:

- Using Standards Editor to create a new 5040 guideline with two N1 loops and no other edits.

- Using Comparator to migrate the CLASS850 changes to this new guideline.

START

1. Open Standards Editor.

2. Create a new **X12_5040** 850 guideline.

3. Be sure that **View | Show Unused** is selected.

4. Open the **850** and Table 1.

5. Click **Loop N1** just below Loop N9 but do not open the loop.

6. Choose **Edit | Copy** and then **Edit | Paste** to insert an additional N1 loop.

   Since there were no other structural changes to CLASS850, we are done.

7. Save the guideline as **850_5040** and exit Standards Editor.

STOP

Choosing What to Compare

START

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

2. In Comparator choose **File | New**.

3. For Base Guideline, select **CLASS850**.

   This is the "present" standard or guideline.
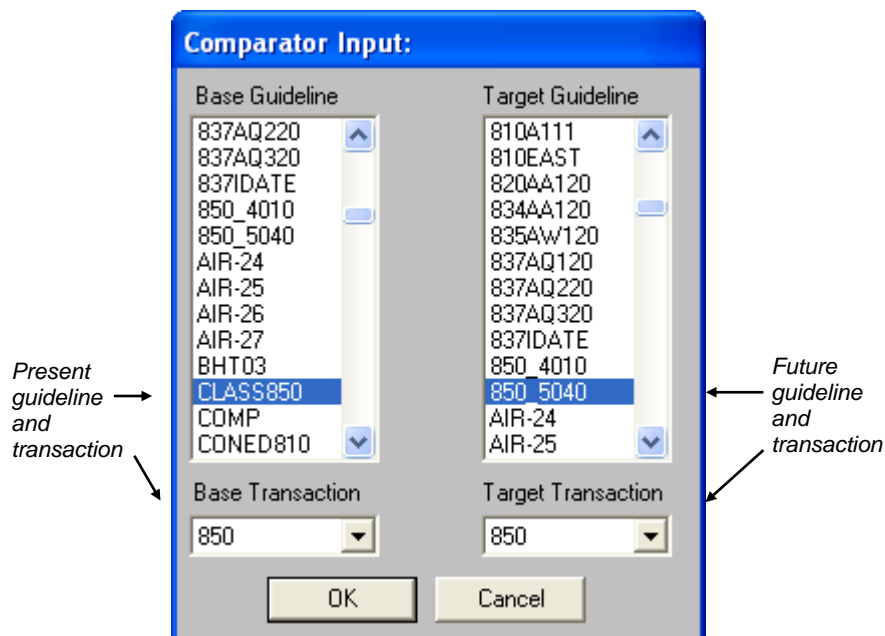
4. For Base Transaction, select **850**.

   This is the "present" transaction set.

5. For Target Guideline, select **850_5040**.
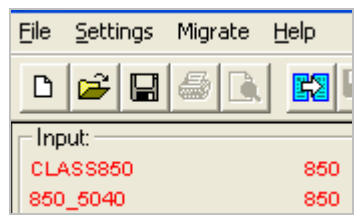
   This is the "future" standard or guideline.

6. For Target Transaction, select **850**.

   This is the "future" transaction set.

Present guideline and transaction →

Future guideline and transaction ←

7. Click **OK**.

The Input area at the top left now shows what will be compared:

STOP

Selecting what Characteristics to Compare

Before comparing the two guidelines, we'll need to tell Comparator which characteristics of each item should be compared.
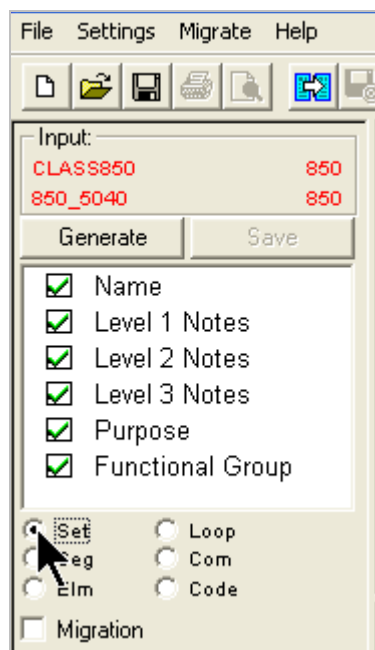
Omit things that are not of interest to speed up the Comparator processing.

Only compared options can be migrated.

START

1. Click the **Set** button in the middle of the Control pane on the left.

   The white area above the Set button now shows items that can be compared for transaction sets.

2. Double-click in the white area to clear all checkboxes at once.

3. Select **Level 1 Notes** and **Level 2 Notes**.

   For the transaction set itself, we want to compare only level 1 and level 2 notes.
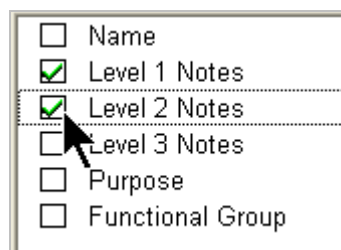
4. Click the **Seg** (segment) button. Double-click in the Item Options area again to clear all options, and select the following options (scroll down to see all of them):

   Requirement
   Max Use
   Name
   User Attribute
   Level 1 Notes
   Level 2 Notes
   Level 3 Notes
   Business Rules

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

   Requirement
   Min Length
   Max Length
   User Attribute
   Type
   Level 1 Notes
   Level 2 Notes
   Level 3 Notes
   Appl Values
   Business Rules

6. Click **Loop**. Notice that you have only two active attributes:

   Max Use
   Loop ID

   Leave these settings alone.

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

   Requirement
   User Attribute
   Level 1 Notes
   Level 2 Notes
   Level 3 Notes

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

   Used
   Level 1 Notes
   Level 2 Notes
   Level 3 Notes
   Description
   Subsets

STOP

Comments

- When migrating, if the base and the target both have a code set for a particular element, Merge Values 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) under Elm.

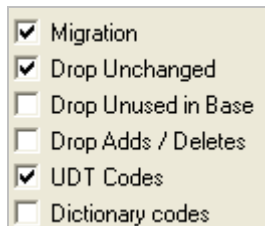# Setting Global Options and Generating a Comparison

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.

   ☑ Migration
   ☑ Drop Unchanged
   ☐ Drop Unused in Base
   ☐ Drop Adds / Deletes
   ☑ UDT Codes
   ☐ Dictionary codes

4. Save these settings in case we want to use them again:

   Choose **File | Save As**.

   Create a directory **ComparatorSettings** in EDISIM's **User Files** directory.
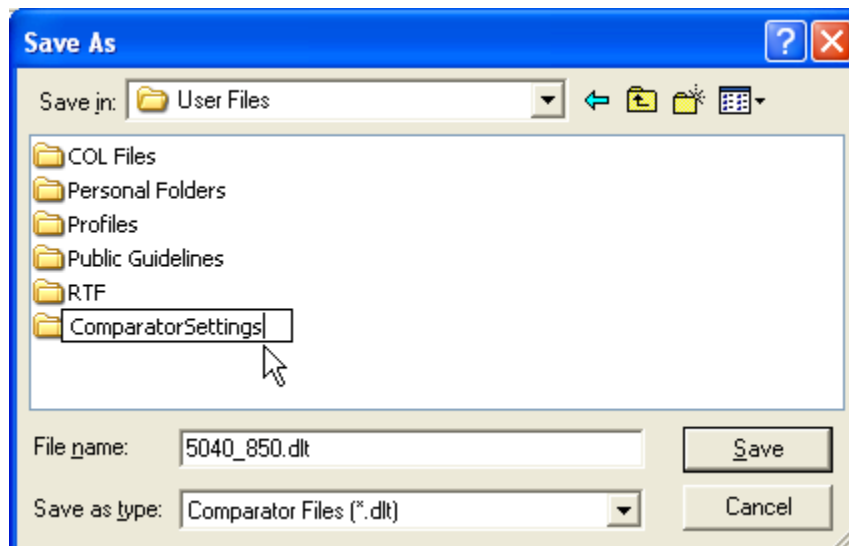
   Save the file in the new directory.

5. Click **Generate** at the top left.

6. When finished, you will see base and target columns like this:

Base .... *Target*

STOP

Comments

- Global Options:

  - Drop Unchanged - if all of an item's compared options are the same in the base and target, do not display the item at all. Select this option for migration.

  - 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.

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

  - UDT (User Defined Transaction) Codes - compare the selected options for code sets used in the transaction set. It will not compare unchanged X12 code lists.

  - 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.

- To re-use the saved settings later, choose **File | New 5040_850**. Then choose **Settings | Input**, and select the base and target.

- 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 right pane so that you can see the entire **850 Purchase Order** line:

   Drag the vertical splitter bar to the left as far as possible.
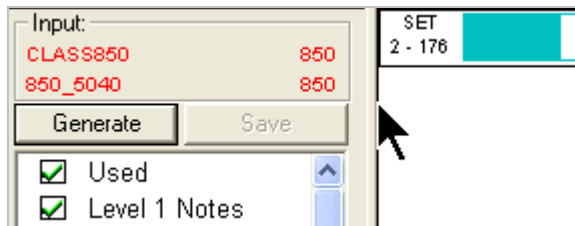
   Drag the horizontal splitter bar downward to view more of the upper pane.

   Click on the **850 Purchase Order** line with the blue rectangles, then adjust the size by repeatedly clicking the magnification buttons on the toolbar.

2. Double-click on the **850** to expand the segments and loops.

   Comparator is showing CLASS850 (base) on the left and 850_5040 (target) on the right.

3. Look in the first column to the left the **850 Purchase Order**.

SET 2 - 176	This means that this line is a transaction set, and Comparator found 2 changes to the 850 itself and 176 changes to the subordinates of the 850. These include changes between the X12-5010 and X12_5040 versions as well as changes that you made.

   The number that displays may differ somewhat, depending on your guideline and Comparator settings.

4. Look in the bottom pane to see the 2 changes to the 850.

   CLASS850 has the Level 1 and 2 Notes that you entered in Standards Editor.

   CLASS850 Notes   850_5040 has no notes

---
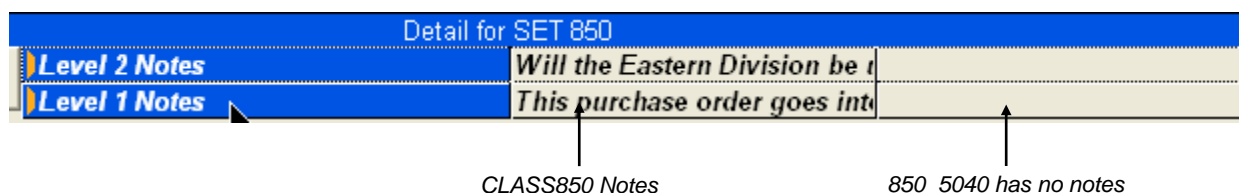
5. Click on the **REF** segment and look at the Detail pane.

   It shows a User Attribute change between CLASS850 and X12_5040. In CLASS850, it Must Be Used.

6. Double-click on the REF to expose its subordinates, and then click on the REF02. There are three differences.

   Differences caused by our own changes are in *Italic*. Those caused by changes to the underlying standard are not Italic, like Max Length below:

Detail for ELM 127		
Appl Values	*REFID, Reference ID,*	
Level 1 Notes	*The required value fo.*	
Max Length	50	80

   CLASS850 has an application value list. This was our own change (we can tell because the font is Italic).

   CLASS850 has a level 1 note. This was our change also. To see the rest of the note, double-click on it. Choose Cancel when finished.

   Max Length changed from 50 to 80. We can see that this is an X12 change because the font is not Italic.

7. Click on the 850 Transaction Set line. Choose **Settings | Expand All Nodes**.

   You will see all items that have changed.

STOP

Changing Views

Before migrating your changes, you may want 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** and save it to a new directory **ComparatorReports** under EDISIM's **User Files** directory.

3. Choose **File | Print Preview**.

4. Click the comparator screen until you have zoomed in completely.

   An asterisk (*) indicates that an item is different between base and target.
   A plus sign (+) indicates that an X12 item has been added to the target.
   A minus sign (-) indicates that an X12 item has been removed from the target.

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

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

7. 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: Loop IDs and syntax rule changes. Code sets migrate, but a user-created code set used in multiple places will migrate as separate code sets.

START

1. Open the **850** and the **PO1 Loop** near the bottom.

2. Click on the **PO1** segment and look in the first column.

   There are 3 changes to the PO1 and 26 changes to its subordinates.

3. With the **PO1** segment selected, look at the Detail pane.

   The PO1 has Level 1, 2, and 3 Notes in CLASS850 but none in 850_5040.

4. Open the **PO1** segment and click on **PO101** (element **350**), which has different min and max lengths between CLASS850 and 850_5040.

5. Double-click on the **639** at **PO105** and click on its **Udt** codes box. To see its code set differences, click **Load** in the bottom pane.

   The three codes are in the code set we made in CLASS850. That is why you see <CLASS850> in the Codes Changed area.

6. Since we want to have this code set in 850_5040, click the **Migrate** button to the left of the codes.

   The entire code set migrates. Notice the 639 Udt shows 0 changes. Since you migrated them, the codes in the base and the target are now the same and so the codes no longer show.

7. Select the **PO1** segment (not the loop).

   The notes that we added appear at the bottom. Double-click on **Level 2 Notes** to see the whole note.

Text Viewer

CLASS850

850_5040

Is there a limit to the number of PO1 loops that we can send?

Migrate | Cancel
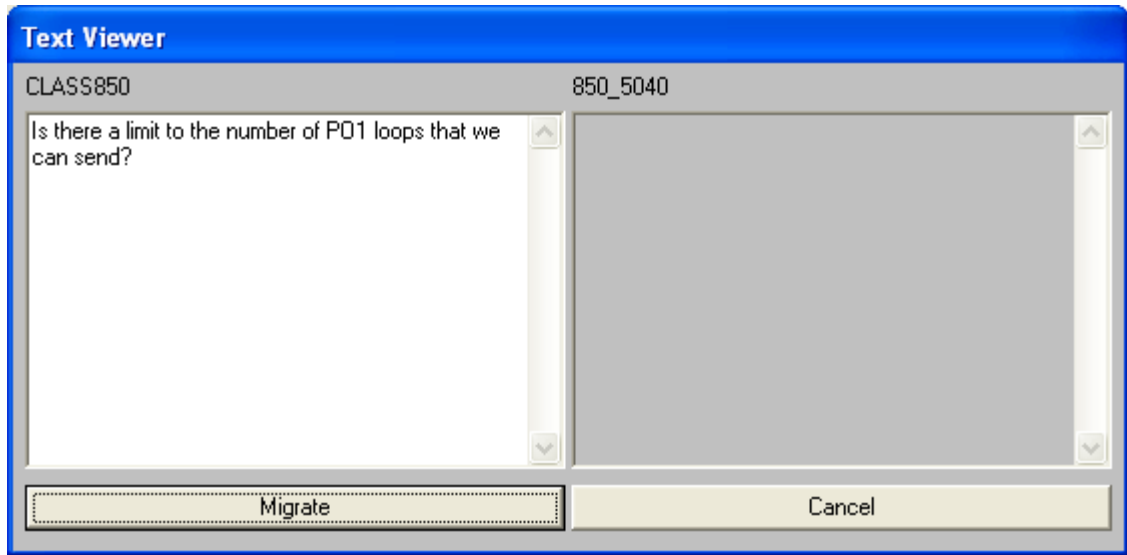
8. You can edit the note before you migrate it. The edits affect only the target (the base is *never* changed by Comparator). Click **Migrate**.

   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 both notes, since they differ.

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

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

10. Migrate the **Level 3 Note** also.

11. Select the **PO105** (**639**). There are level 1 and 2 notes in the base. Drag slowly across the gray box in front of the 639 in the base. When the cursor turns into a little bottle, 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 element and its subordinates all at once. Two other ways to do this are choosing **Migrate | Current** on the menu bar or selecting the Migrate Current and Subordinates button ⌐ on the toolbar.

12. Scroll up and double-click on the first **N1 loop** (Ship To), and view the differences. Click the **Migrate Current Only** button ~ on the toolbar or choose **Migrate | Current Only**.

   This method only migrates changes for the currently selected item.

13. Migrate the remaining user-created changes. Click the **Migrate All** button ⌐ or choose **Migrate | All**.

   Only X12 changes are left.

STOP

Saving the Target Guideline

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 .

2. Leave the Standard Name as **850_5040** and replace the Description text with **X12-4050 Purchase Order for EDISIM Training Class**. Click **OK**.

3. Choose **File | Exit**.

Standards Reference

View structure and details of a guideline

For more information: See **TIB_fsp_edisim_ *<n.n>*_fseditor.pdf** in EDISIM's Documentation directory (you will be in read-only mode and cannot change the guideline.)

Purpose

The purpose of this exercise is to view the structure of a standard and our guidelines online. We'll look at how to navigate through them, print a report to file, and view the dictionary.

Why use Standards Reference?

- Since Standards Reference is read-only, there is no chance of accidentally changing the guideline.

- For network installations, one person can leave the guideline open all day in Standards Reference without locking the standard so others cannot use it.

- You can view all of information about a transaction set.

- You can view complete dictionaries that are categorized by transaction set, segment, composite, or element.

- You can easily locate items using the cross-reference feature.

Opening a Standard

We begin by looking at an X12 standard.

START

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

2. Maximize.

3. Choose **File | Open | X12-5040** (on the Published Standards tab).

   All transaction sets in X12-5040 appear.

4. Open the **850, Table 2,** the **PO1 loop,** and the **PO1 segment.**
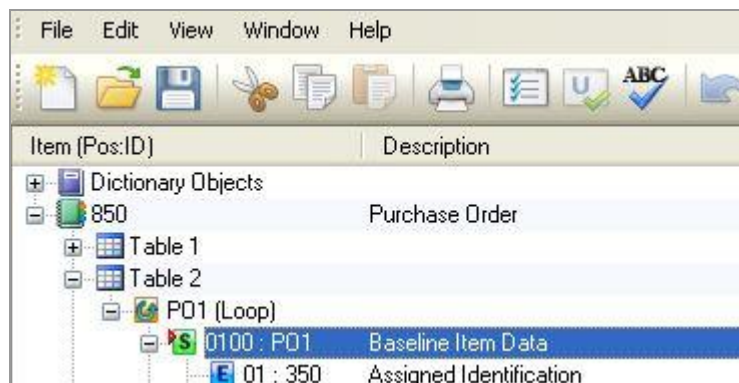
STOP

Viewing Cross-References

START

1. Click on **PO105**, element **639**.

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

   Element 639 is used in 7 segments in X12-5040:
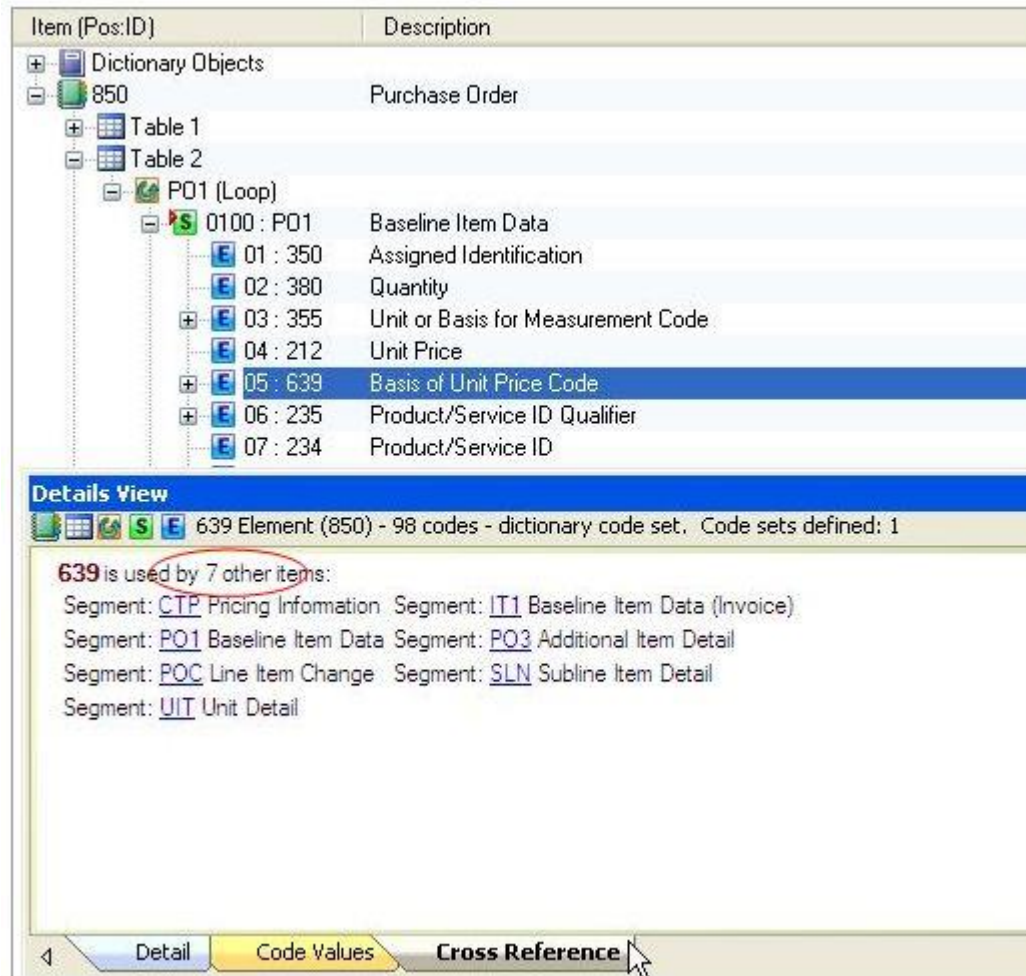
3. Click on IT1 to see that segment in the Segment dictionary.

4. Where else is the IT1 used? Click on it in the top pane and look at the cross-reference tab. It is used in 9 transaction sets.

5. Scroll up and close **Dictionary Objects**, and then scroll down to the **850** transaction set. Click on the **PO105** once again.

6. Select the **Code Values** tab at the bottom of the screen.
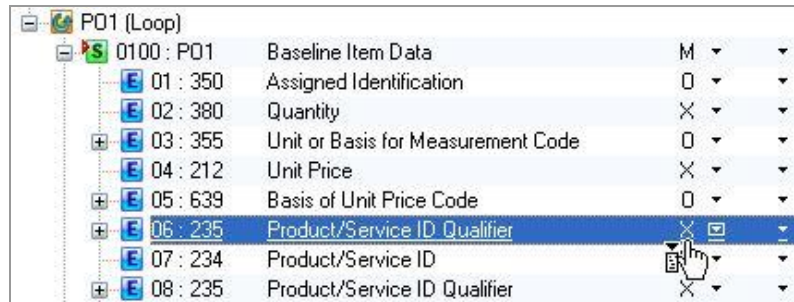
   All code values for the PO105, data element 639 are here.

STOP

Viewing Syntax Notes

START

1. Click on **PO106**, element **235**, which has a Requirement of X.
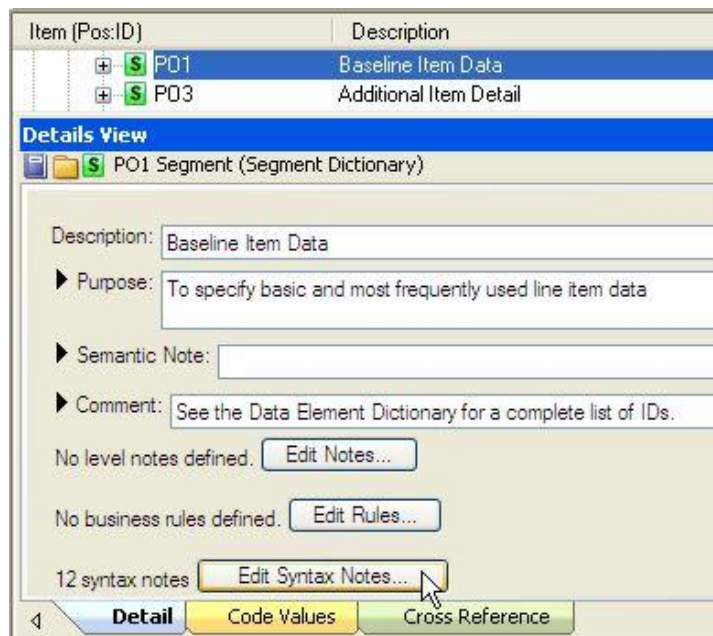
    X means that X12 has a syntax note (or dependency) on its usage. We can see the syntax notes in the segments dictionary.

2. Open **Dictionary Objects | Segments** at the top.
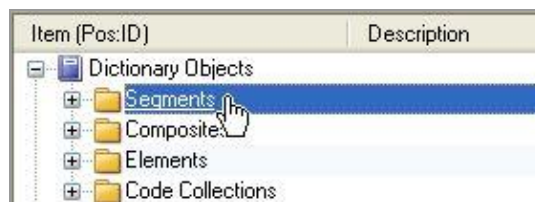
3. Click on the **PO1** segment, open the **Detail** tab at the bottom, and click **Edit Syntax Notes**. All syntax notes for the PO1 segment are listed. When finished, **Cancel**.

STOP

Viewing a Guideline
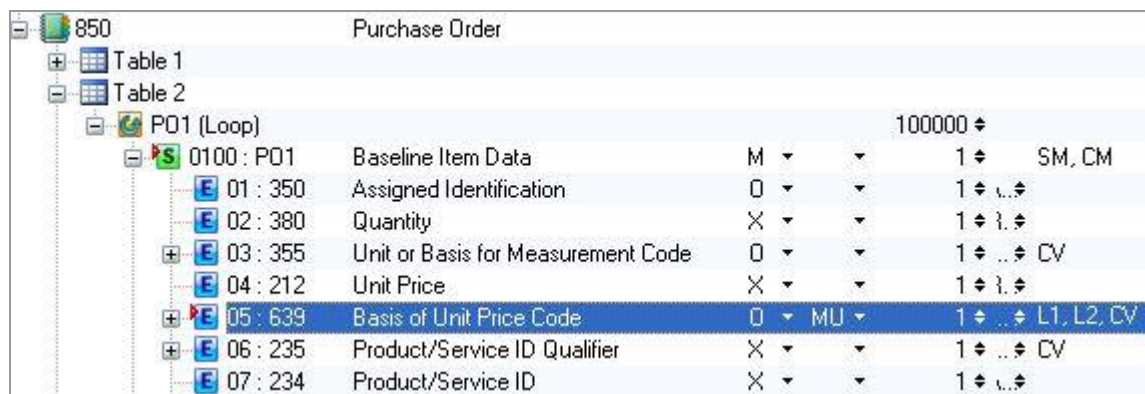
Now we will look at our CLASS850 guideline and view some of the changes that we made in Standards Editor.

1. Close X12-5040.

2. Click the **Open** button 📁 on the toolbar and select the **User Guidelines** tab.

3. Open **CLASS850**.

4. Open the **850 Purchase Order** and **Table 1**.

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

    The used segments appear.

6. Open the **PO1** segment in Table 2 and click on the **PO105**.

7. Select the **Code Values** tab at the bottom.

8. Right-click on the **PC** local code and select **Level Notes/Properties**.

9. Click Cancel.

Comments

- You can toggle **View | Show Unused Items** to show or conceal unused segments, elements, composites, and codes.

- You can see semantic notes and comments, and industry guideline segment notes in the lower portion of the Detail pane in the Segment dictionary.

Printing Standards or Guidelines

You can print a report about the open standard or guideline. This is useful as a quick reference or when searching for items.
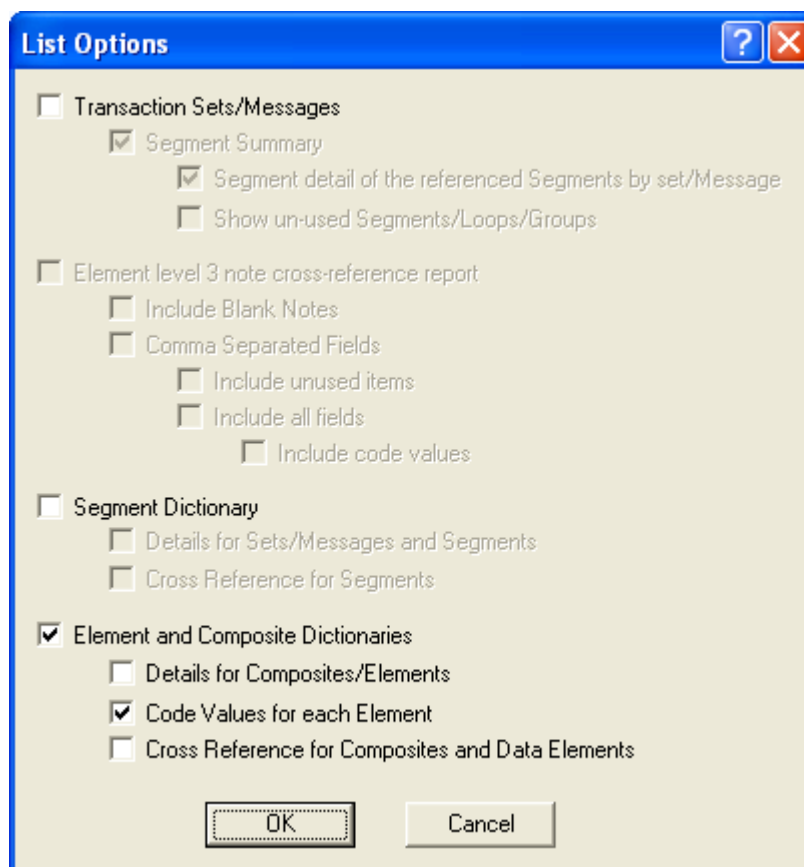
For a formatted document, use Doc Builder.

START

1. Scroll to the top of the top pane and click on Dictionary Objects.
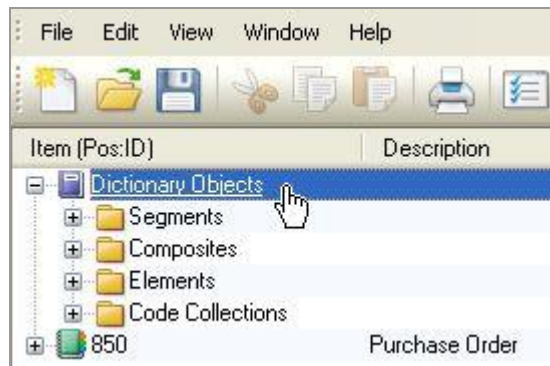
2. Choose **File | Print**. The List Options dialog box appears.

3. Select these check boxes and choose **OK**:

---

4. Scroll through the data. Notice the Save As and Print buttons.

5. Click **Close**.

STOP

Comments

- The List Options dialog box enables options according to what you have selected:

  - Guideline 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.

Reference

Indicator Codes

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
CV	Code Value

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