



SCISSOR LIFT TUTORIAL

An Introduction to DriveWorksXpress

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INTRODUCTION TO DRIVEWORKSXPRESS

DriveWorksXpress is entry-level Design Automation software included in SOLIDWORKS. It is ideal if the designs and projects you work on are the **Same but Different**. It is widely used in many industries from trailers to conveyors, furniture to machinery, mechanical seals to pressure vessels, windows & doors.

You'll find it already installed and waiting for you to use under the SOLIDWORKS Tools menu. It is ideal for everyday repetitive design tasks. Use it to create multiple variations of SOLIDWORKS Parts, Assemblies and Drawings quickly and accurately.

This DriveWorksXpress tutorial is intended to provide a quick introduction to using DriveWorksXpress.

The Tutorial will show how DriveWorksXpress can:

- Reduce the cost of custom designs
- Create SOLIDWORKS assembly, parts and drawings quickly
- Enhance product quality
- Eliminate or reduce repetitive tasks

Upon successful completion of this tutorial, you will be able to:

- Drive SOLIDWORKS part and assembly geometry with DriveWorksXpress
- Create a DriveWorksXpress input form and link the input fields to the SOLIDWORKS model
- Write rules to configure and run your design projects
- Generate new parts, assemblies and drawings

ABOUT THE TUTORIAL

In this tutorial, we will be automating the creation of a scissor lift. Imagine you work for a company that designs and manufactures scissor lifts, where each scissor lift you design is the same but different.

The length and width of the scissor lift can be changed. This means a custom design needs to be produced for each sales enquiry or order. This could take a few hours and be very repetitive for you, the engineer. Time spent creating custom files manually could also delay the company's sales cycle.

However, by using DriveWorksXpress, it is possible to reduce both the lead time and the cost of custom designs to help companies to be more competitive and win more business.



GETTING STARTED

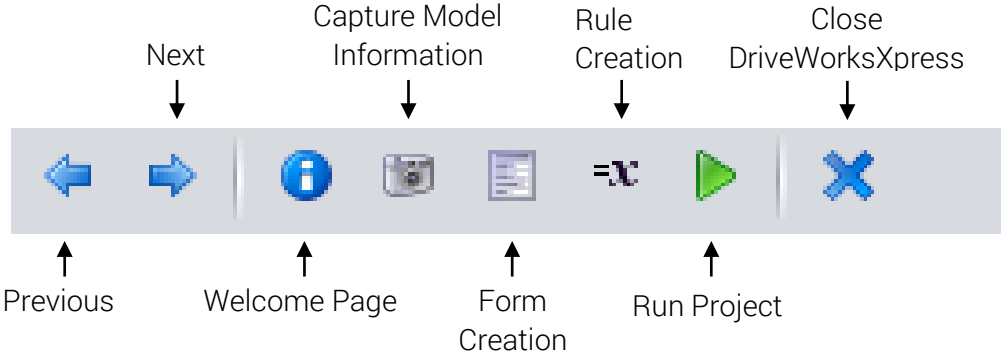
To begin this tutorial, you will need to download the SOLIDWORKS project files for the scissor lift from the DriveWorksXpress website: www.driveworksxpess.com

The SOLIDWORKS files are contained within a zipped file.

Ensure you extract the files from this folder before beginning the training.

By following this DriveWorksXpress tutorial you will soon be automating your own designs in SOLIDWORKS.

DRIVEWORKSXPRESS NAVIGATION BAR



REGISTRATION AND BASIC SETUP

DriveWorksXpress is included in every seat of SOLIDWORKS. You'll find DriveWorksXpress by navigating to Tools > Xpress Products > DriveWorksXpress in the SOLIDWORKS tool bar.

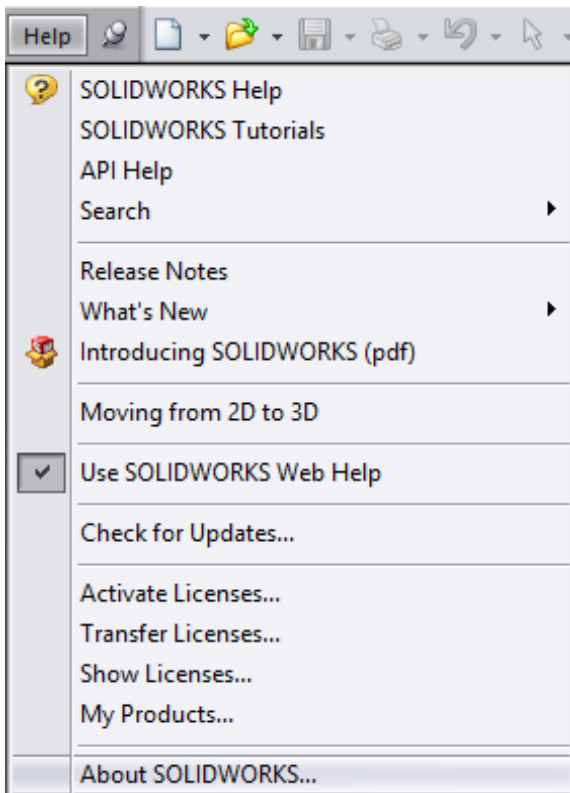
The first time you use DriveWorksXpress in SOLIDWORKS 2015, you will need to log into your SOLIDWORKS account and register DriveWorksXpress.

If you are using SOLIDWORKS 2014, you can skip directly to Basic Setup.

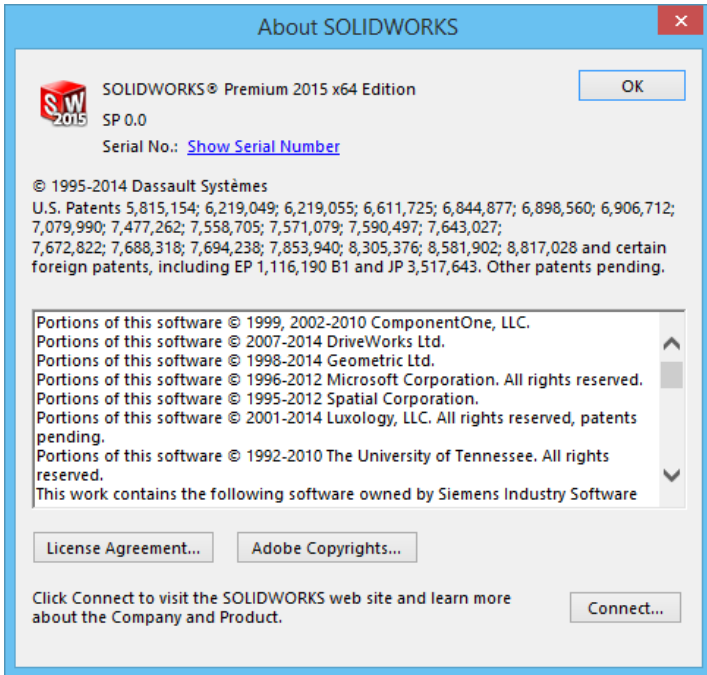
DRIVEWORKSXPRESS 2015 REGISTRATION

STEP 1

You'll need your SOLIDWORKS Serial Number. You can find this by navigating to: Help > About SOLIDWORKS.



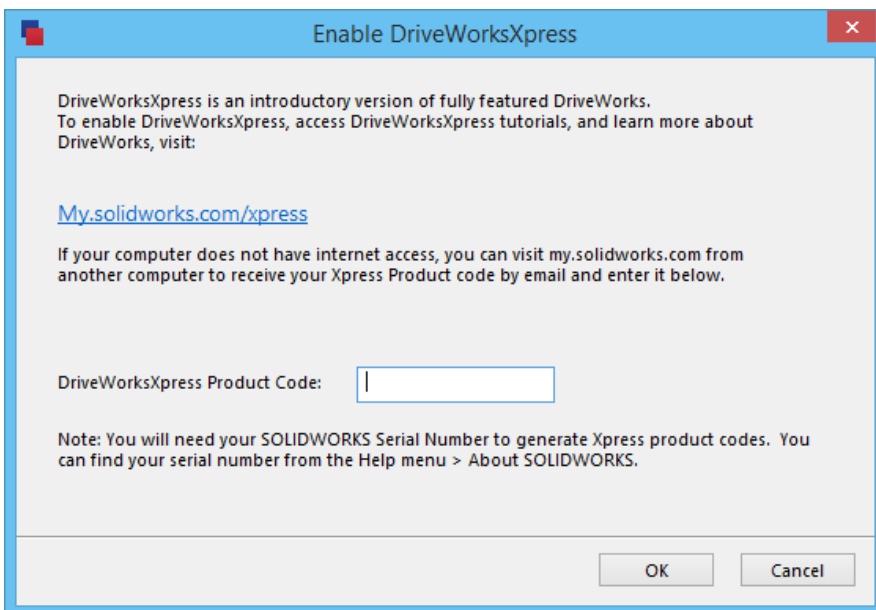
Click 'Show Serial Number' and copy it ready for step two.



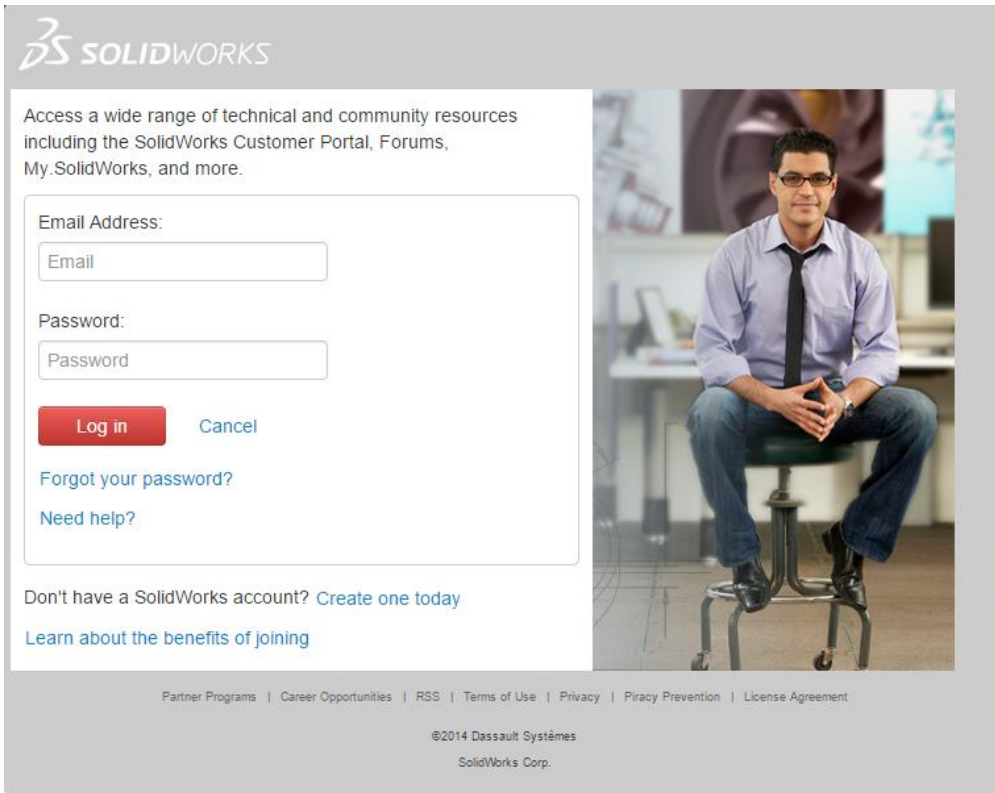
STEP 2

Once you've got your SOLIDWORKS Serial Number, open DriveWorksXpress.

You will be asked to log into your My SOLIDWORKS account and register DriveWorksXpress with your SOLIDWORKS Serial Number.



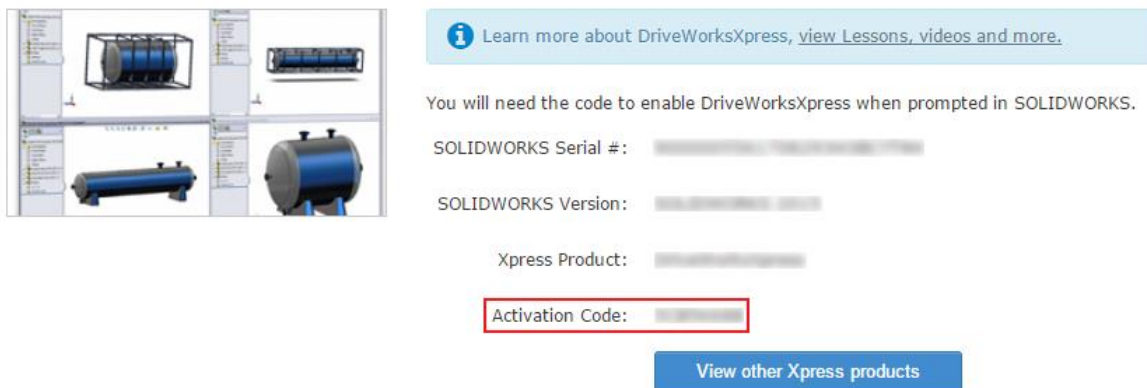
If you don't already have an account, you will need to create one with SOLIDWORKS. This is really easy, just click the link to create an account.



STEP 3

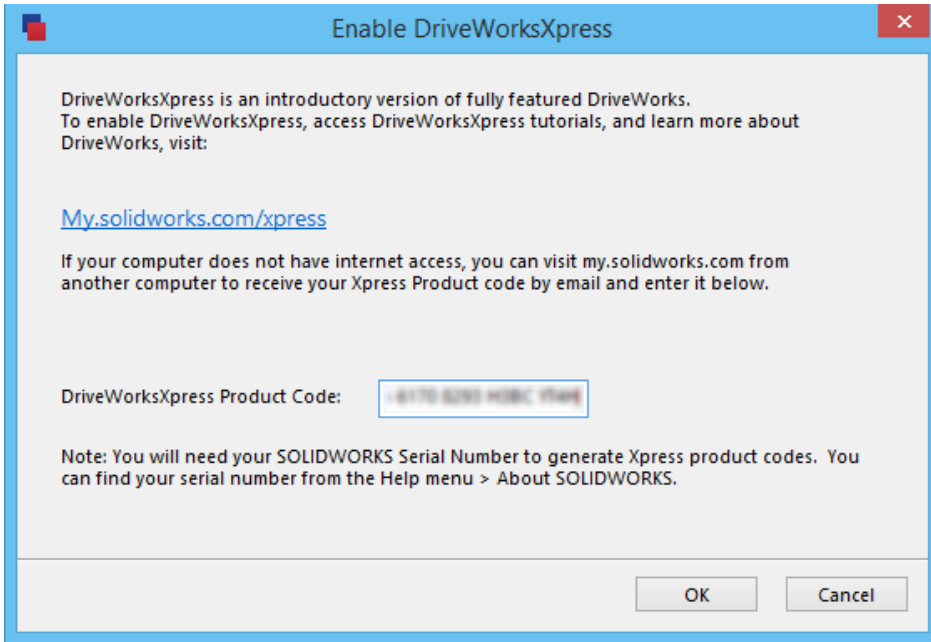
Once you're logged in, you will get an Activation Code for DriveWorksXpress.

SOLIDWORKS 2015 DriveWorksXpress



Copy the Activation Code and return to SOLIDWORKS.

Paste the code into the activation window and click OK.

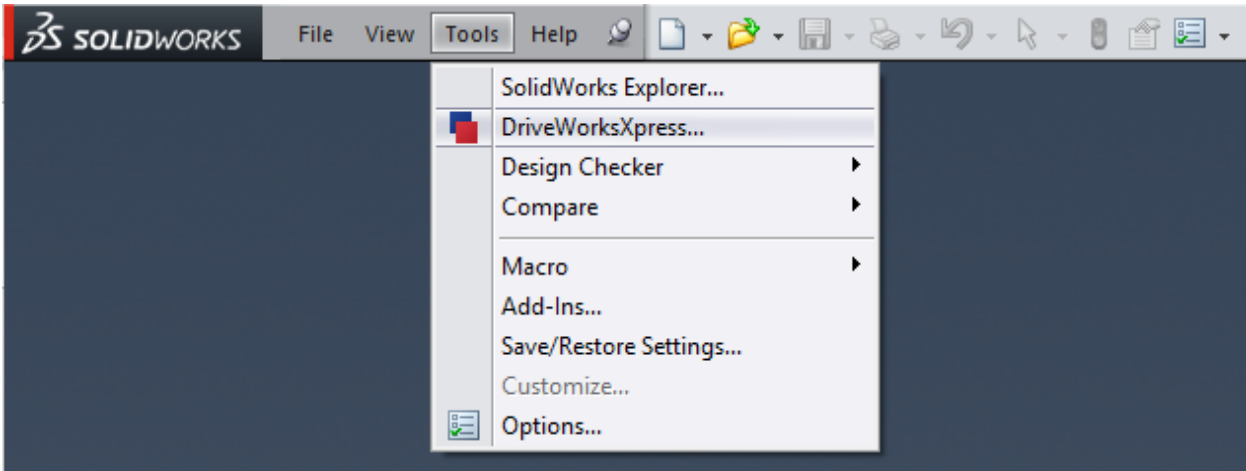


DriveWorksXpress will be activated and the task pane will open.
You're now ready to automate your designs with DriveWorksXpress!

BASIC SETUP

LAUNCHING DRIVEWORKSXPRESS

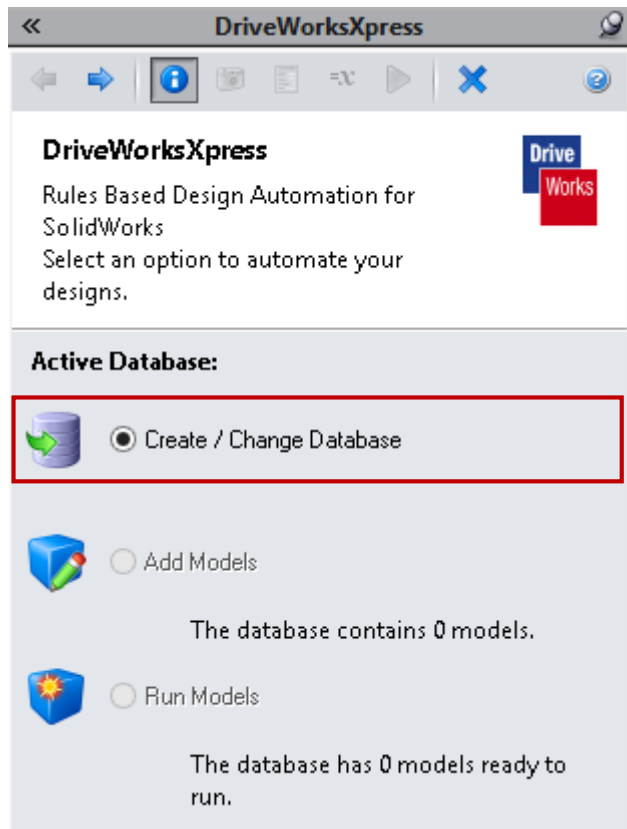
Click on the 'Tools' tab in the SOLIDWORKS menu bar and select DriveWorksXpress from the drop-down options.



This activates DriveWorksXpress, which will open on the right hand side of the screen.

The DriveWorksXpress Welcome Page gives you three options:

- **Create / Change Database**
This allows you to create a new project or load and change an existing project
- **Add / Edit Models**
Add more models to the existing project
- **Run Models**
Jump directly to Run to specify a new variation using a completed project



CREATE A NEW DATABASE

To create a new DriveWorksXpress Database, click the 'Create/Change Database' radio button.

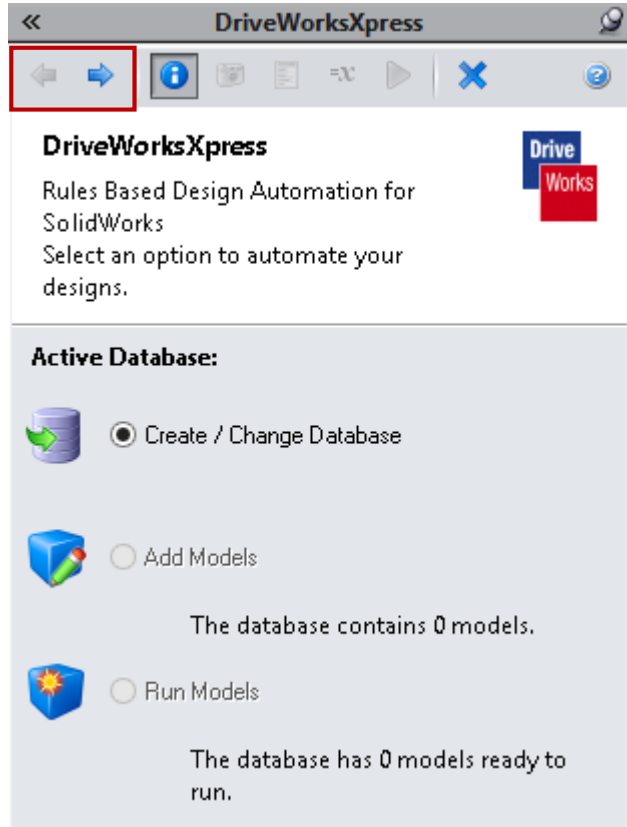
Click 'Next' in the DriveWorksXpress Task Pane.

You will automatically be asked to open a new database.

Browse to the location where you want to create your new database and name it 'Scissor Lift'.

Click 'Open' to save the database and continue.

A new DriveWorksXpress database will be saved in your specified location.



CAPTURING MODELS AND DIMENSIONS

CAPTURING SOLIDWORKS MODELS

Click Next to navigate to the next window within the DriveWorksXpress Task Pane.

This will allow you to capture your models and the parameters that will be controlled. There are three ways to select which models are to be captured and driven using DriveWorksXpress:

- **Browse for new model**

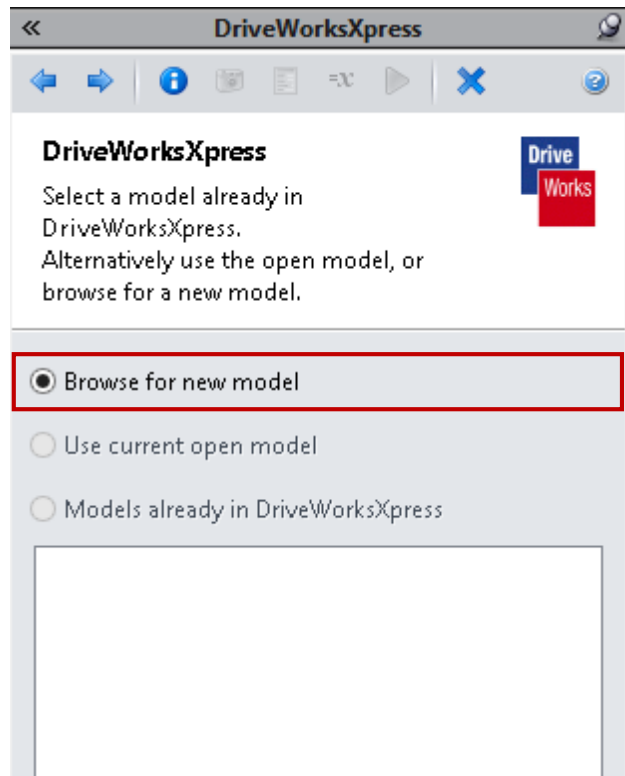
If no models are open in SOLIDWORKS, you can browse to the location of the files you would like to be driven and automated using DriveWorksXpress

- **Use current open model**

If you currently have a part or assembly open in SOLIDWORKS, DriveWorksXpress can capture these models to be driven and automated

- **Models already in DriveWorksXpress**

If you have previously captured models within DriveWorksXpress, these will be displayed in the DriveWorksXpress Task Pane



Make sure 'Browse for new model' is selected and click 'Next'.

Browse to the location on your hard drive where you have saved your Scissor Lift files, open the folder and select the SOLIDWORKS Assembly Document called 'Scissor Lift Assembly.SLDASM'

Click 'Open'

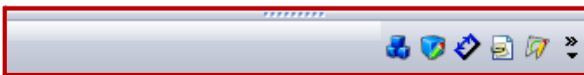
The Scissor Lift assembly will open in SOLIDWORKS.

Once DriveWorksXpress knows the location of the files, we'll need to tell it which components we want to capture dimensions and features from.



The DriveWorksXpress Task Pane will now show all of the options for capturing model information.

NOTE – It is possible for these tabs to be dragged and compressed into one line in order to create more work space in DriveWorksXpress. If you don't see the tabs displayed in the image above, it is possible that you may see the following at the bottom of the Task Pane:



- **Captured Models**

This shows the models that have been captured inside of DriveWorks.

- **Captured Assembly Structure**

This allows you to see all the models making up the structure of the assembly within a 'tree' formation. This option allows you to select which models within the assembly that you would like to capture.

- **Dimensions and Features**

Dimensions and features can be controlled by double clicking a model from the model list within the model tree above the Capture options and then clicking 'Dimensions and Features'.

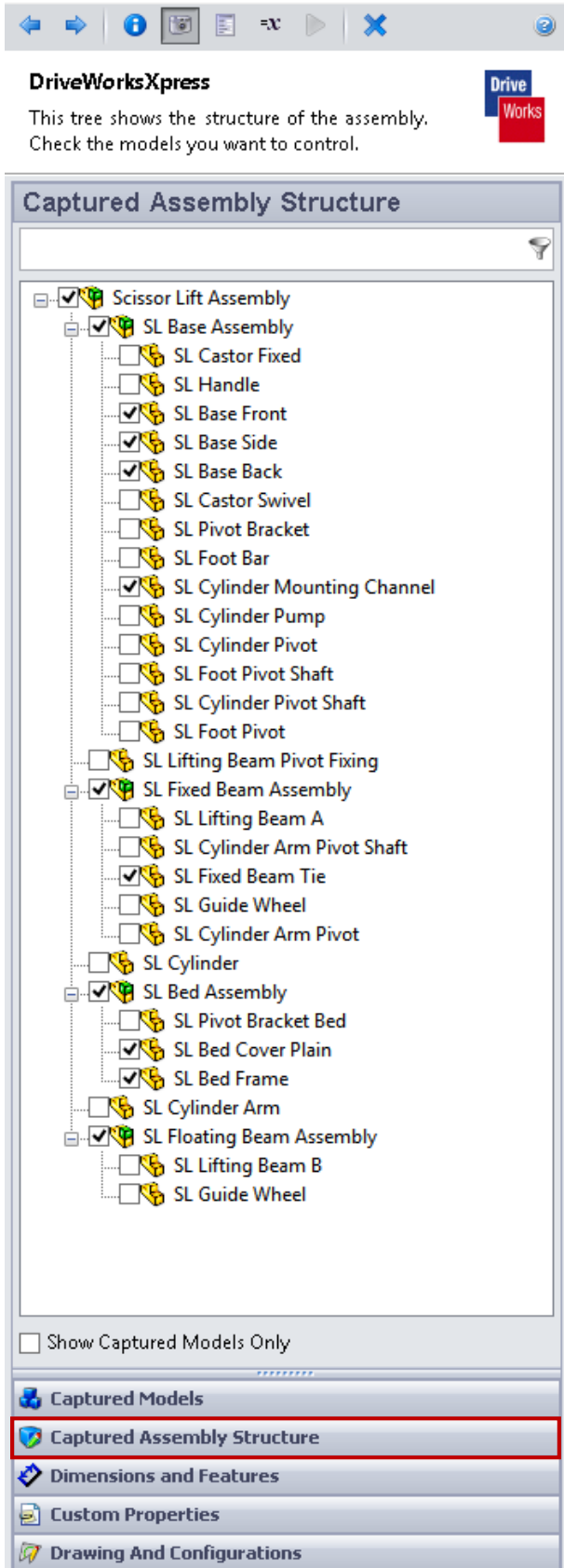
- **Custom Properties**

Any custom property that has been assigned in SOLIDWORKS can be captured and driven by DriveWorksXpress.

- **Drawings and Configurations**

If there are manufacturing drawings, they can be located and added to DriveWorksXpress by selecting this tab and hitting 'Browse' under the 'Drawings' heading.

If rules are required to drive which configuration of a part or assembly gets used, this can also be indicated in the 'Drawing and Configurations' option.



At the bottom of the Task Pane, select the 'Captured Assembly Structure' tab. This will display check boxes next to each model in the assembly. Select the parts/assemblies that you want DriveWorksXpress to control by checking the box next to each item.

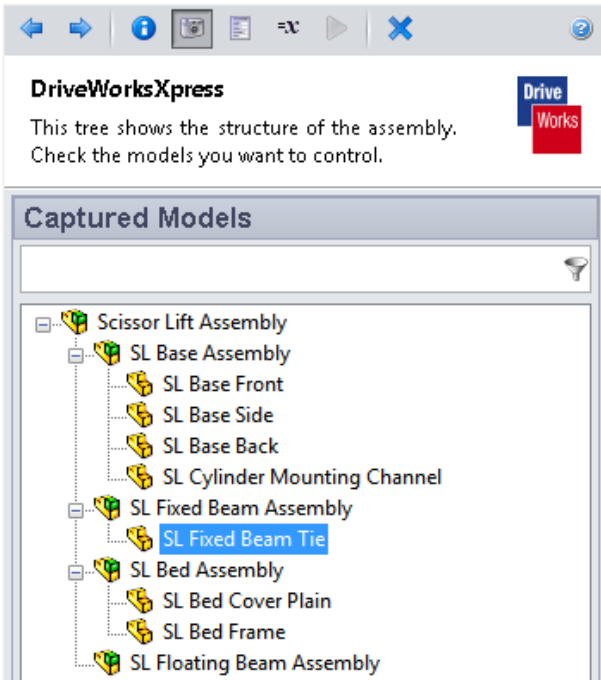
The top level assembly (Scissor Lift Assembly) will already be checked. Not every component in the assembly needs to be controlled.

Capture the models that have been checked in the image.

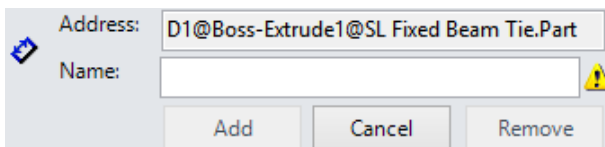
CAPTURING DIMENSIONS AND FEATURES

The Dimensions and Features tab allows you to choose which parameters from a model or assembly you would like to capture and drive. You can then assign a descriptive name to the dimension or feature you have selected to make them easier to identify and create rules for.

A parameter is captured by selecting the 'Captured Models' tab and then double clicking on the model that the parameter exists in.



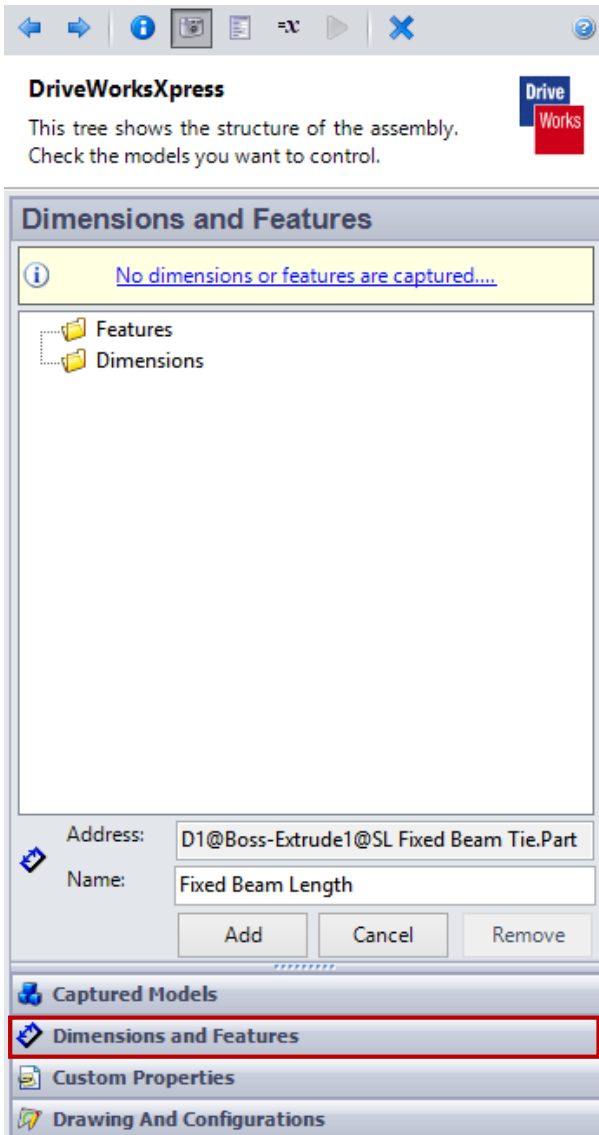
This will open the model in SOLIDWORKS allowing you to select the dimensions and features you wish to capture and control.



With the models captured, begin the process of capturing the parameters that you need to control. The following steps will guide you through the process of capturing dimensions and features in DriveWorksXpress.

CAPTURING THE SL FIXED BEAM TIE

Double click on the SL Fixed Beam Tie in the 'Captured Models' tab and then select the 'Dimensions and Features' tab.



For the SL Fixed Beam Tie, capture the dimension associated to length of the Boss-Extrude1. This dimension determines the length of the beam. Double Click 'Boss-Extrude1' in the SOLIDWORKS Feature Tree, and then select the dimension displayed in the graphical view.

With this dimension selected, DriveWorksXpress will display the dimension address in the text box labelled 'Address'. DriveWorksXpress allows you to provide the dimension with a reference name.

Type 'Fixed Beam Length' in the 'Name' box and Click Add.

Providing a reference name will make it easier for you to comprehend what the dimension controls, when building your rules.

The captured dimension will be listed under Dimensions and Features in the DriveWorksXpress task pane.

The next step is to capture the rest of the features and dimensions from each individual part of the Scissor Lift assembly. The process of capturing dimensions is the same throughout every part and assembly

CAPTURING DIMENSIONS AND FEATURES

Navigate back to the original Scissor Lift assembly window in SOLIDWORKS by clicking 'Back', clicking 'Captured Models' or by closing the open model and clicking 'Next'.



Continue through every captured part and capture the following dimensions using the process explained above.

Be careful to ensure that you are capturing the dimension and not the feature, this can be checked by looking at the icon next to the Address and Name fields:

Capturing Features:

A screenshot of the 'Capturing Features' dialog box. The 'Address' field contains 'Distance4'. The 'Name' field is empty. A red circle highlights a feature icon (a green and yellow globe) next to the 'Address' field. A yellow warning triangle is visible next to the 'Name' field. At the bottom are 'Add', 'Cancel', and 'Remove' buttons.

Capturing Dimensions:

A screenshot of the 'Capturing Dimensions' dialog box. The 'Address' field contains 'D1@Distance4@SL Floating Beam Assembly.A'. The 'Name' field is empty. A red circle highlights a dimension icon (a blue square with a white 'D') next to the 'Address' field. A yellow warning triangle is visible next to the 'Name' field. At the bottom are 'Add', 'Cancel', and 'Remove' buttons.

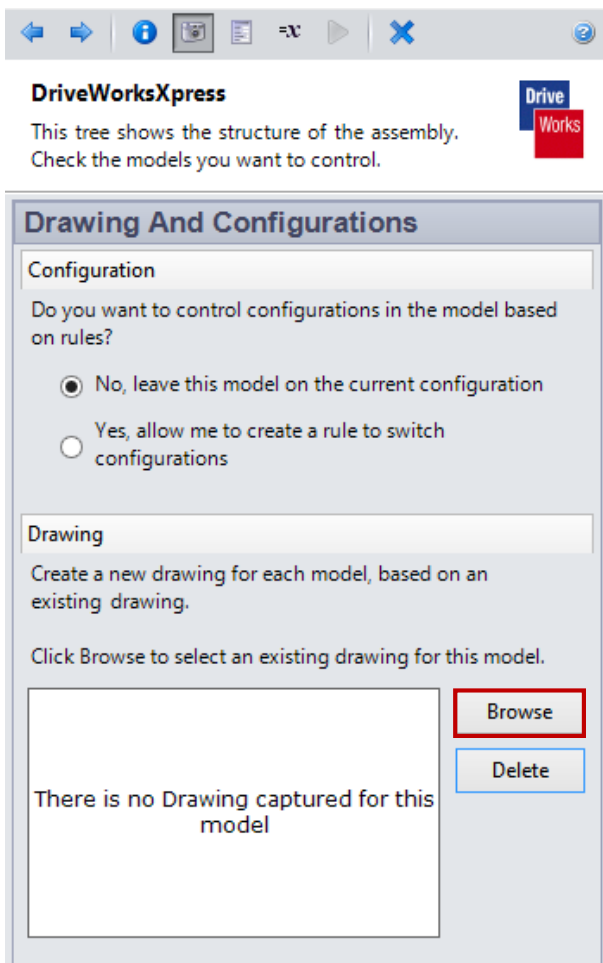
Capture the following dimensions and features:

Component	Dimension Address	DriveWorks Name	Dimension Value (mm)	Captured
SL Floating Beam Assembly	D1@Distance4	Floating Beam Width	317	
SL Fixed Beam Assembly	D1@Distance4	Fixed Beam Width	263	
	D1@Distance5	Half Fixed Beam Width	144	
SL Fixed Beam Tie	D1@Boss-Extrude1	Fixed Beam Length	288	Yes
SL Bed Assembly				
SL Bed Frame	D2@Sketch1	Frame Length	480	
	D1@Sketch1	Frame Width	346	
SL Bed Cover Plain	D1@Sketch1	Width	350	
	D1@Extrude-Thin1	Length	500	
SL Base Assembly	D1@Distance6	Half Base Width	175	
	D1@Distance5	Half Base Width Handle	175	
	D1@Distance3	Bracket Distance	263	
SL Base Front	D1@Boss-Extrude1	Width	350	
SL Base Side	D1@Boss-Extrude1	Length	500	
SL Base Back	D1@Boss-Extrude1	Width	350	
SL Cylinder Mounting Channel	D1@Boss-Extrude1	Length	500	

CAPTURING DRAWINGS

With DriveWorksXpress you can capture your SOLIDWORKS engineering drawings. This feature means that when a new version of the model is made using DriveWorksXpress, the engineering drawing of that component is also updated to match it. Click the 'Browse' button to search your hard drive for the corresponding SOLIDWORKS drawing file.

To add a drawing, open the model or assembly using the tree view in the Captured Models tab and then browse to the Drawings and Configurations tab. Click the 'Browse' button to search your hard drive for the corresponding SOLIDWORKS drawing file.



Open the Scissor Lift assembly and browse for the drawing 'Scissor Lift Assembly.SLDDRW'.

Click 'Open' to capture the drawing.

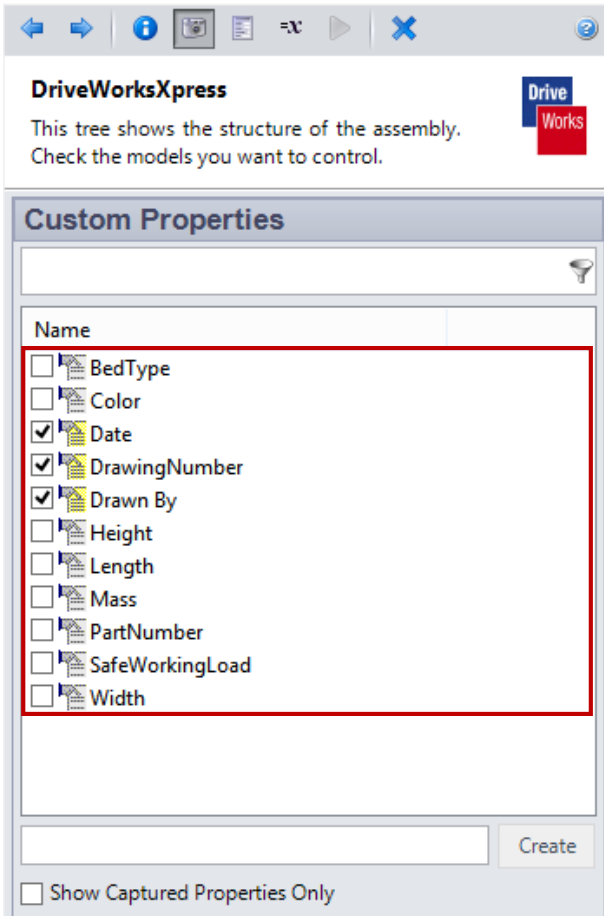
It is possible to add one drawing per part or assembly.

The captured drawing will be updated and saved with a new name every time a specification is run.

CAPTURING CUSTOM PROPERTIES

By linking Annotation Text to the value of a Custom Property, DriveWorksXpress can automatically populate Drawing Borders with information that is specific to each newly created model.

To do this you must capture the Custom Properties that you would like to control. Then, build a rule for these properties and ensure that a value is driven into them every time.



With the Scissor Lift Assembly still open, select the 'Custom Properties' tab. Here you will see a list of the Custom Properties that have been created in this Assembly.

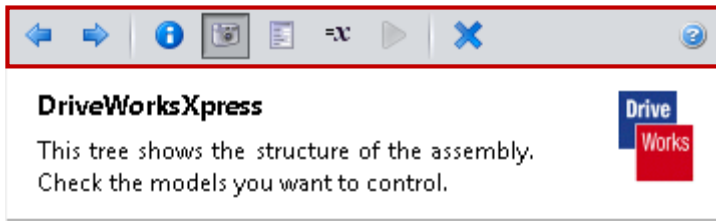
Check the Custom Properties Date, DrawingNumber and Drawn By properties and close the model.

CREATING INPUT FORMS

In DriveWorksXpress you can create an input form for entering the values for your new parts and drawings. This form can be used again and again to specify and generate all the new parts and drawings, based on the rules you set and values you enter.

ADDING CONTROLS

To access the Form Designer, click the 'Next' arrow at the top of the DriveWorksXpress Task Pane, or select the Form Creation Icon shown below.

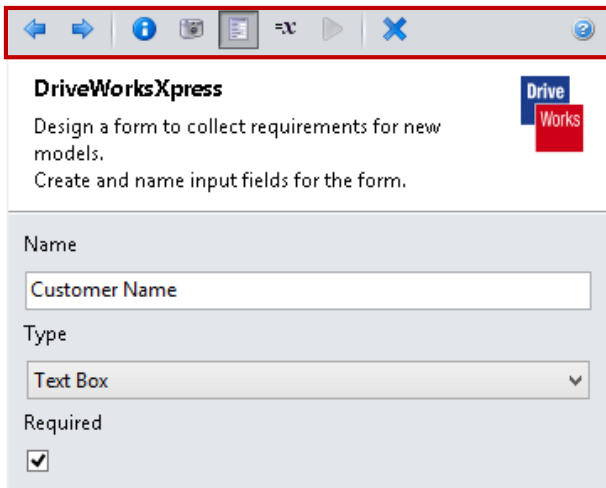


Each control added requires three things:

- **Name** - You must provide a descriptive name for the form control. This name will be the title the user sees as they fill in your form i.e. Customer Name
- **Type** - Choose from five types of controls (inputs)
 - Text Box:** Input text directly by typing
 - Numeric Text Box:** Input numeric values and specify a minimum and maximum value
 - Drop Down:** Provides a list of options to choose from
 - Spin Button:** Users can select from a range of numeric values. A maximum and a minimum value, as well as the incremented value
 - Check Box:** Places a checkbox on the form
- **Required** - Enforces a value to be entered

In this example, create a Customer Name and an Order Number Text Box control. Later you will need to build a rule to append the Customer Name and Order Number to each file so that each set of new files in a specification can be easily identified. These two values will also be used to create a Drawing Number which will be driven into the Drawing Border of the Scissor Lift drawing.

Begin by adding a Text Box to your Form to allow a Customer Name to be entered.



Add a new Text Box called Customer Name:

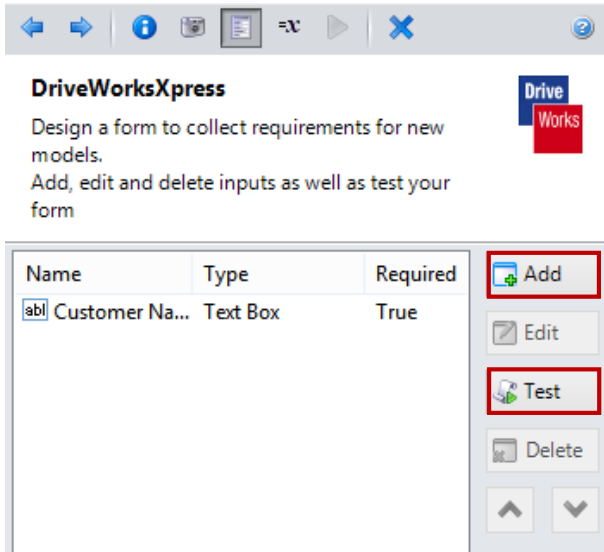
Control Name	Control Type	Required
Customer Name	Text Box	Yes

Type the name of the Control and then use the Drop Down to select the Control Type.

Tick the 'Required' check box to ensure a Customer Name is entered for each new specification.

Click 'Next' to register the control and display the Control List.

Notice that the form control appears in the form designer within the DriveWorksXpress Task Pane. As you create your form, you can edit and delete controls, as well as change the order by using the Up and Down arrows.



You can also test your form by clicking 'Test'.

Click 'Add' and repeat the steps to add the following controls:

Add a new Text Box called Order Number:

Control Name	Control Type	Required
Order Number	Text Box	Yes

Add a Spin Button called Length to control the length of the lifting bed.

Add a new Spin Button called Length:

Control Name	Control Type	Min Value	Max Value	Increment
Length	Spin Button	500	850	10

Add a Spin Button called Width to control the width of the lifting bed.

Add a new Spin Button called Width:

Control Name	Control Type	Min Value	Max Value	Increment
Width	Spin Button	300	850	10

TEST MODE

DriveWorksXpress

Enter information to test the form.
Click on Set Defaults to see the current values when you fill out the form.

Scissor Lift Assembly

Customer Name

Order Number

Length

Width

500

300

Clear Set Defaults

Click 'Test' within the Form Designer to preview the form you have created and test it out for yourself.

You will notice that as you fill out the forms with valid information, the background of the text boxes will change from pink to white and the yellow warning triangles will disappear.

If you hover over a control, information about the control will be shown in a tooltip.

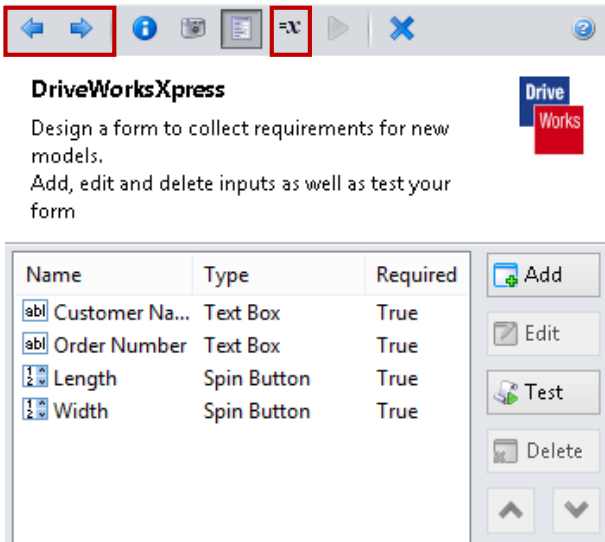
It's good practice to set default values for controls. Setting default values makes building rules easier because the controls have values.

To set a default value, enter values into the form controls and click 'Set Defaults'.

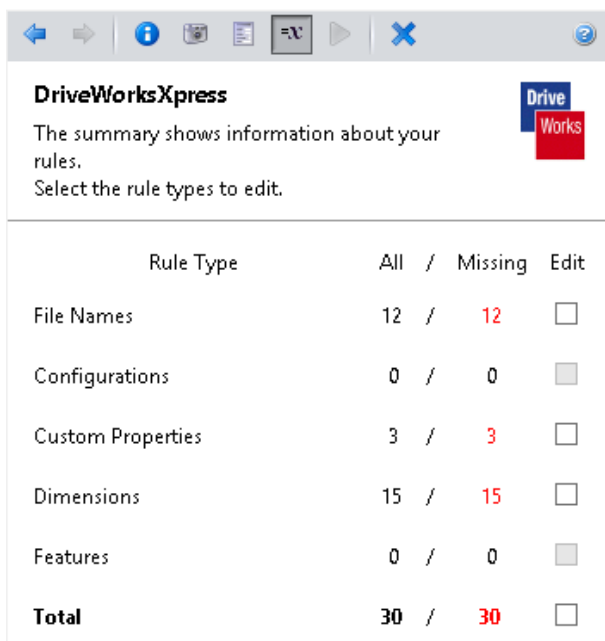
RULES BUILDER

DriveWorksXpress lets you use Excel syntax to build rules to automate your SOLIDWORKS models.

Navigate to the Rules tab by clicking 'Next' or 'Rules' in the DriveWorksXpress Task Pane.



The Rules tab shows a summary of your rules and how many of each rule type there are. The summary also shows the number of unbuilt rules that must be completed before a new Specification can be created:

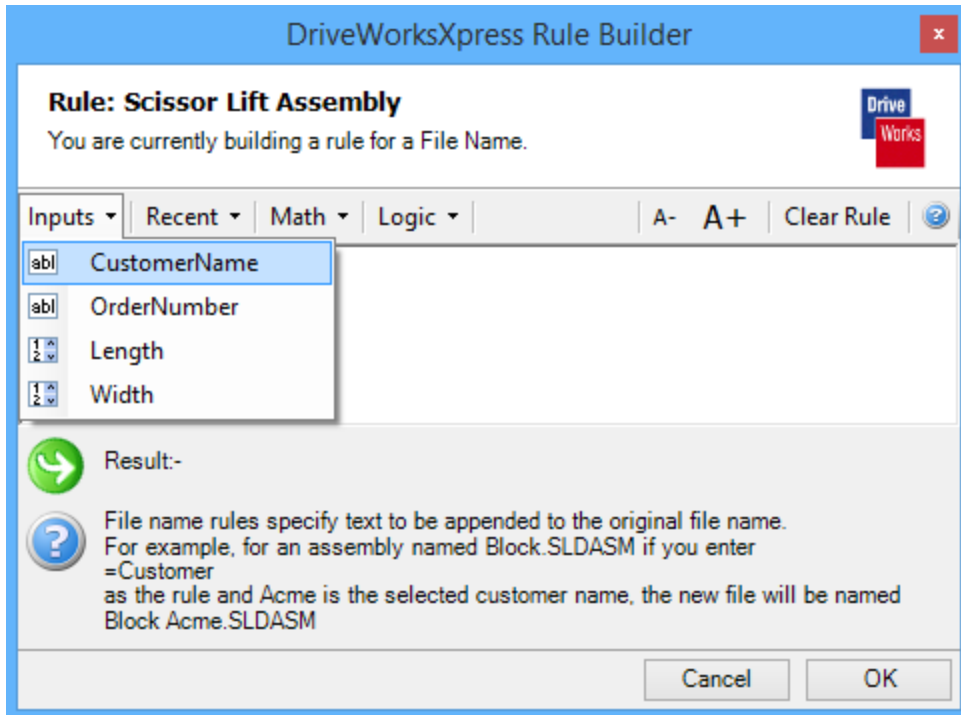


Selecting the check box against a Rule Type filters the rules by that type. It is possible to select more than one rule type at a time.

Filtering is helpful where there are large numbers of rules to be built. Where Rules are not required for a particular type, the check box will not be enabled.

The summary provides constant feedback on Total Number of Rules and Missing Rules (which still require rules to be built against them).

Before you begin to create rules within this tutorial, here is a quick summary of the DriveWorksXpress Rules Builder.



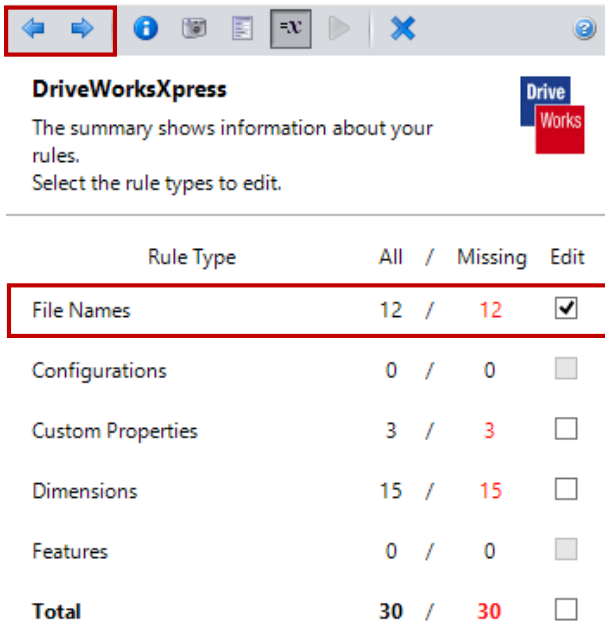
There are four drop-down menus within the DriveWorksXpress Rule Builder:

- **Inputs**
Lists the available controls that can be used in rules
- **Recent**
This can be populated with commonly used text strings or equations
- **Math**
Mathematical operators (e.g. add, subtract, and divide)
- **Logic**
Logical operators (e.g. IF, <, >, =)

FILE NAME RULES

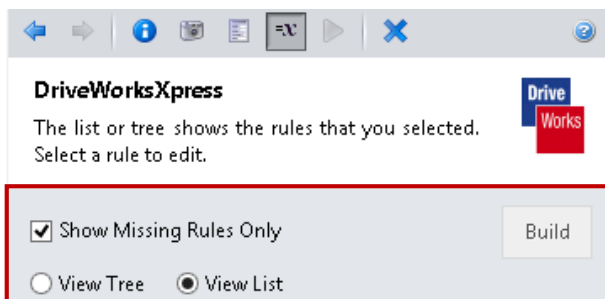
File Name rules allow different sets of files that are generated for each new specification of an automated design to be easily identifiable. Each of the files being driven will be for a unique project and therefore will require a unique set of file names.

Check the File Name check box and click 'Next' in the DriveWorksXpress Navigation.



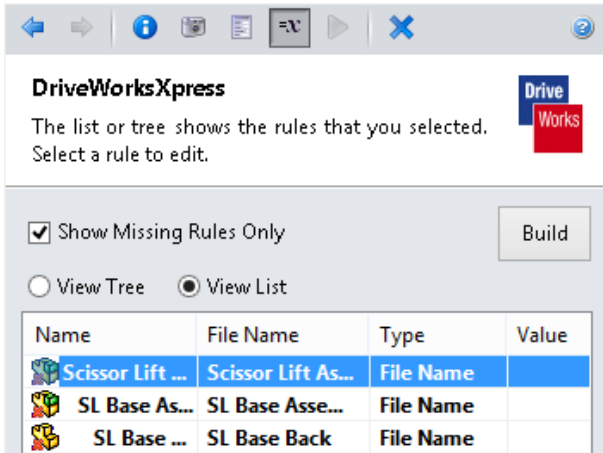
The captured components requiring a File Name rule will be listed.

You can choose to show all rules or just missing rules that have not been built yet. You also have the option to view your files as a List or in a Tree View. The default is List View.



BUILD A RULE FOR THE SCISSOR LIFT ASSEMBLY FILE NAME

Select the Scissor Lift Assembly and then click 'Build' or double-click.



The DriveWorksXpress Rules Builder will now open.

All the file names that are going to be created for these models will need to be unique to that particular specification and therefore the file names should be populated with information that will allow the files to be easily identified. For this tutorial, this will involve using the customer's name and their order number within the file name.

To create unique file names for each new specification, file names should be populated with information that will allow you to recognise what they are. For the Scissor Lift Assembly, use the Customer Name and Order Number inputs to create file names that have the format, "Component OrderNumber – CustomerName".

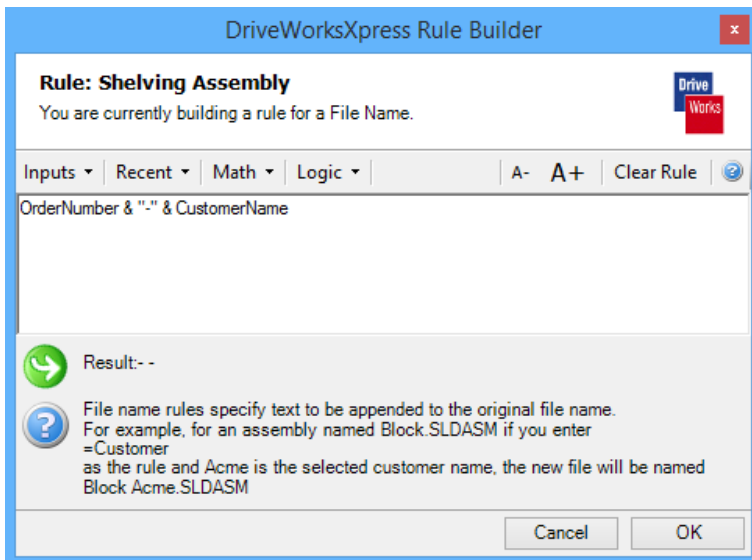
Start by selecting the 'OrderNumber' control from the Inputs tab. This will now place the value from the control into the rule

To incorporate the hyphen between OrderNumber and CustomerName, ampersands (&) need to be used. This is because OrderNumber is one string and the hyphen is another string. Therefore, you need to type & "-".

The rule should now read OrderNumber & "-".

To finish off the File Name rule, type & CustomerName (or select 'CustomerName from the Input menu).

The completed rule should read: **OrderNumber & "-" & CustomerName**



An example of this rule when put into practice will read:

'Scissor Lift Assembly 123 – John Smith'

Click 'OK' to save the rule and exit the Rules Builder

There will now be 11 File name rules remaining.

BUILDING MULTIPLE FILE NAME RULES

DriveWorksXpress enables you to build multiple rules at the same time. By holding Ctrl and selecting multiple File Names you can build them at the same time, applying the same rule to each selected item.

The rules for the Parts and Sub-Assemblies are going to be different from the main assembly. You are going to incorporate the dimensions that will change in each part into the File Name rule. This will mean that DriveWorks will create and build up a standard set of components as their file name will be based on their actual size. This will speed up generation as DriveWorks will not create the part if it already exists.

Name	File Name
SL Base Assembly	SL Base Assembly
SL Base Back	SL Base Back
SL Base Side	SL Base Side
SL Cylinder Mounting Channel	SL Cylinder Mount
SL Base Front	SL Base Front
SL Bed Assembly	SL Bed Assembly
SL Bed Frame	SL Bed Frame
SL Bed Cover Plain	SL Bed Cover Plain
SL Fixed Beam Assembly	SL Fixed Beam Ass
SL Fixed Beam Tie	SL Fixed Beam Tie
SL Floating Beam Assembly	SL Floating Beam

To begin with, select all the Parts that will change when the length of the Scissor Lift changes. **The parts to select are the 'SL Cylinder Mounting Channel' and the 'SL Base Side'.**

Click 'Build' and create a rule that uses Length Input.

Build the following rules using the Length input:

Name	Rule
SL Base Side	Length
SL Cylinder Mounting Channel	Length

Name	File Name
SL Base Assembly	SL Base Assembly
SL Base Back	SL Base Back
SL Base Front	SL Base Front
SL Bed Assembly	SL Bed Assembly
SL Bed Frame	SL Bed Frame
SL Bed Cover Plain	SL Bed Cover Plain
SL Fixed Beam Assembly	SL Fixed Beam Ass
SL Fixed Beam Tie	SL Fixed Beam Tie
SL Floating Beam Assembly	SL Floating Beam

Next select the parts and assemblies that will change when the Width of the Scissor Lift is changed. **The selected components should be 'SL Base Front', 'SL Base Back', 'SL Fixed Beam Assembly', 'SL Fixed Beam Tie' and 'SL Floating Beam Assembly'.**

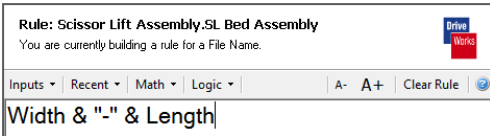
Click 'Build' and create a rule that uses the Width input.

Build the following rules using the Width input:

Name	Rule
SL Base Back	Width
SL Base Front	Width
SL Fixed Beam Assembly	Width
SL Fixed Beam Tie	Width
SL Floating Beam Assembly	Width

Name	File Name
SL Base Assembly	SL Base Assembly
SL Bed Assembly	SL Bed Assembly
SL Bed Frame	SL Bed Frame
SL Bed Cover Plain	SL Bed Cover Plain

Finally select the components that will change when both Width and Length are changed. **These are 'SL Bed Assembly', 'SL Bed Frame', 'SL Bed Cover Plain', and 'SL Base Assembly'.**

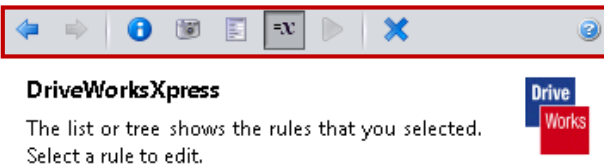


As mentioned previously, to combine values in one rule, they must be separated by an ampersand (&). Add a dash to your rule, to separate the two values.

Build the following rules using the Width and Length inputs:

Name	Rule
SL Base Assembly	Width & "-" & Length
SL Bed Assembly	Width & "-" & Length
SL Bed Frame	Width & "-" & Length
SL Bed Cover Plain	Width & "-" & Length

Click 'Back' in the DriveWorksXpress navigation to return to the Rules Summary page.



DIMENSION RULES

Dimension rules allow you to automate your SOLIDWORKS models by taking information entered on the Form, calculating a result and then sending it to the SOLIDWORKS model. This allows you to control multiple dimensions at the same time using only a few inputs.

Deselect the check box next to 'File Names' and select 'Dimensions'. Click 'Next' in the DriveWorksXpress navigation.

The tables below show the name of each dimension and the rule that should be created for that dimension.

Build the following rules for the captured dimensions:

Rule Name	Component	Rule
Frame Width	SL Bed Frame	= Width
Width	SL Bed Cover Plain	
Width	SL Base Front	
Width	SL Base Back	
Length	SL Base Side	= Length
Length	SL Cylinder Mounting Channel	
Length	SL Bed Cover Plain	
Half Base Width	SL Base Assembly	= Width /2
Half Base Width Handle	SL Base Assembly	
Fixed Beam Width	SL Fixed Beam Assembly	= Width - 87
Bracket Distance	SL Base Assembly	
Half Fixed Beam Width	SL Fixed Beam Assembly	= (Width - 87) /2
Floating Beam Width	SL Floating Beam Assembly	= Width - 33
Fixed Beam Length	SL Fixed Beam Assembly	= Width - 62
Frame Length	SL Bed Frame	= Length - 20

Click 'Back' to return to the Rule Summary Page.

You should now have 0 rules missing for Dimensions.

Deselect the 'Dimensions' check box and select the 'Custom Properties' check box. Click 'Next' in the DriveWorksXpress navigation.

To begin with select the Date Custom Property and click 'Build'.

For this rule the Today() function will be used. This function will return the date at which the model was generated. By default the function will return a date formatted in your local Windows date format.

TIP: The returned date format can be changed using the Text() Function. The Text function will take the information provided and convert it to the format specified. See examples below.

Text(Today(),"mm.dd.yy") would return 12.16.13

Text(Today(),"dddd dd mmmm yyyy") would return Monday 16 December 2013

Have a play around to see what other formats you can achieve.

Build the following rules for the Date and Drawing Number Custom Properties:

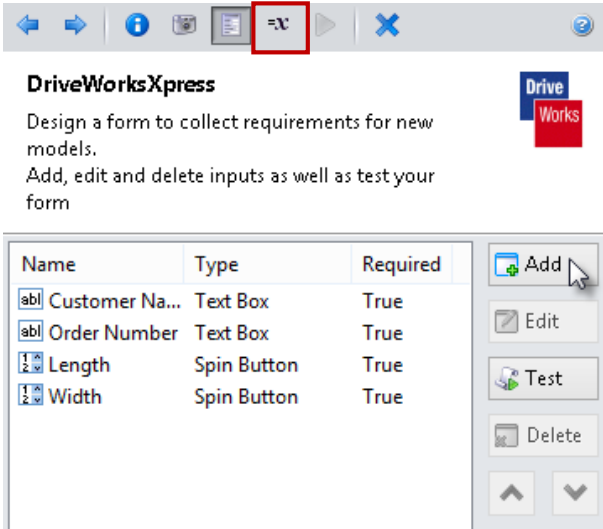
Name	Rule
Date	= Today()
Drawing Number	= OrderNumber & "-" & Left(CustomerName,3) & "-" & Width & "-" & Length

As with other rules in this tutorial, Ampersands (&) have been used to combine multiple values.

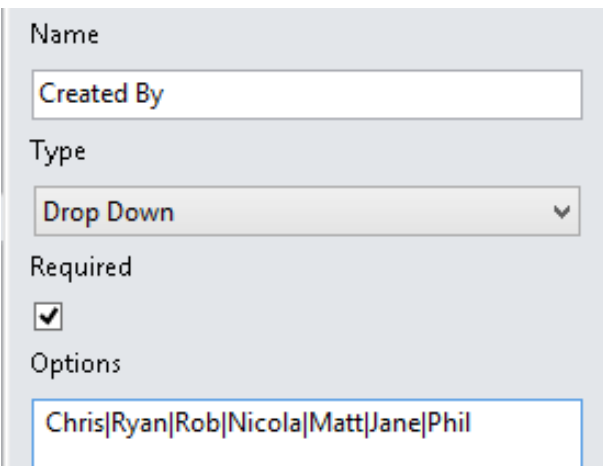
The Left() function has been introduced in the rule for the Drawing Number. This function takes the given value, and returns the number of letters specified, starting from the left.

Finally, for the Drawn By Custom Property, a new control needs to be created.

Click 'Back' to return to the Rules Summary then click the Form Creation button.



Click 'Add' to create a new control.



Give the control the name, 'Created By' and change the Type to Drop Down.

Each item in the Options field must be separated by a pipe bar (|).

Type in a series of names to populate your Drop Down Control.

Link the new Created By control to the Drawn By Custom Property.

Click the Rule Creation button and build the following rule for the Drawn By Custom Property:

Name	Rule
Drawn By	= CreatedBy

Click 'Back' to return to the Rules Summary page.

There are now 0 rules missing for the captured dimensions.

RUNNING THE PROJECT

Click 'Next' or 'Run' in the DriveWorksXpress navigation, which will take you to the 'Run' Task Pane.



Now that all of the rules are written for the project, new specifications can be created.

DriveWorksXpress

Enter the required values in the form below.
Complete the form to generate a new version of Scissor Lift Assembly.



Within the Run Task Pane, you will be able to see the Form Controls you created earlier in the project.

A screenshot of the DriveWorksXpress 'Run' task pane. At the top, there are two buttons: 'Defaults' and 'Last Used', both highlighted with red boxes. Below them is a form titled 'Scissor Lift Assembly'. The form contains several fields: 'Customer Name' with a text input containing 'Chris' and a tooltip that says 'Type in a value for Customer Name'; 'Order' with a text input containing '1234'; 'Length' with a spin box containing '500'; 'Width' with a spin box containing '300'; and 'Created By' with a dropdown menu containing 'Ryan'. At the bottom of the form are two buttons: 'Clear' and 'Create', with 'Create' highlighted by a red box.

Follow the Tool Tips and complete the Form.

All of the controls which require a value will appear with a pink background which will turn white once a suitable value is entered.

There are three methods of completing the Form. You can use the Default control values, Last Used values or enter new details.

Click 'Create'. DriveWorksXpress will now generate your new models and drawings.

A generation report is produced which shows activity on all of the driven values. Any errors will appear with a red cross, and all successful values will appear with a green tick.

To view the modified SOLIDWORKS Drawing File for this new model, open the folder where all the files have been saved to.



Congratulations! You have now completed this DriveWorksXpress Tutorial.

There is lots more downloadable content available at:

www.driveworksxpess.com

Now you've tried DriveWorksXpress, GO AUTOMATE your own SOLIDWORKS projects!