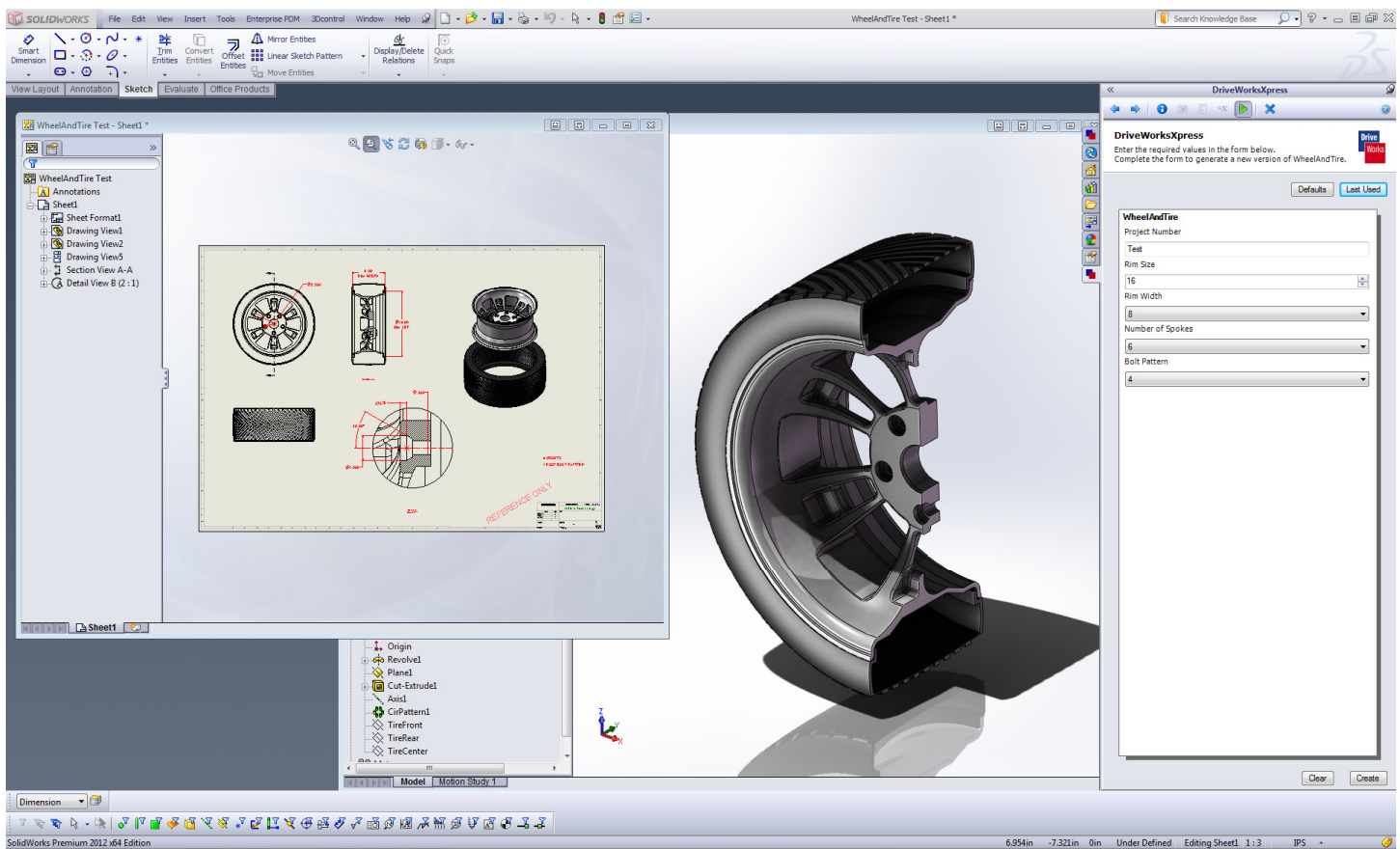THE Easy-to-Use Design Automation Choice for SolidWorks

DriveWorksXpress Hands on Test Drive

Presented by:

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Inflow Technology

Introduction

Start by installing the Models to a NEW FOLDER on your C: Drive. You should call this folder

C:\DWX_HOTD_SWW

This example will allow you to step through DriveWorksXpress from start to finish.
You will have the opportunity to follow a script, capture the features and dimensions that you want to change, create the user interface and assign rules.
At the end you will be able to specify a new Wheel set.

About
DriveWorksXpress

DriveWorksXpress is the easy-to-use design automation tool for SolidWorks Users. It is the Xpress version of DriveWorks - the SolidWorks Certified Gold Partner for Design Automation www.driveworks.co.uk
There are two aspects to working with DriveWorksXpress

1. Set up - Setting up your Project
- **Capture Mode**- Decide which parameters in your model you want to drive or change
- **Form Mode** - Create a simple form for entering input values
- **Rules Mode** - Define rules to determine how your model will behave and link to the inputs on your form
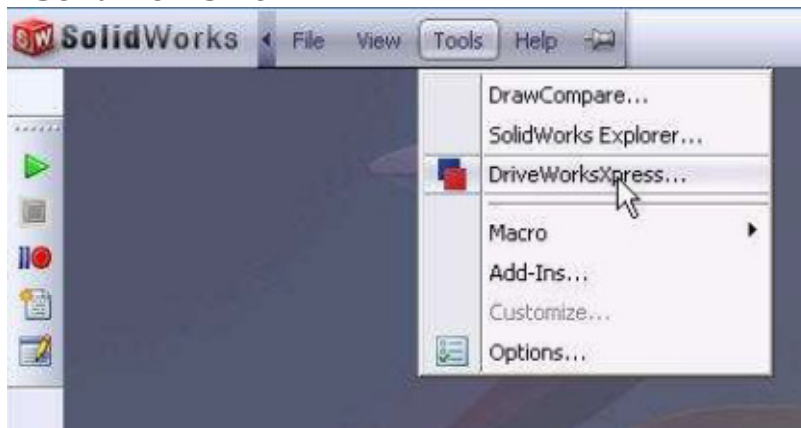## *2. Run - Running new models from the Project you have set up*
- Enter input values into your form
- Generate new models and drawings

Once a project has been set up, it can be **Run** over and over again to automatically create the new models and drawings
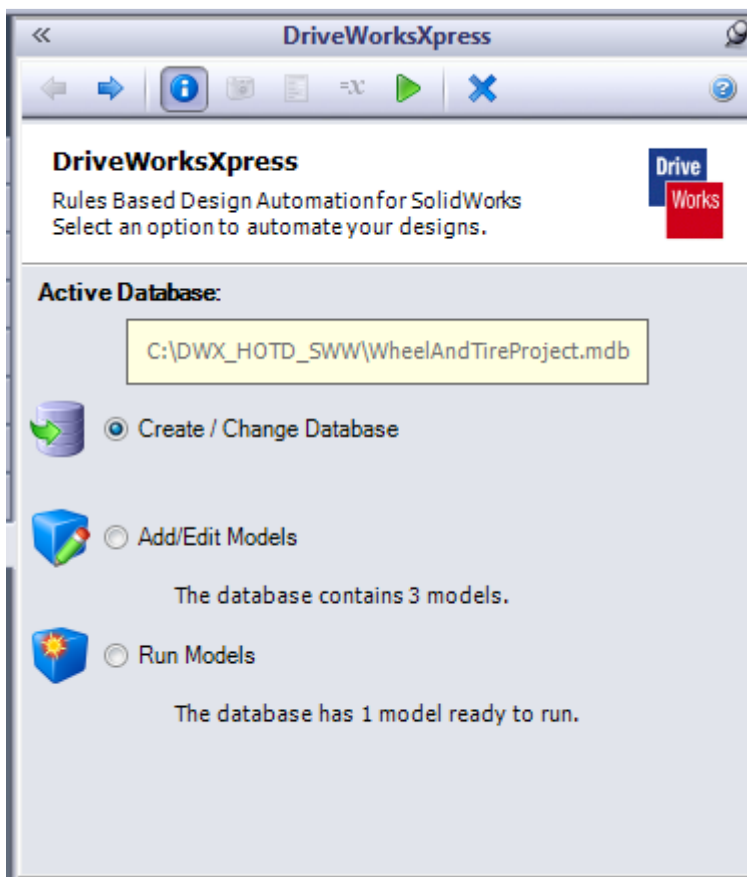
Let's get started

1. Launch SolidWorks 2012

2. Launch DriveWorksXpress

The DriveWorksXpress Welcome Screen Opens in the Task Pane

The Welcome Screen provides you with feedback on the options available to you, on the state of the project you are working on, or on the projects
you have already set up and that are ready to Run.

Note - DriveWorksXpress uses a database to store all the knowledge and information needed to generate the outputs, clones and specifications. When you open DriveWorksXpress for the very first time it creates a default database to store your work called **driveworksxpress.mdb**.

The DriveWorksXpress Welcome screen always displays the **Folder Path**
and **File Name** of the active database.

The Welcome Screen also displays feedback on options available

> **Create/Change Database**
> When DriveWorksXpress is launched **without a database loaded**, the only options are to use the default database OR to create a new one.
>
> **Add/Edit Models**
> This is only enabled once a database has been selected.
>
> **Run**
> This is only enabled once you have the completed the Administration/Set up of your project. You can then generate new models using the Input form you have created, and based on the parameters you have already captured and the rules you have assigned rules.

For this exercise we will create a new database

3. Create a New Database

Check the Radio Button ⊙ against Create/Change database

Click **Next**

A Windows dialog box appears

Browse to where you want to create your new database (.mdb file)

C:\DWX_HOTD_SWW

For this exercise - name the database **WheelAndTireProject**
Type the **New** File Name into the File Name Field and click **Open**

.... NOTE: Database Names must be unique.

If a name is not unique a warning message will appear advising that an existing database will be overwritten. You may choose YES or NO to overwrite.

NOTE: Once you start using DriveWorksXpress regularly you may choose to store multiple projects in a single database or in different databases.

4. Add your SolidWorks Models to DriveWorksXpress

DriveWorksXpress lets you create variants of your CAD assemblies, parts and drawings. To do this you need to open your CAD file and add the relevant models to DriveWorksXpress.

As this a new database, you only have one choice, which is to browse for a new model to add to DriveWorksXpress.

Check Next
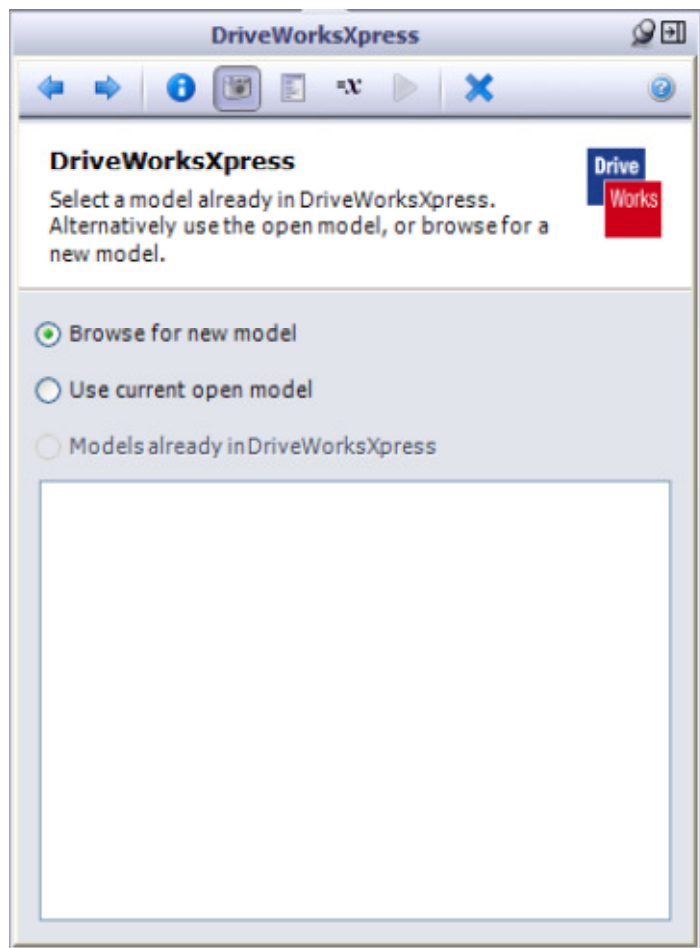
A new window will open so you can **Browse** to the location of your file –
in this case **c:\DWW_HOTD_SWW**

Now select the top level SolidWorks Assembly file **WheelAndTire.SLDASM**

File name:	WheelAndTire.SLDASM		Open
Files of type:	SolidWorks Assembly (*.sldasm)		Cancel
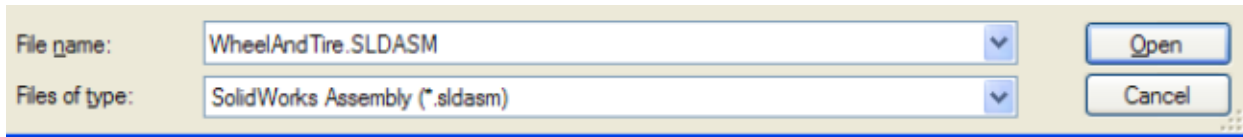
Click **Open**

5. Capture Mode

Select which models you want to drive

DriveWorksXpress now displays the assembly in the "Captured Models" window.

Captured Models

WheelAndTire
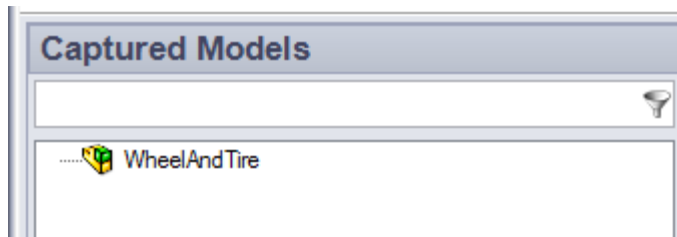
Note: The Model will now also be open and visible in SolidWorks behind the DriveWorksXpress wizard

Select the "Captured Assembly Structure" button
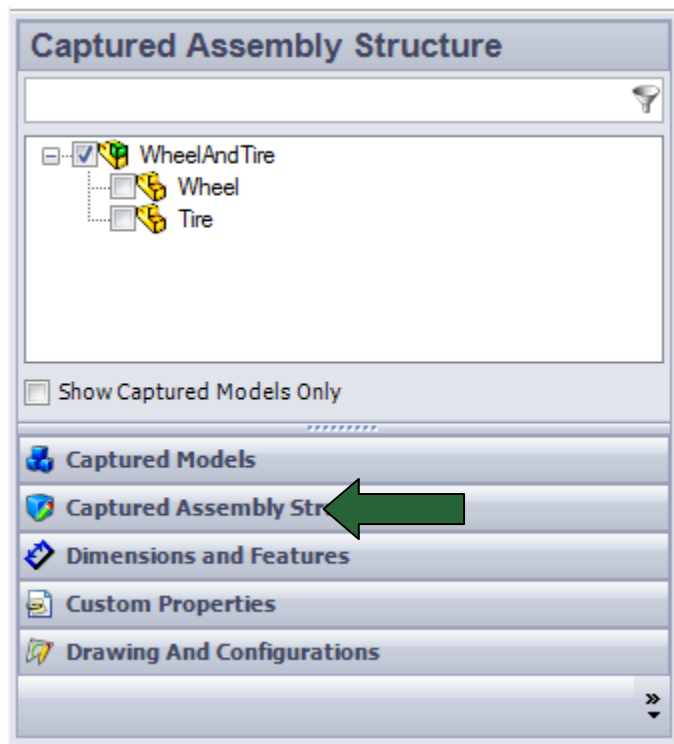
Double Click the **+** symbol by the Assembly to expand / collapse the tree.

Select the parts/assemblies that you want DriveWorksXpress to control by checking the box next to each item

The top level assembly (WheelAndTire) will already be checked.

For this exercise check all of the remaining models: -

-Wheel
-Tire
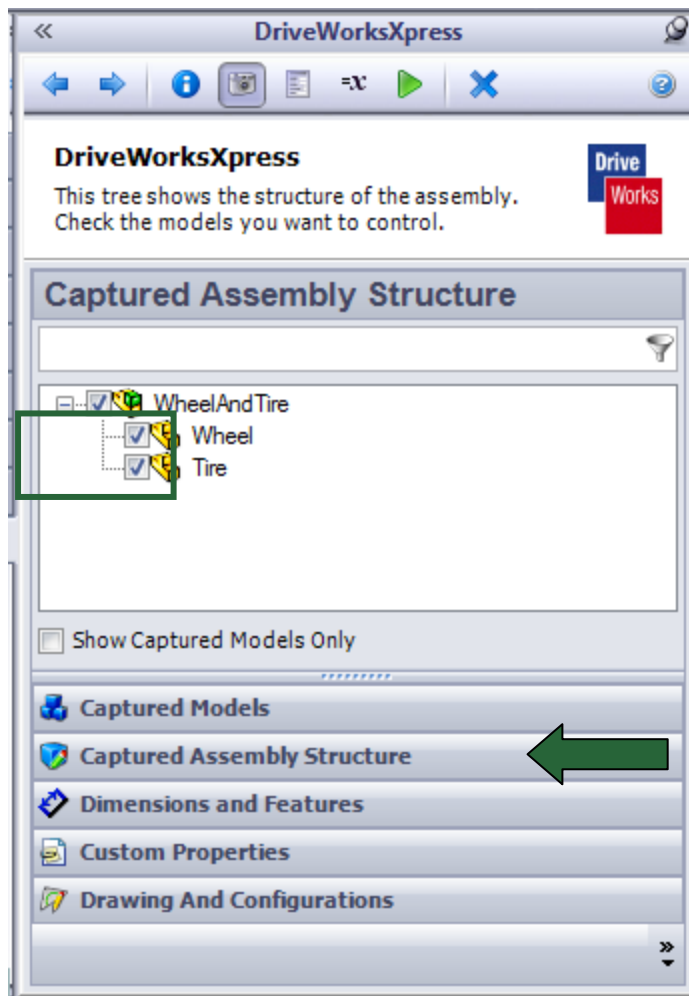
Click **Next**

5.1 Capture - Dimensions and Features

DriveWorksXpress lets you capture specific Dimensions and Features so that you can apply rules and / or so you can specify new models based on the original criteria using your own input forms. A parameter is captured by selecting the model that the parameter exists in from the tree view and clicking **Add**

Follow the steps below which will guide you through each parameter that is to be captured for this Wheel and Tire.

Capture Wheel

Select the Wheel from the tree view and click **Next**

Select **Dimensions and Features**
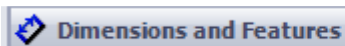
Capture the **Radius** of the **Wheel**

In the SolidWorks Feature Manager double click the icon of the feature that contains the dimension to be captured (for the Radius it is the **feature** named **Revolve1**. The dimension will appear on the model.
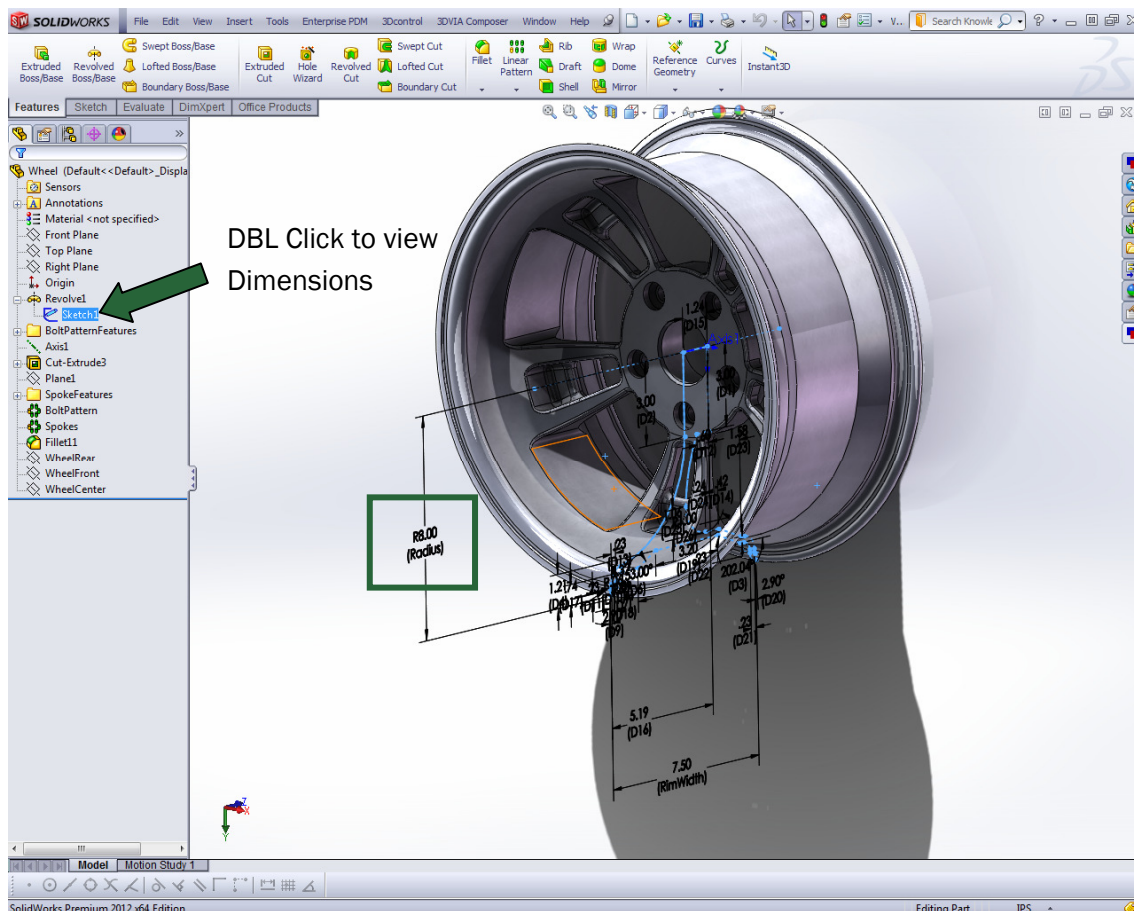

Select the dimension from the model (by clicking on the dimension value).

The **SolidWorks Name** of the dimension will appear as SolidWorks Selection in the Parameter Manager
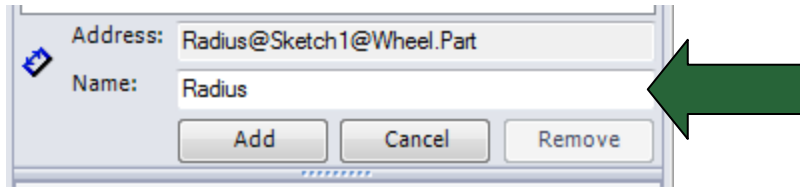
Enter a **New DriveWorks Name**

This can be something more meaningful (eg **Radius**)

Click Add
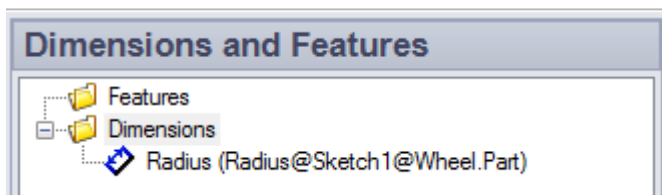
The captured parameter will also now appear in the Parameter list at the bottom of the Parameter manager.

Next **Capture** the **Width** of the **Wheel.**

In the SolidWorks Feature Manager double click the feature that contains the dimension to be captured (for the **RimWidth** it is the **Revolve1**). The section dimension will appear on the model
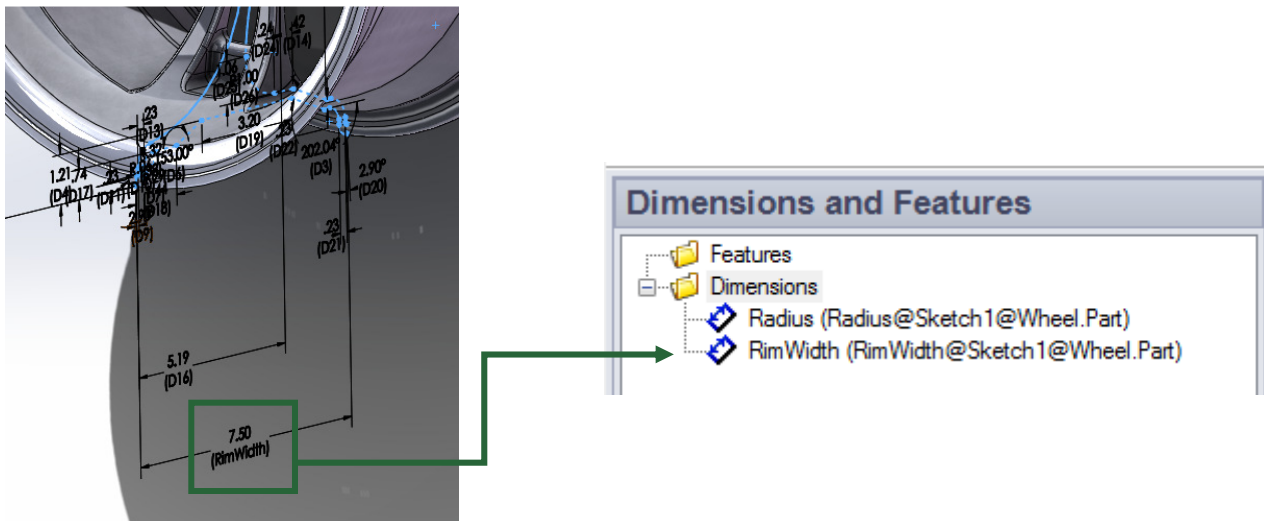
Click on the dimension and then, in the DriveWorksXpress Parameter Manager, enter a new name – **RimWidth**

Next **Capture** the **Number of Spokes** of the **Wheel.**

In the SolidWorks Feature Manager double click the feature that contains the dimension to be captured (for the **Number of Spokes** it is the **Spokes** feature). The instance dimension will appear on the model

Click on the dimension and then, in the DriveWorksXpress Parameter Manager, enter a new name – **NumberOfSpokes**
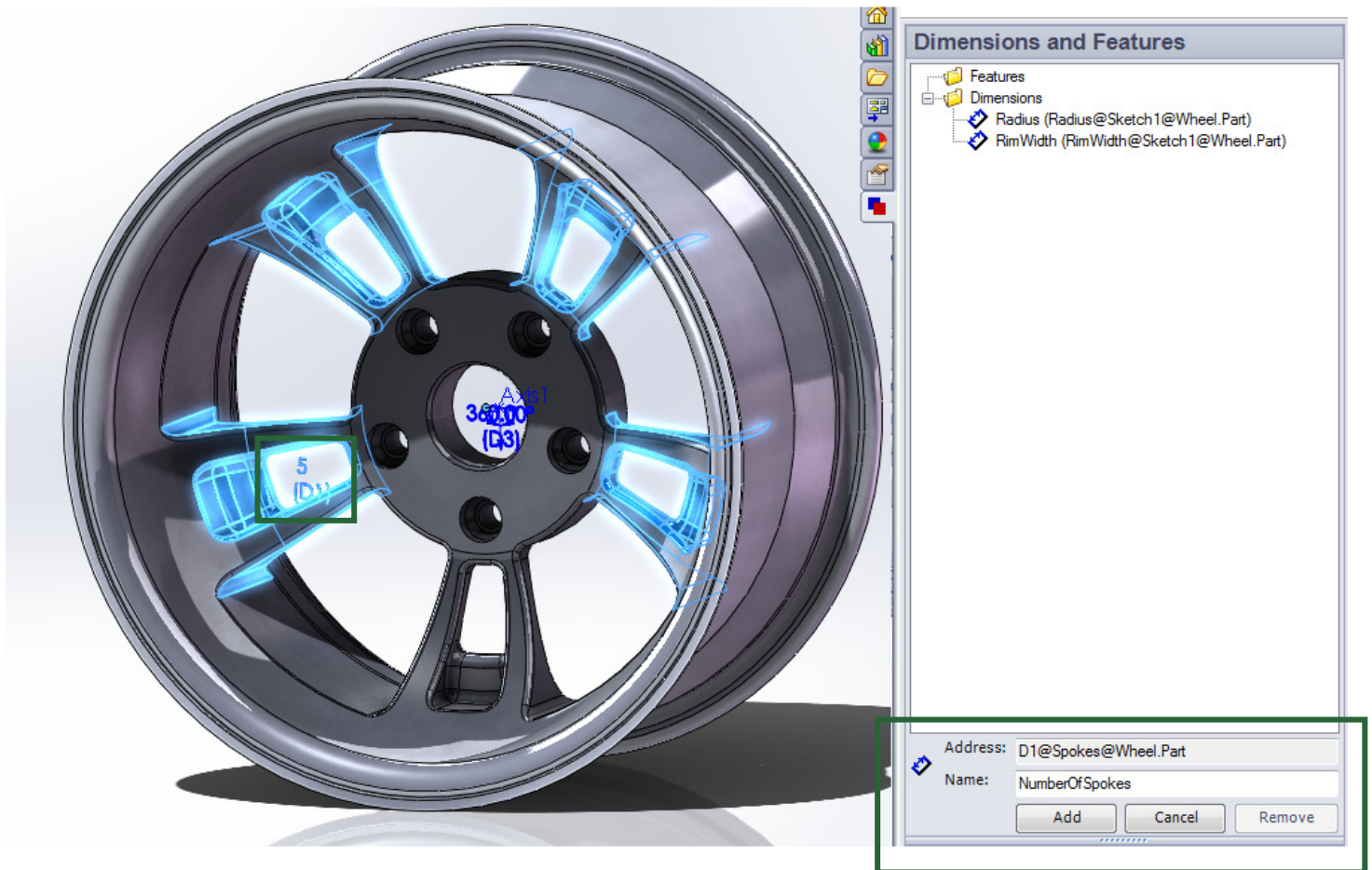
Click Add

On your Own: Add a Dimension to drive the Bolt Pattern

Capture the Tire Model

Select **Previous**

Select the Tire from the tree view and click **Next**
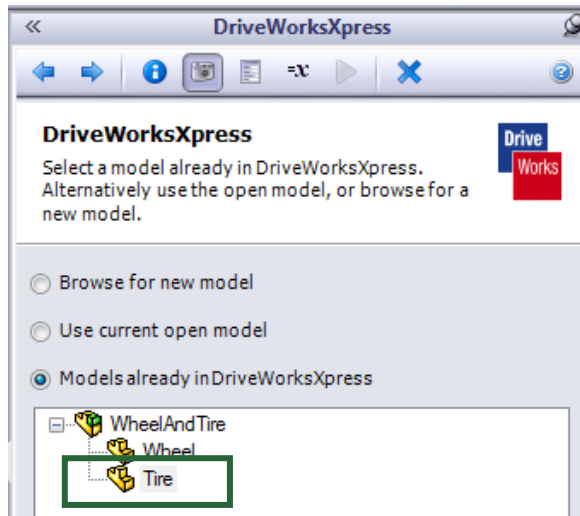
Select **Dimensions and Features**
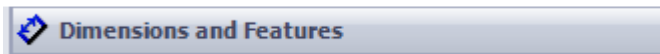
Capture the **radius** of the **Tire**

In the SolidWorks Feature Manager double click the feature that contains the dimension to be captured (for the tire radius it is in the feature Revolve1)

The Tire radius dimension will appear on the model.
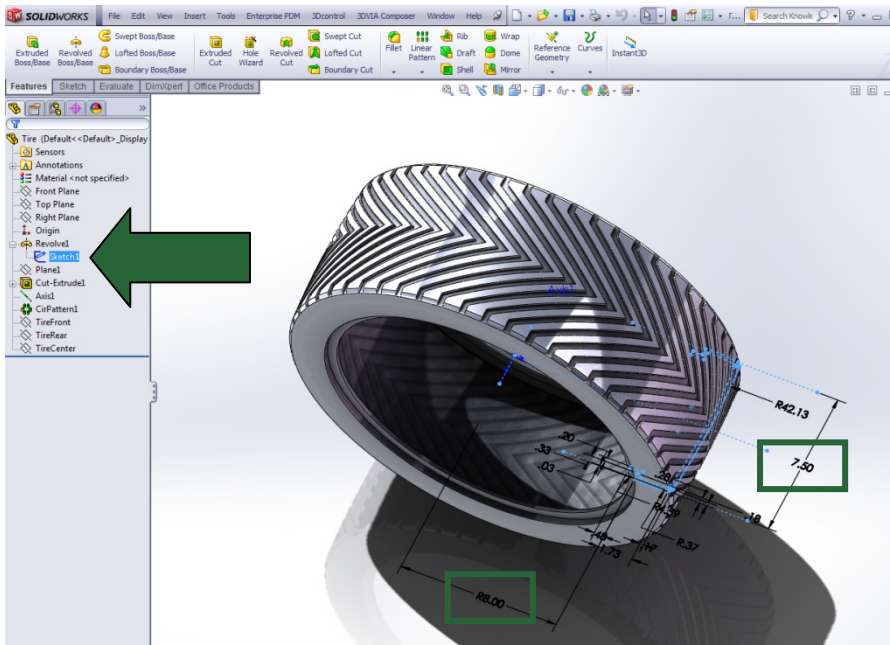
Click on the dimension that controls the radius and, in the parameter manager, enter a new DriveWorks Name –**RimRad**

Click **Add**

Next **Capture** the **Width** Dimension

Give this a DriveWorks Name **RimWidth**

Click **Add**

5.2 Capture - Custom Properties

DriveWorksXpress can also drive values to Custom Properties:

Select the Properties TAB to start capturing **Custom Properties**
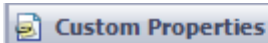
Custom Properties

This will drive the custom properties for the active model.

These are useful for linking to annotations in drawings to drive text on drawing borders or notes. They are also useful for acting as placeholders where complex rules can be broken down into more manageable portions.

From the Assembly Tree, double Click on WheelAndTire to expand the tree, if it is not already expanded.

Highlight the WheelAndTire Assembly in the tree view.
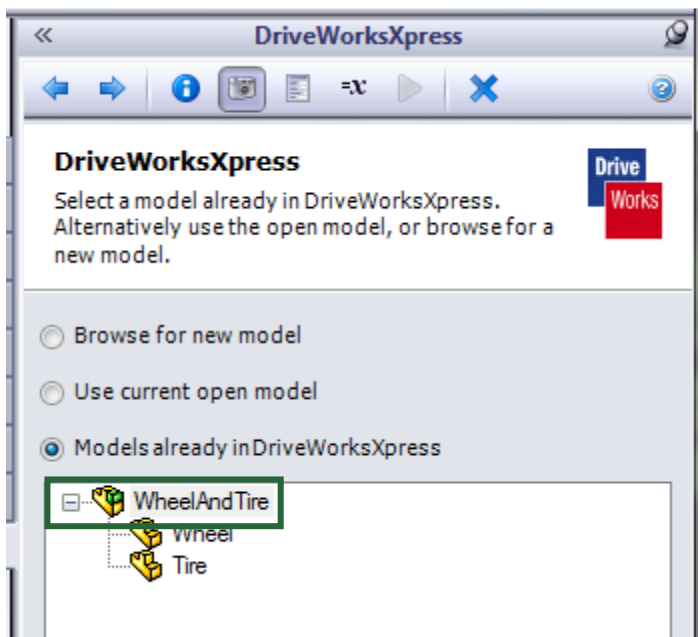
click **Next**

Select the Properties TAB to start capturing **Custom Properties**
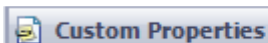
Custom Properties

Any existing Custom Properties will appear in the left hand pane under the window Non-Captured Properties

Select all the available custom properties

5.3 Capture Drawings

As well as generating new models DriveWorksXpress can also produce new drawings of the model. This is done by associating a previously produced template drawing of the master model.

To add a drawing, highlight the WheelAndTire top level assembly from the tree view and **Browse** to the location of the template drawing

Select **WheelAndTire.SLDDRW** and click **Open**
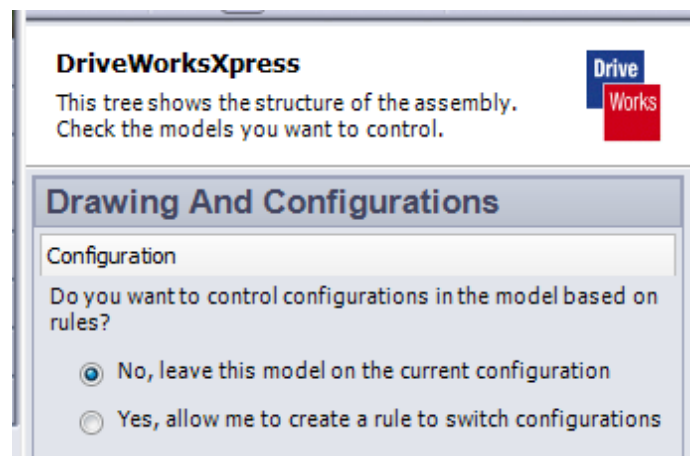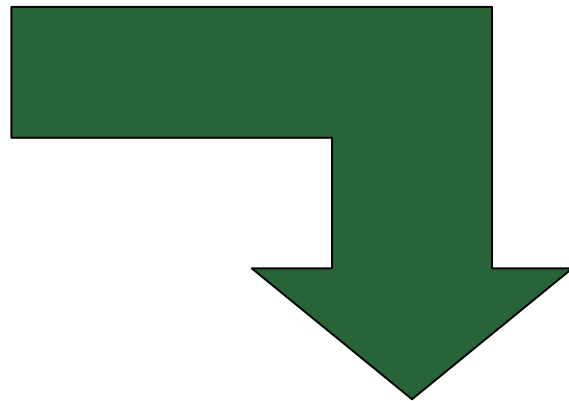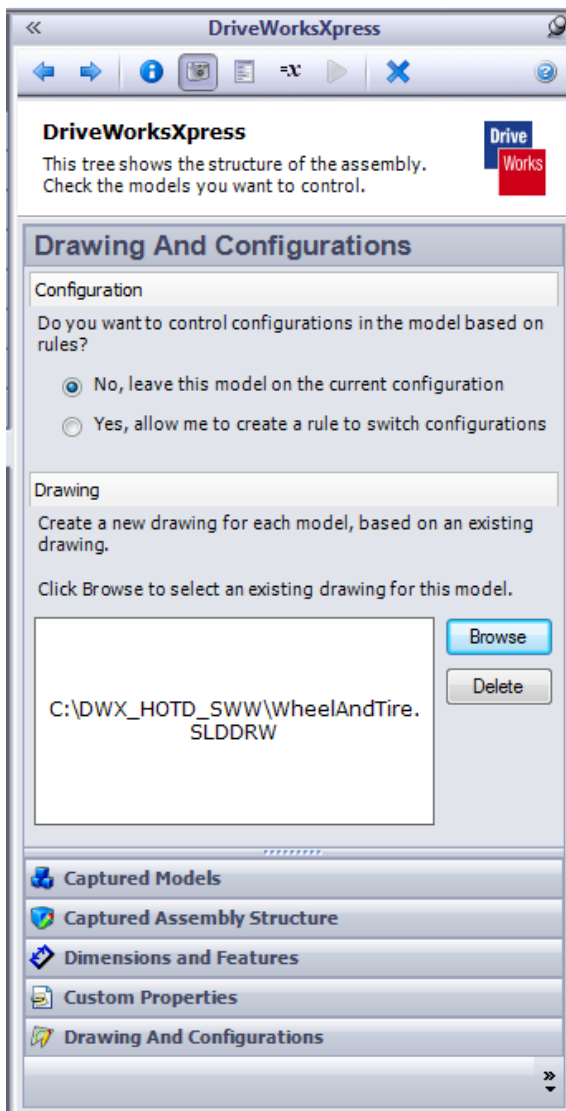
5.4 .Capture Configurations

DriveWorksXpress allows you to build a rule to switch the configuration of your model to any other **existing** configuration. In this example we are not capturing any configurations.

click **Next – This Ends Capture Mode**

6 Input Form Mode

You now need to 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 parts and drawings based on the rules you set and values you enter.

Start by adding the input controls

You can use the following form controls types in DriveWorksXpress

Text Box, Numeric Text Box, Drop Down, Spin Button, Check Box
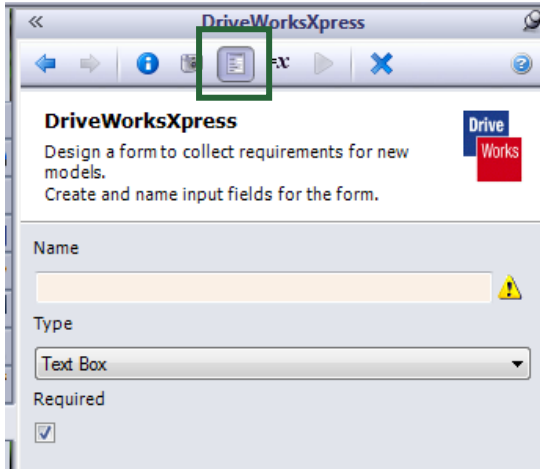
6.1 Order Number

Input Name = Order Number

Input Type = Text Box

Type Project Number as the name for the control in the Input Name field.

Use the Drop Down to select the control type.

Note: The **Required** checkbox denotes that an input must be entered

Click **Next** to register the control and display the Control List
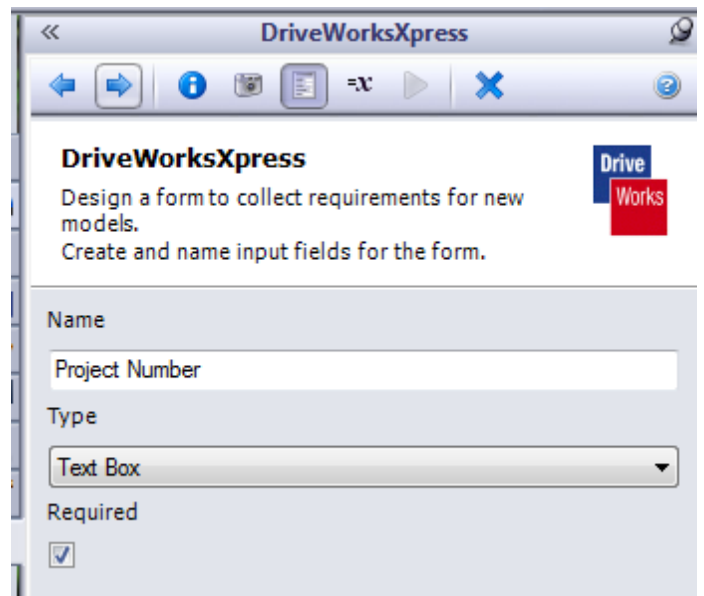
Repeat for the following input controls by clicking **Add** on the Control List
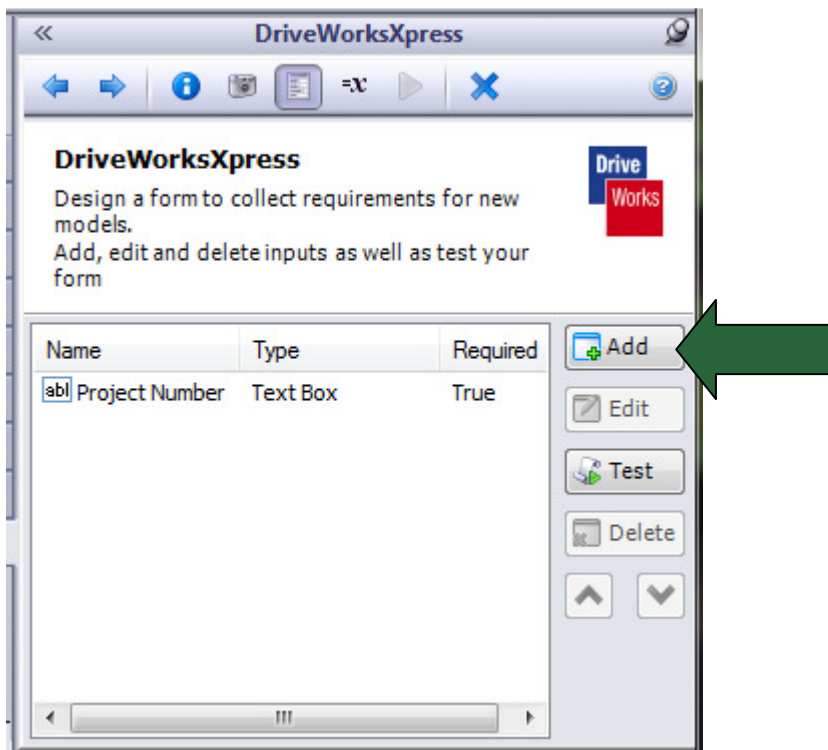
6.2 Rim Size

Input Name = Rim Size

Input Type = Spin Button

6.3 Rim Width

Input Name = Rim Width

Input Type = Drop Down

6.4 Number of Spokes

Input Name = Number of Spokes

Input Type = Drop Down

6.5 Bolt Pattern (optinal)

Input Name = Bolt Patterm

Input Type = Users Choice

Name

Bolt Pattern

Type

Drop Down

Required

☑

Options
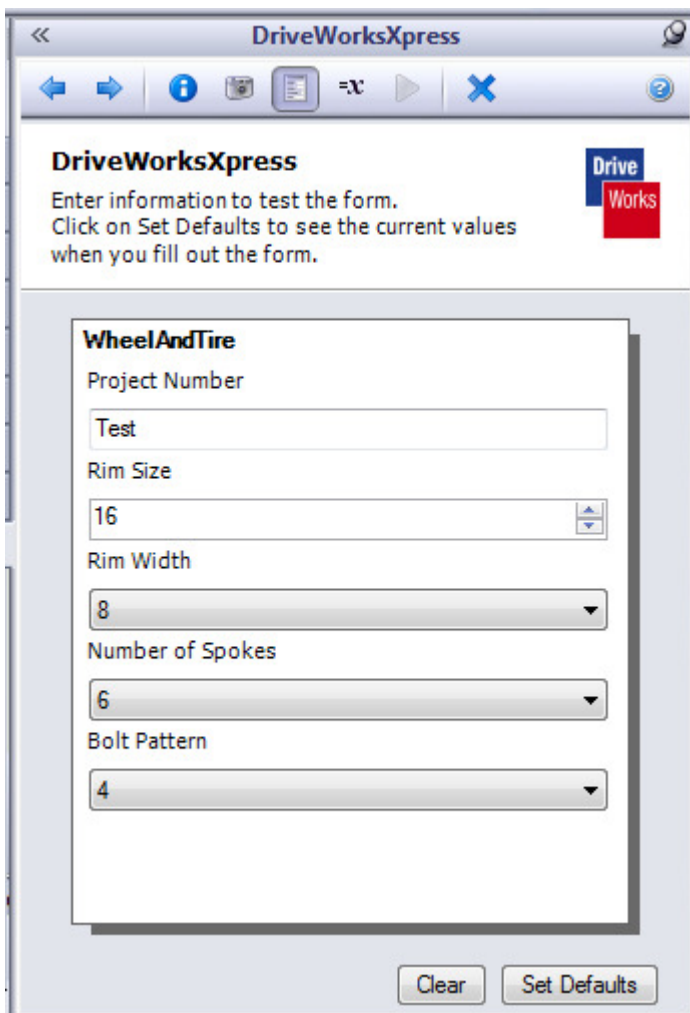
4
5
6

Name

Bolt Pattern

Type

Spin Button

Minimum Value

4

Maximum Value

6

Increment

1

6.6 Test the Form

Once all the required controls have been added, you have the option to test the form.
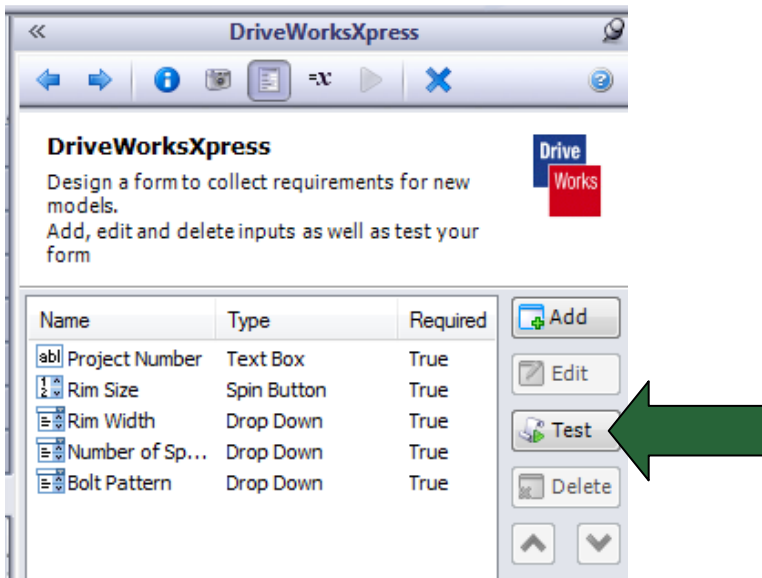
Enter test values to check that items such as spin buttons and drop downs have the information you require. You can even set default values.
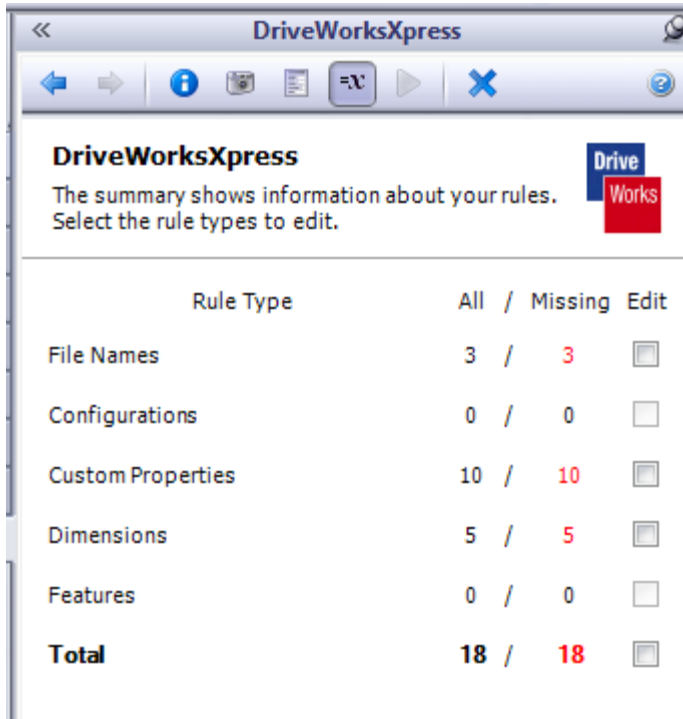

Click **Next (x2)** to proceed to the **Rules** Tab to set up and build your rules

7 Rules Mode

DriveWorksXpress lets you use Excel syntax to build rules. Rules tell driveworks how to link the input from the form to the captured models.

A summary appears of the rules by type that need building.

Checking an Action, against one of the Rule Types, 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).

7.1 File Name Rules (3)

Start by building rules for File Names.

Edit File Name Rules
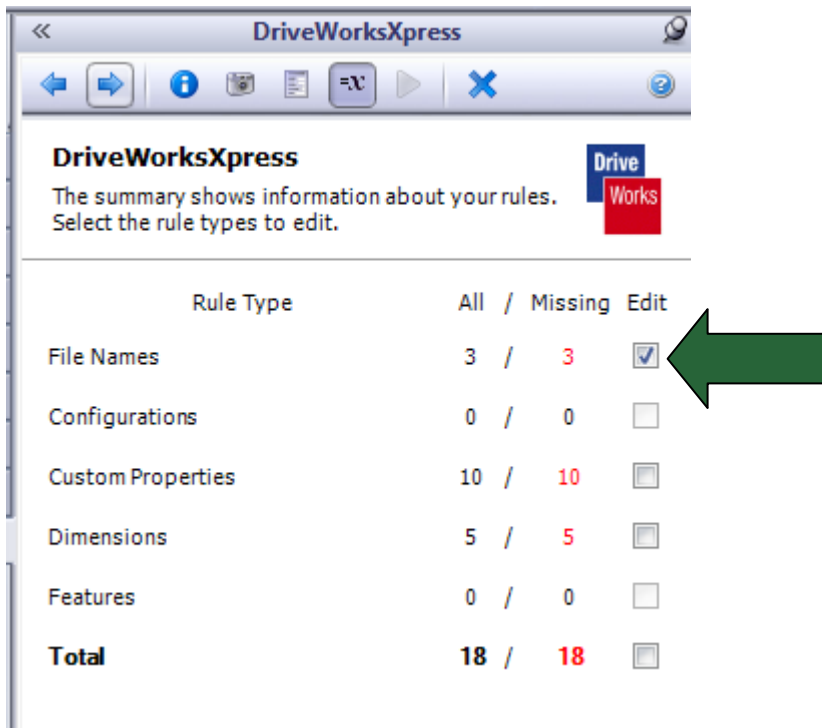
Click Next

Note: By default the next screen displays only the files with Missing Rules and which need building. The file icon includes a red cross where rules are missing. To
view all Files in the category uncheck the box

You can choose to view your Files as a List or as a Tree View

View List View Tree

The **Default View** is **View List**
Note: To build a rule highlight and double click the file.
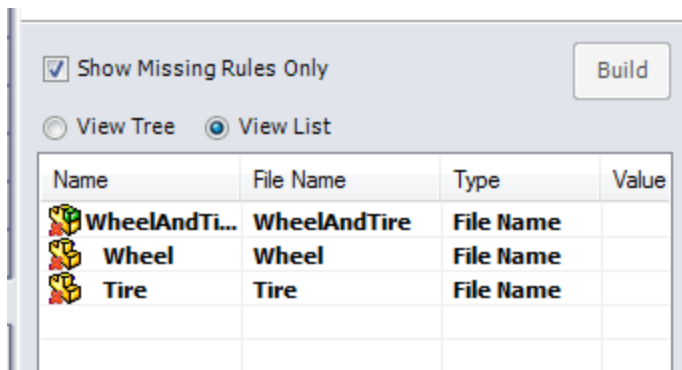
1. Build Rule for Wheel and Tire File Name

Use the value **Order Number** so that every time a new WheelAndTire project is created it will be named based on the Order Number.

Highlight **WheelAndTire. Double Click –** the DriveWorksXpress Rules
Builder Wizard appears.
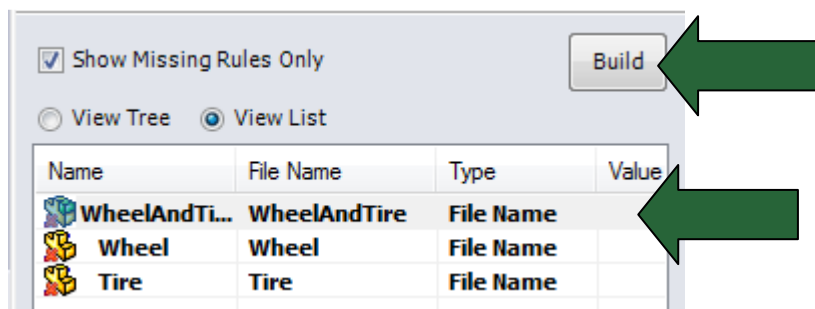
Note: You can use the tabs – **Inputs | Recent | Math | Logic**
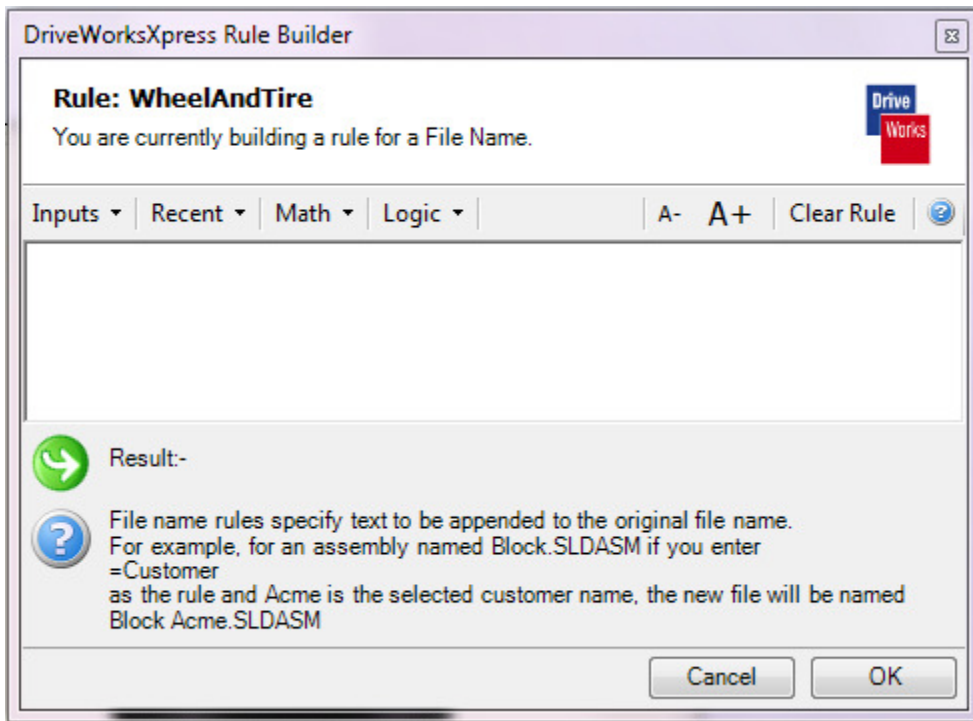
Note –The items listed in Inputs are all the controls that you have created for entering inputs on your form.

Map these inputs to your parameters to build your rules.

Click on the **Input Value Project Number**
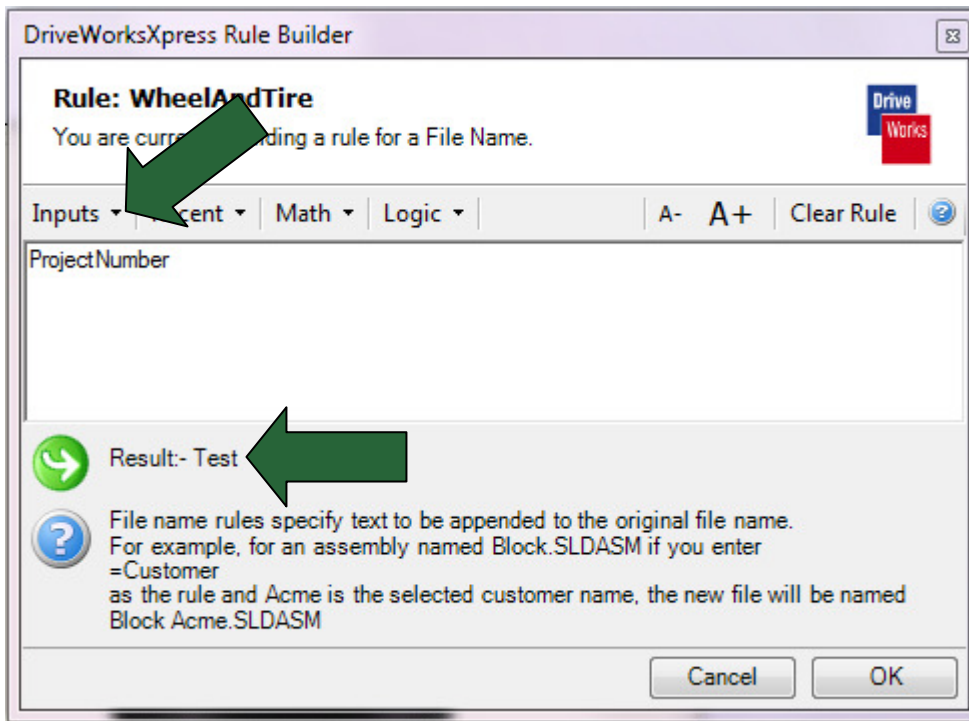
The effect of this will be to append the Order Number to the WheelAndTire file name when you run your new specification. In the Result area of the rule builder, you will see the default value you set during the test.

Click OK

Continue by adding rules for the remaining 2 File Names

NOTE: In Rule Building the symbol **&** is used to link strings.

2. **Build Rule for the Wheel File Name**

Build the rule as follows
ProjectNumber & " " & RimSize & " " & RimWidth & " " & NumberofSpokes
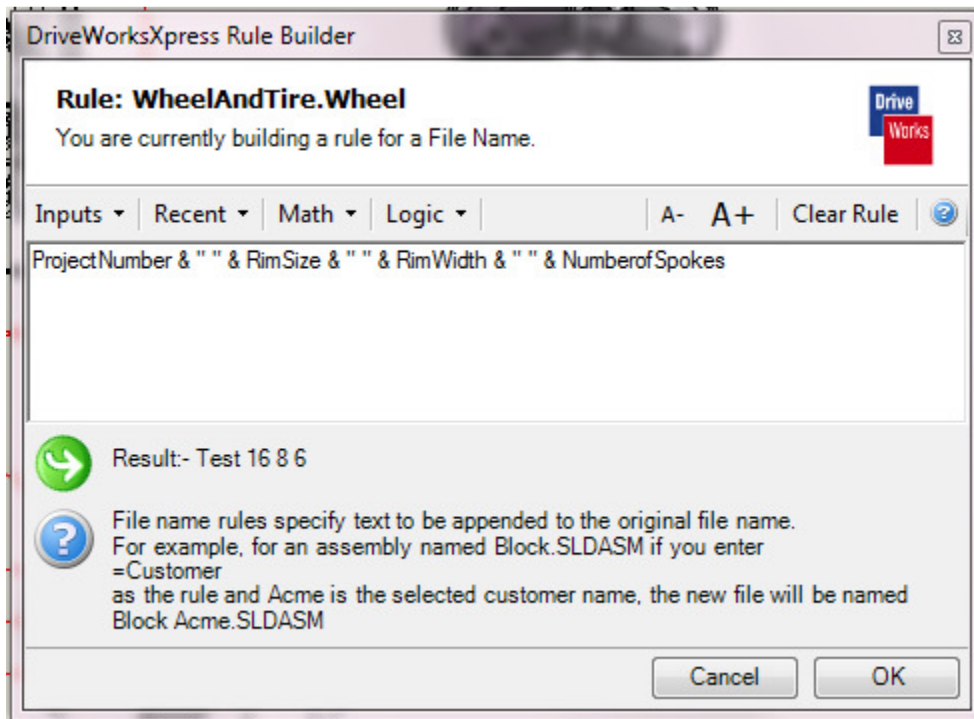
Note - You can add and use Quick Text for frequently used text strings for example "wide", "high", "Opening Width" and so on.
Just Add your text to the Recent Menu by selecting **Change Quick Text**.

3. **Build Rule for Tire Name**

Build the rule as follows (note the space in front of *Tire*)

Click **OK**

Click **Previous** to return to Rules Summary
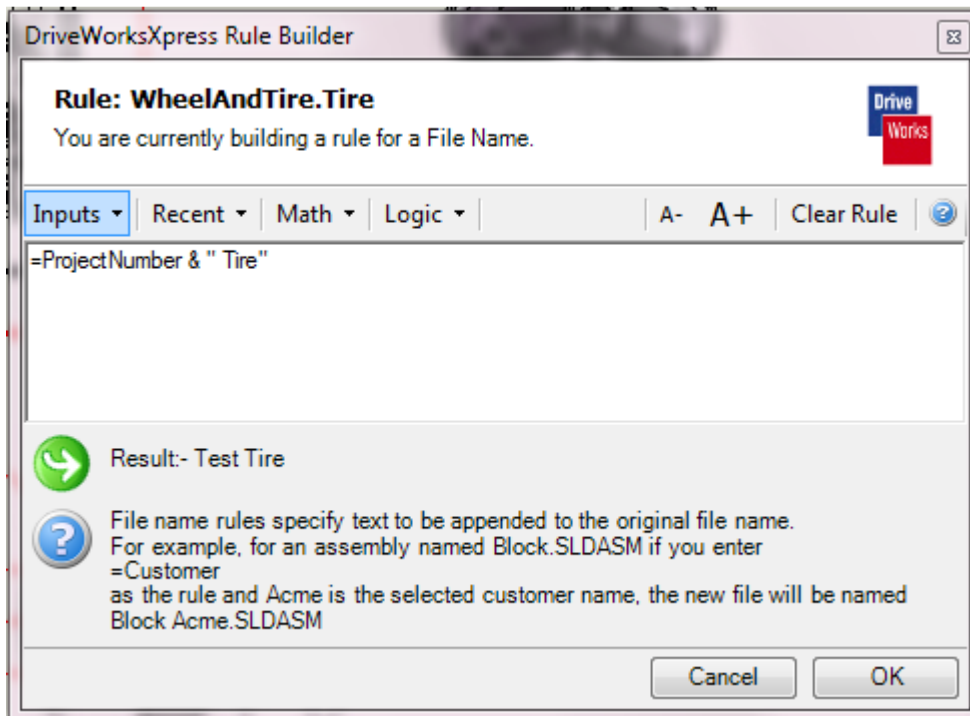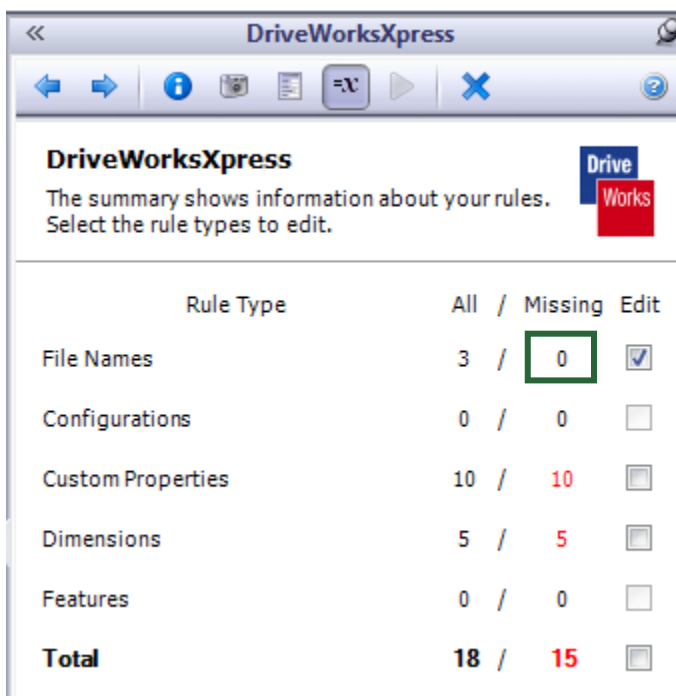
You will see that the **File Names** rule are complete

Un-check Edit Configuration Rules

Check Edit Custom Property Rules

Click Next
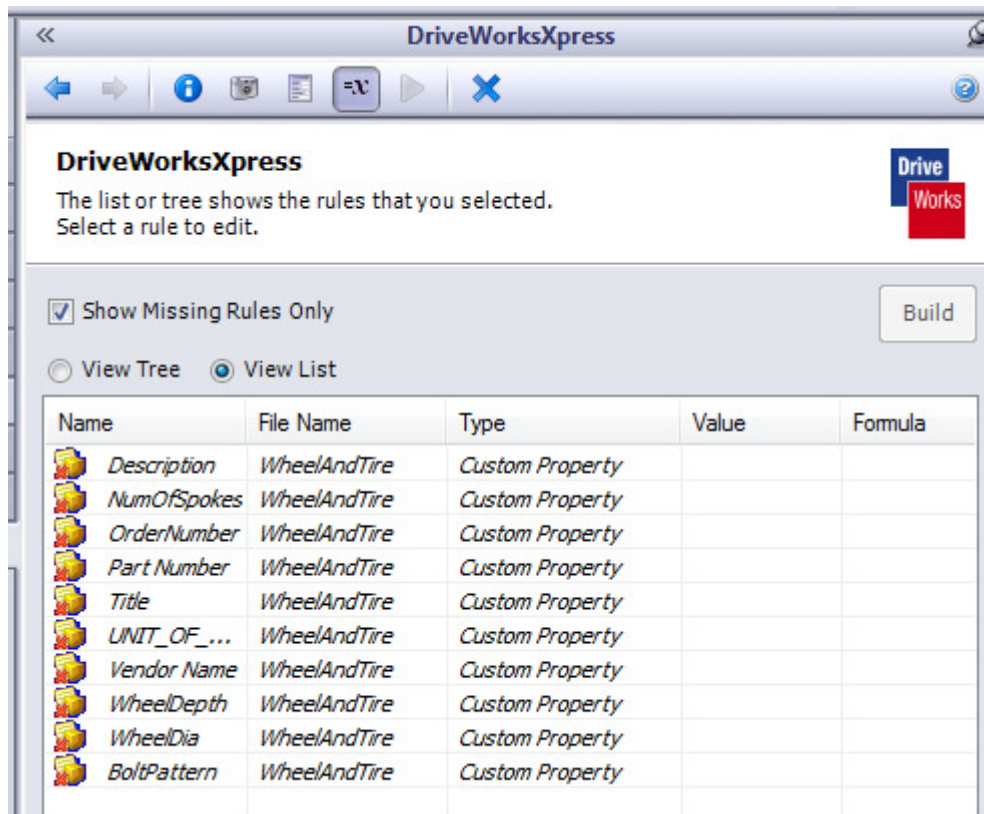
1. **Build Rule for OrderNumber**
Use the input value *ProjectNumber*
2. **Build Rule for WheelDia**
Use the input value *Rim Size*
3. **Build Rule for WheelDepth**
Use the input value *RimWidth*
4. **Build Rule for NumOfSpokes**
Use the input value *Number Of Spokes*
5. **Build Rule for BoltPattern (optional)**
Use the input value *BoltPattern*

Click **Previous** to return to the Rules Summary

7.3 Build Rules for Dimension (5)

Un-check Custom Property Rules

Check Edit Dimension Rules

Click Next

1. **Build Rule for Radius Dimension of the Wheel**
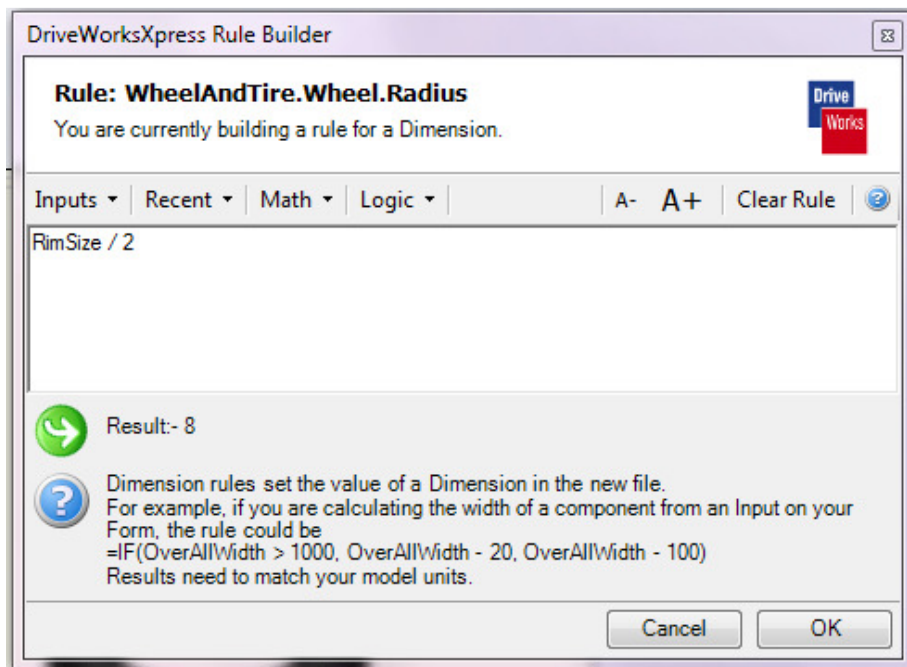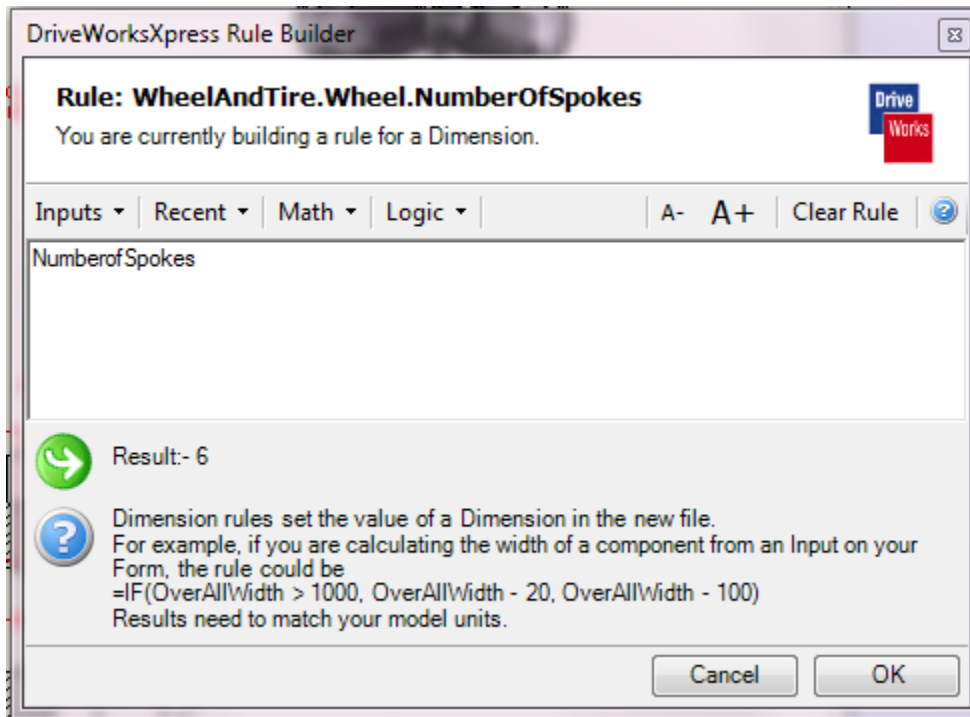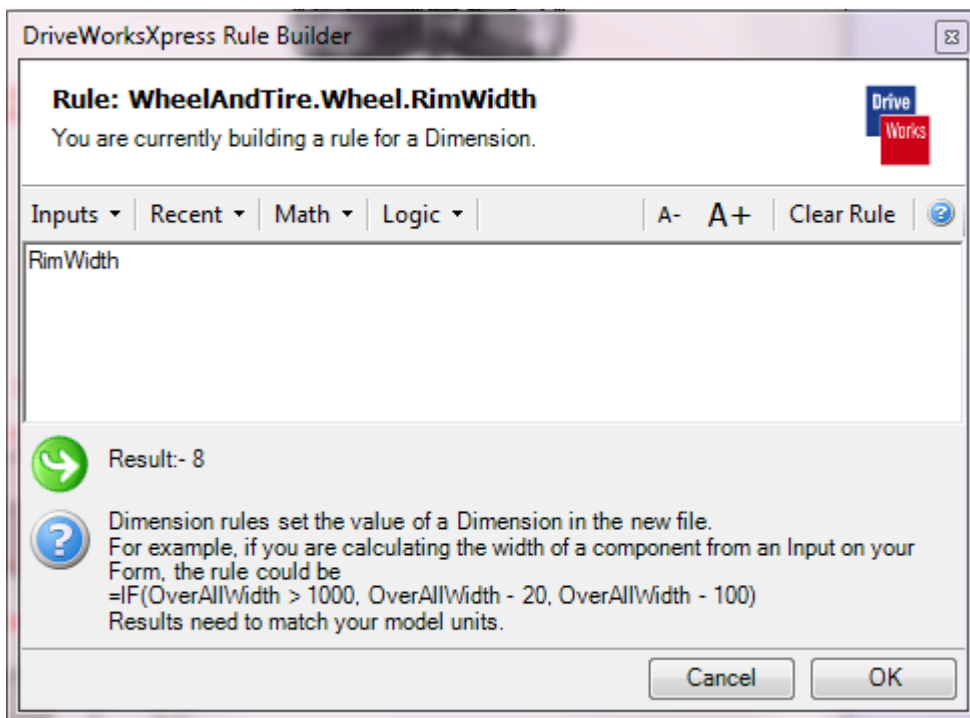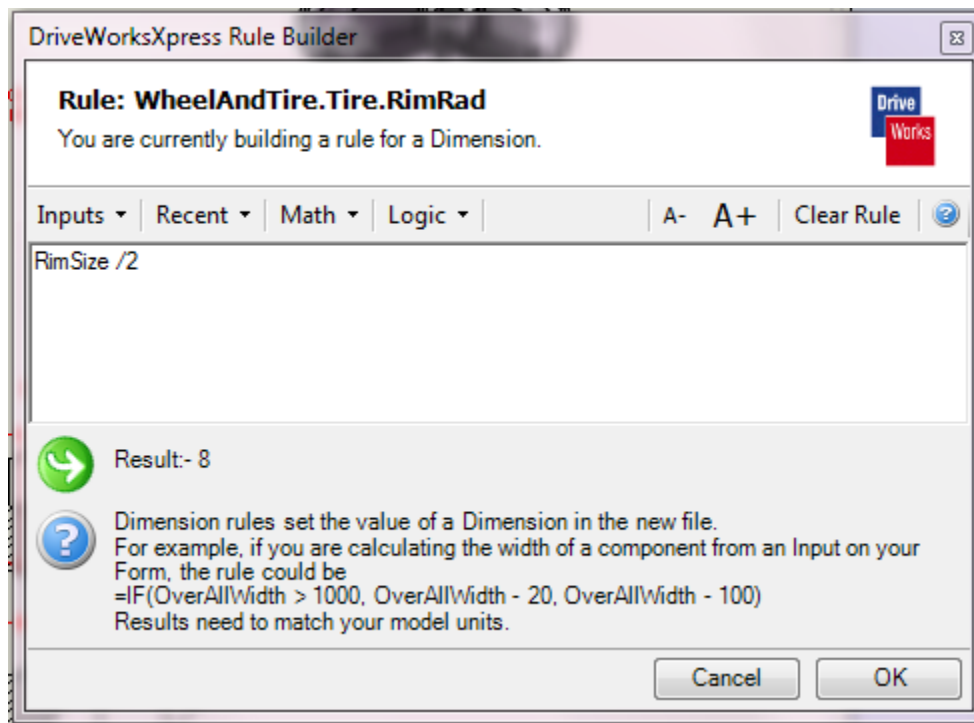

We divide by 2 because the input value is a diameter.

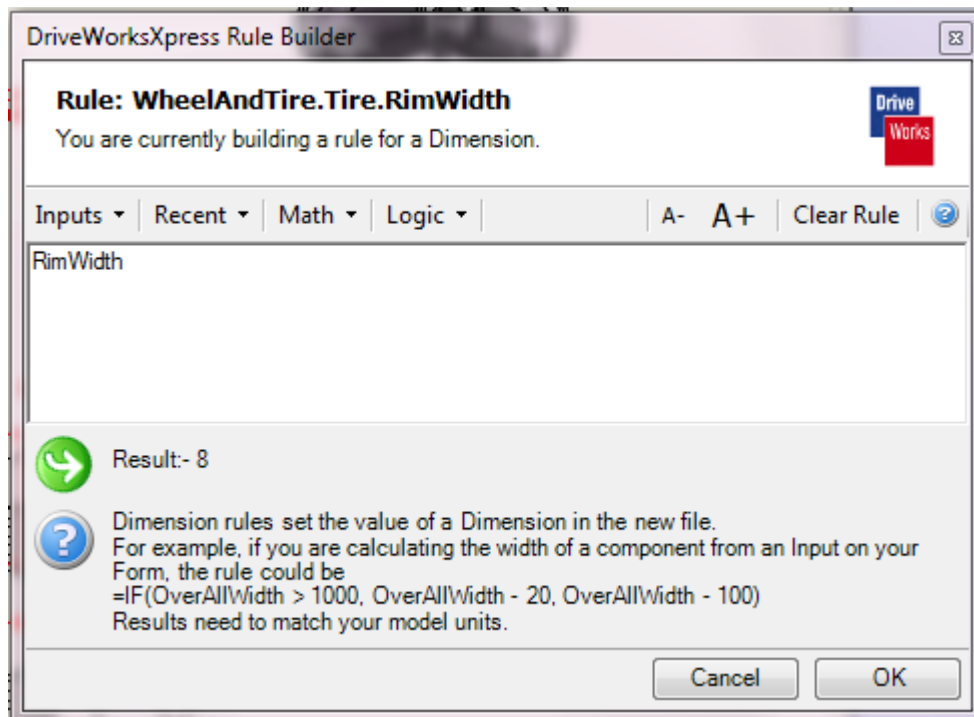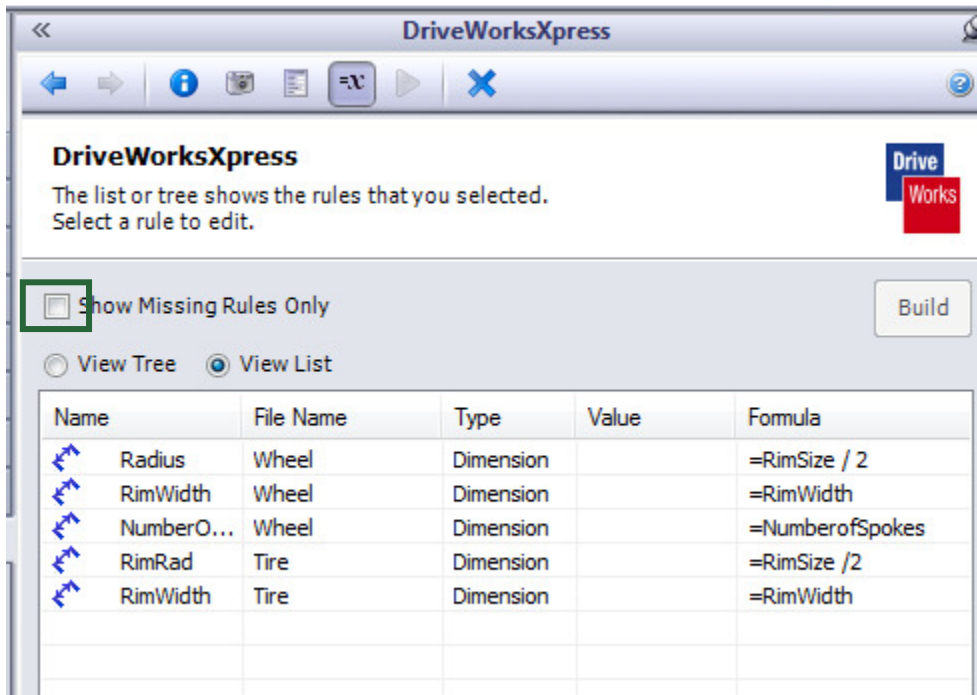2. **Build Rule for the Wheel Number Of Spokes**

3. Build Rule for Wheel Width

4. Build Rule for Radius of Tire

Rule: WheelAndTire.Tire.RimRad

You are currently building a rule for a Dimension.

Inputs ▾ | Recent ▾ | Math ▾ | Logic ▾ | A- A+ | Clear Rule

RimSize /2

Result:- 8

Dimension rules set the value of a Dimension in the new file. For example, if you are calculating the width of a component from an Input on your Form, the rule could be
=IF(OverAllWidth > 1000, OverAllWidth - 20, OverAllWidth - 100)
Results need to match your model units.

We divide by 2 because the input value is a diameter.

5. Build Rule for Width of Tire

Rule: WheelAndTire.Tire.RimWidth

You are currently building a rule for a Dimension.

Inputs ▾ | Recent ▾ | Math ▾ | Logic ▾ | A- A+ | Clear Rule

RimWidth

Result:- 8

Dimension rules set the value of a Dimension in the new file. For example, if you are calculating the width of a component from an Input on your Form, the rule could be
=IF(OverAllWidth > 1000, OverAllWidth - 20, OverAllWidth - 100)
Results need to match your model units.

*You have now assigned all the rules that you need to apply.*

But we cannot advance?

There are rules that have not been completed. All rules must be complete before we can run the project.
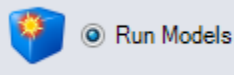
*Click **Previous** until Capture Mode is activated. We can navigate forward and back to make changes to the project. Uncheck the Customer Properties we do not need to drive*
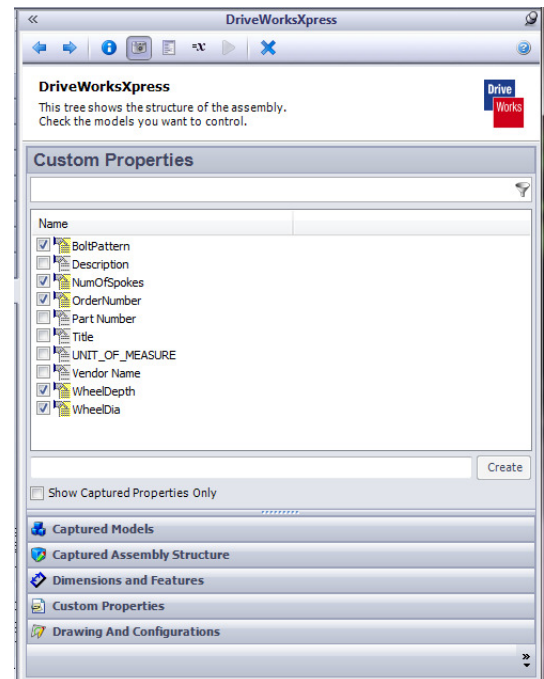
If you added the Bolt Pattern, this Rule will also need to be built.
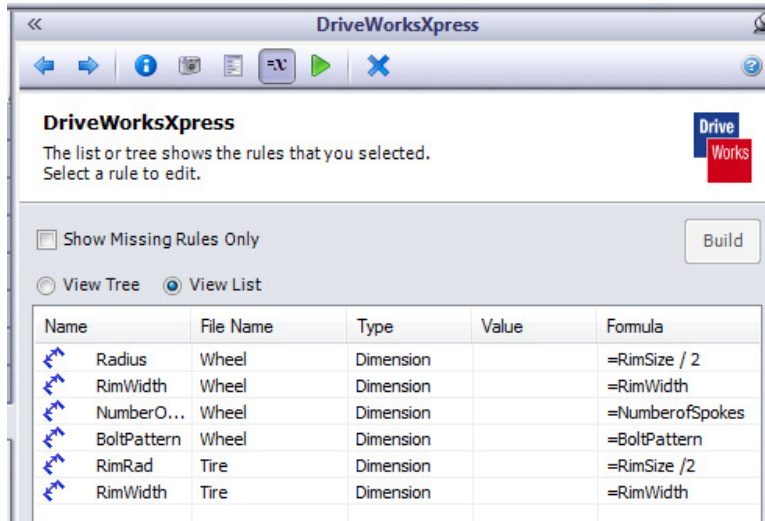
Click to **Run**

(If the is not available, select and

8 Run
The input form you created now appears. Follow the Tool Tips and complete the form.
You can use the default values, base your specification on one you created earlier OR clear the form so that you enter the details from scratch.
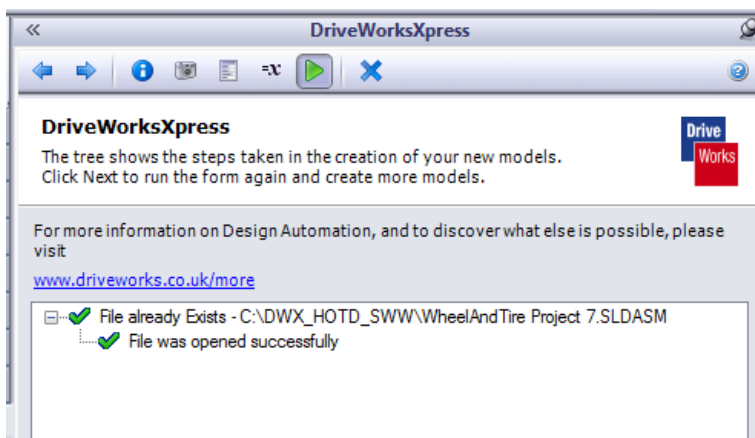
DriveWorks will now generate your new outputs.
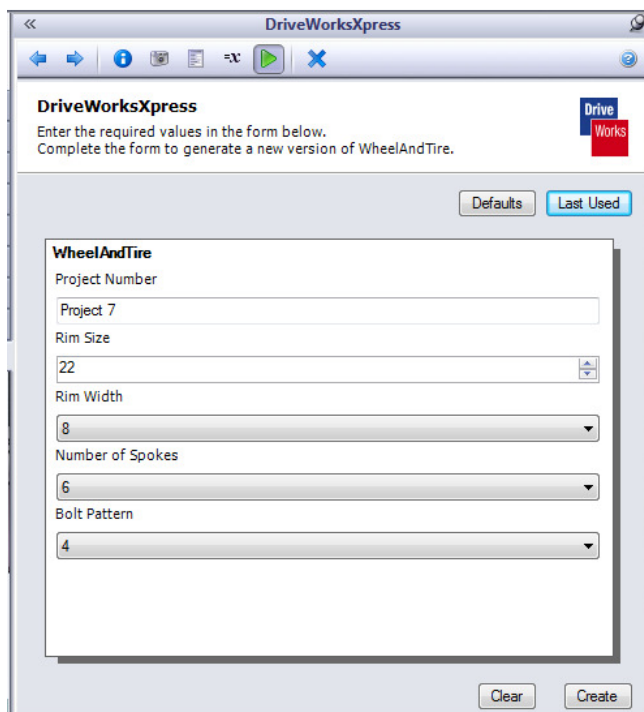
Once the project is complete, the new model will be displayed with the project log file.
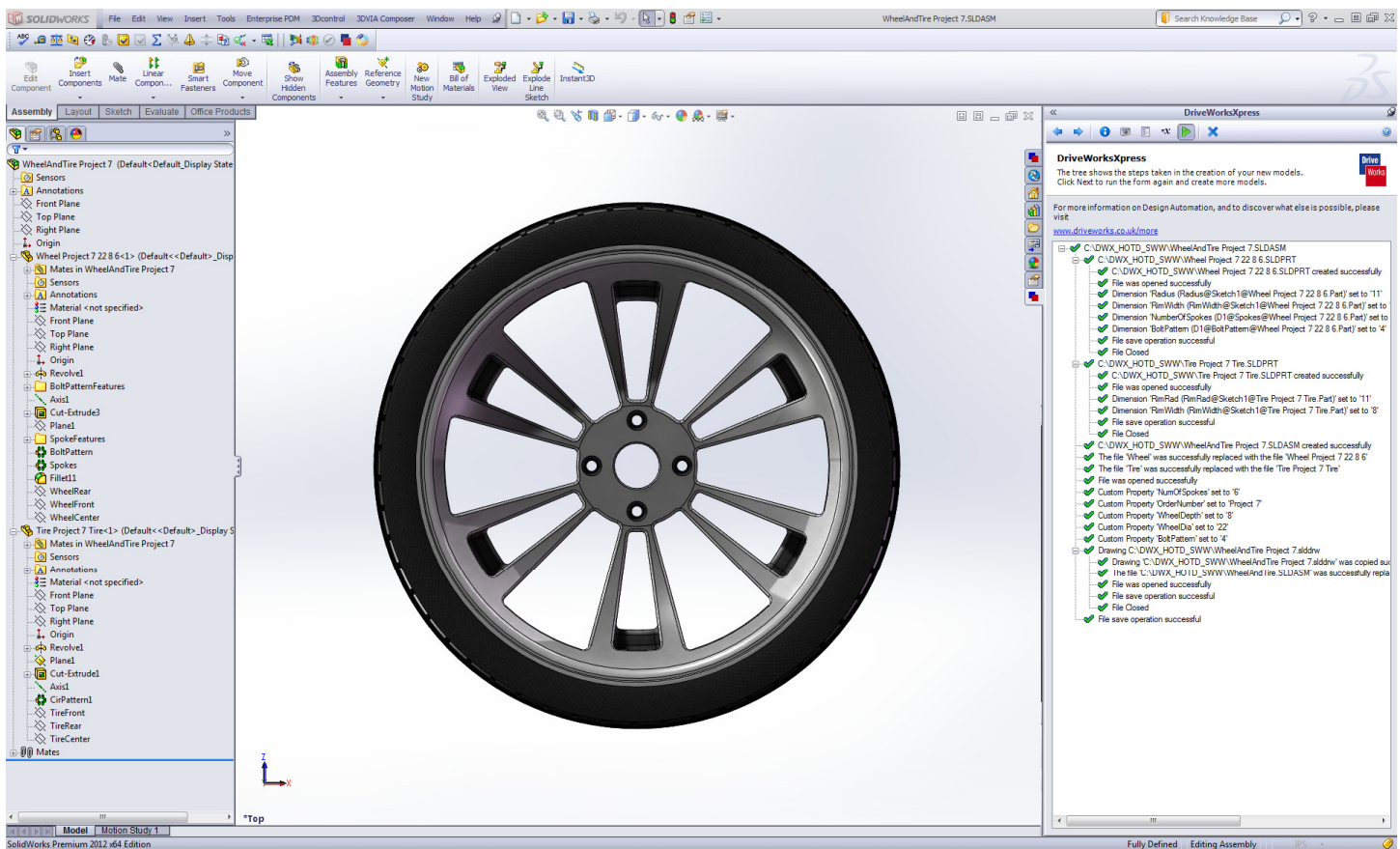
If this configuration exists, the project will open and the log will display -

All files are saved in the **C:\DWX_HOTD_SWW** directory where we opened the master models from. The master models are the parts that all future projects will be run against. Any changes or modification will need to be made to these files for future projects. You can modify any output files without affecting the master model.

Try running different configurations of the Wheel and Tire project.

Tips for Building Better Projects

1. Design intent is 9/10ths of what makes driveworks project successful
2. Minimize in-context relationships
3. If in-context relationships are required, use non-time dependant features
4. Name features that will be automatically driven
5. Name automated dimensions as they are created
6. When using patterns, make relationships (dimensions / constraints) to the first instance, not the seeds.
7. Gather all variables before designing
8. Design for minimum and maximum conditions
9. Use fully constrained Sketches
10. Use fully mated assemblies
11. Use the help file, user forums, and sample projects - driveworks.co.uk

More to Try - Modify this project to make it more complete

1. Drive the title block properties
    a. add a description based on rim size
    b. DWG No. with order number
    c. Drawn by with your name
2. Add a feature that can be suppressed if required
    a. Valve stem hole with a checkbox
    b. Change the tread pattern based on a spin box
    c. Change the Rim material with a drop down
3. Modify the File Name Rules to have a more descriptive output
4. Change the form to allow larger or smaller values and see what can happen