9 Relationships.

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9.1 Introduction.

In this section you will retrieve an existing part. You will define a relationship both in its base feature and between the cut feature and the slot feature.

9.2 Pre-requisites.

You must have successfully completed:

- Pro/ENGINEER: 4 The starter part.
- 6 To create an assembly.
- 7 To create additional features.

9.3 To start your session.

Start Pro/ENGINEER in the usual way.

Ensure your working directory is set to project_1.

Open the STARTPART.prt file.

9.4 How to display dimension values and dimension names.

Every dimension in every feature that you have created has an explicit value. These values control the geometry of the model.

To view a feature's dimensions, simply begin the feature Modify process:

Click on geometry in the display to select the feature (edges change to red). Click the RMB and click Modify.

The display reveals all of the dimensions that define that particular feature.

Repeat the process for another feature, the cut for example. Feel free to spin and re-fit the model. Use Redraw and start again if the display gets too full of dimensions. See figure below.
Each of these dimensions also has a name, a symbol, as do variables in mathematical equations.

To reveal these names use this pull-down menu utility:

**Info > Switch Dimensions**

Observe the dimension values in the display. These names, or symbols if you like, are the mathematical variable names that Pro/ENGINEER uses when setting relationships.

Redraw the display. Examine other feature dimensions, switching between values and names.

**9.5 Creating a relationship in the protrusion.**

You are going to set a relationship that makes the height of the rectangular groove, shown below, 1/3 of its width.

Reveal the dimension names of the STARTPART.prt extruded protrusion feature. Your actual suffixes will probably be different.

In the Menu Manager:

**PART Relations**

Before you enter the relationship equation consider how the slot geometry is to be driven. The left hand side of the equation defines the **dependant** dimension. That is to say, the dimension whose value is defined, or driven, by the result of any combination of other dimensions, mathematical functions and constants.

For example, do you want the slot width to be driven by the height, or the height driven by the width? It can only be one way!

You will drive the height with the width, for this example.

From your display make a note of the symbol name for:

- `rectangle_height`
- `rectangle_width`

In the **Relations** window enter **your relationship equation**, which is equivalent to:

```
rectangle_height = rectangle_width/3
```
For my example, as on page 4, it would be:

Click on **OK**.

From the Menu Manager, regenerate the model:

**PART**

**Regenerate**

Test the relationship by modifying the slot width to 80mm. See section 8.5. Also, modify the height as well. Read any resulting error messages!

9.6 **Creating a relationship between the cut and the slot.**

Reveal the feature dimension names for the cut and the slot.

You will create a relationship to ensure that the rectangular slot is always central to the cut.

For the dimension names assigned on page 5, the following equation should be entered:

\[ d33 = (d11-d31)/2 \]
Define what your model specific dimension names are and then enter the appropriate relation equation.

![Relations dialog box]

Click on OK.

Menu Manager:

**PART**

*Regenerate*

Test the relationship by modifying the cut width to 100mm. See section 8.5 if in doubt.

9.7 **Creating a relationship between the LUG and BOLT in ASSM1.**

The above procedures can be used to set a relationship across parts in an assembly. This is a very powerful facility of parametric associative modelling.

9.7.1 **Dimension names in part features.**

You may have noticed that all of the dimension names start with a lower case d. e.g. d5, d26. This d indicates that the name or symbol represents a dimension in a solid part model. Every dimension for every feature, and every subsequent feature, in this part will have its own unique suffix number to the d.

9.7.2 **Dimension names in sections (sketcher mode).**

Click the MMB a few times to ensure that the Modify command is done. Redraw and select the first feature again. Click the RMB and then click Redefine. Select Section in the PROTRUSION:Extrude window and click the Define button. Click on Sketch.

You are now in sketcher mode, looking at your original sketch of the base feature section. Reveal the dimension names:

**Info > Switch Dimensions**
Observe that the dimension name prefix is \textit{sd}. This indicates that the name or symbol represents a \textit{sketch} or \textit{section dimension} in a section, in skinner mode. Again, every dimension for every entity sketched in that particular section will have its own unique suffix number to the \textit{sd}.

\textbf{Note:} You would only usually define relationships in the \textit{section} if you intend to save the section as an independent file (*.sec) for use in other features, and features in other parts.

Click the \textbf{tick} and then the \textbf{OK} button in the \textit{PROTRUSION:Extrude} window to return to the model.

9.7.3 Dimension names in assemblies.

Assemblies are collections of parts and constraint dimensions (which define how the parts are positioned). Different parts may have instances of identical dimension names, so they have to be differentiated when used in an assembly. This is achieved with the use of a colon suffix, which explicitly identifies the part or constraint.

Retrieve the \texttt{ASSM1.asm} assembly.

Make the following selections in the Menu Manager:

\begin{itemize}
  \item ASSEMBLY
  \item Modify
  \item ASSEM MOD
  \item Mod Dim
  \item MODIFY
  \item Dimension
\end{itemize}

Now pick each part in the assembly by clicking on the geometry in the display.

Examine the values, switch to show the names.

Observe that all of the dimensions are of the format: \texttt{d23:4}

Close the assembly window.

Activate the \texttt{STARTPART}.

9.7.4 Assembly relation task.

Retrieve or activate the \texttt{ASSM1.asm} assembly. Define a relationship to ensure that the BOLT shank diameter is the same as the hole in the LUG. Make the BOLT drive the hole diameter in the LUG.

Use procedures in section 9.7.3 and from the Menu Manager: \texttt{ASSEMBLY > Relations}

Close all of your part and assembly windows.
9.8 Introduction.

In this section you will create a solid model of a rectangular block.

You will define what are called relations between its dimensions. These relations control its geometry. For this example the width dimension will control both the height and length dimensions.

The height will be half the width. The length will be three times the width.

9.9 Pre-requisites.

You must have successfully completed:

- Pro/ENGINEER: 4 The starter part.
- 5 Additional parts.

9.10 To create the 'cuboid' part.

Start Pro/ENGINEER in the usual way.

Create and set a new working directory called relations.

Create a new part file called cuboid, and create the first feature, an extruded protrusion:

Insert > Protrusion... > Extrude

Make the attributes One side, and select a plane of your choice to sketch on.

Okay the extrusion direction and select the Default sketch orientation.

Close the references box.
Follow these steps to sketch a rectangle, making use of symmetry about the vertical reference:

Create a **vertical centerline**, snapping to the vertical reference as shown here.

This centerline will prompt the sketching intent manager to look for possible symmetry when sketching.

Sketch the rectangle:

Do not worry about the actual dimension values.

Look for the symmetry constraint symbols as you position the rectangle.
Continue with the current section and make the extrusion **blind** and accept the default value.

Complete the feature and spin the model to check your geometry.

The next step for this example is to Modify the Dimension Symbols to give them a meaningful value.

**9.11 The dimension symbols.**

Every dimension in every feature that you have created has an explicit value. These values control the geometry of the model.

To view a feature's dimensions, simply begin the feature **Modify** process:

Click on geometry in the display to select the feature (edges change to red). Click the RMB and click **Modify**.

The display reveals all of the dimensions that define that particular feature.

Each of these dimensions also has a name, a **symbol**, as do variables in mathematical equations.

To reveal these names use this pull-down menu utility:

**Info > Switch Dimensions**

Observe the dimension values in the display. These names, or symbols if you like, are the mathematical variable names that Pro/ENGINEER uses when setting relationships.

Redraw the display.

**9.11.1 Dimension symbols in part features.**

You may have noticed that all of the dimension names start with a lower case **d**. e.g. d5, d26. This **d** indicates that the name or symbol represents a **dimension** in a solid **part** model. Every dimension for every feature, and every subsequent feature, in this part will have its own unique suffix number to the **d**.

**9.11.2 Dimension symbols in sections (sketcher mode).**

Click the MMB a few times to ensure that the Modify command is done. Redraw and select the protrusion feature. Click the RMB and then click **Redefine**. Select **Section** in the **PROTRUSION:Extrude** window and click the **Define** button. Click on **Sketch**.

You are now in sketcher mode, looking at your original sketch of the base feature section. Reveal the dimension names:

**Info > Switch Dimensions**
Observe that the dimension name prefix is **sd**. This indicates that the name or symbol represents a **sketch** or **section dimension** in a section, in skinner mode. Again, every dimension for every entity sketched in that particular section will have its own unique suffix number to the **sd**.

**Note:** You would only usually define relationships in the **section** if you intend to save the section as an independent file (***.sec**) for use in other features, and features in other parts.

Click the **tick** and then the **OK** button in the **PROTRUSION:Extrude** window to return to the model…

### 9.12 Changing the dimension symbol names.

You can change dimension symbols to something more meaningful than the Pro/E automatically assigned symbols. Names such as d3 or sd7 are not very useful.

From the Menu Manager on the side:

**PART**
**Modify**
**DimCosmetics**
**DIM COSMETIC**
**Symbol**

Then pick the cuboid protrusion feature. Observe the display of the defining dimensions. Click on the dimension symbol you wish to change, and then enter its new value where prompted in the message window. Change all three dimensions for the cuboid, as shown below:
9.13 Creating Relations.

Now you will create the dimension relations.

From the Menu Manager:

PART
Relations

This opens up the Relations window, as shown in part below.

Click on the **Insert dimension symbol from screen** button and then click on the cuboid in the main display screen. This should display the dimensions of the part. If the dimension values are displayed rather than the symbol names, use the **Toggle between dimension values & names button** to display their names.

Now you can clearly see what is what.

Remember that: The **height** will be half the **width**.
The **length** will be three times the **width**.

Enter the relations (equations) as shown here.

Then click on **OK**.

The **cuboid** is now setup with relations between its dimensions.

Modify the width in the normal way and observe how the model is automatically driven by its relations.

This is all part of the **parametric** capability of solid modeling software.

Save your work and work on the following example.
9.14 Relations example.

Still working in the Relations directory, create a new part called **hex_bolt**.

Create a solid model of a blank (no thread) hexagon headed bolt.

Use dimension **relations** to ensure that:

- The across flats dimension of the hex head is 1.75 X the shank diameter.
- The height of the hex head is the same as the shank diameter.
- The length of the shank is 6 X the shank diameter.

Tip:

Create the hex head first, ensuring that there is an **across_flat** dimension:

The whole hexagon section should be driven by the across flat dimension.
9.15 **Tutorial for creating relations between parts in the same assembly.**

Create and set a new working directory called `Assembly_relations`.

Create two new parts as shown below called `HOUSING` and `STEP_SHAFT`.

Take care to use appropriate sketching references, and make sure that shaft and hole diameters are displayed as diameters.
In each part file, modify the dimension cosmetics for the symbols for the diameters and the housing width, as shown below, (as you did in section 9.8).

Menu Manager
PART
Modify
DimCosmetics
Symbol

Now create a new assembly file called drive_assembly and assemble the housing and shaft as shown below:
As the design engineer you are informed that:

1) The mounting hole positions on the flanges remain constant relative to the end faces of the flanges.

2) The shaft **boss_dia** must be 1.5 times greater than the **shaft_dia** value.

3) The **shaft_dia** of the drive shaft must mate with another housing, with an internal diameter ranging between 10 and 65mm.

4) Housing height, width and bore centerline height must all retain their original specified values.

5) The **housing_width** must be 10mm greater than the **shaft_dia**.

As this assembly stands, if you wish to change the shaft diameter, you have to modify features in both parts separately. Parametric solid modeling software is capable of doing this for you. You can set up your design model so that it is driven by one variable dimension, let’s describe it as the nominal shaft diameter value. You should then be able to generate a model for any nominal shaft size, ranging in this case from 10 to 65mm diameter.

Now you must modify your part and assembly models to achieve this.

9.15.1 Preparing the individual parts.

In the **step_shaft** part, create this relation:  

```
boss_dia=shaft_dia*1.5
```

This ensures that the boss diameter is driven by the shaft diameter, and has a value 1.5 times larger.

In the **housing** part, create this relation:  

```
housing_width=bore_dia+10
```

This ensures that the width of the housing is always 10mm larger than the bore diameter.

To ensure that the other features remain constant, check your sketching dimension references, feature dimensions should look like those on page 13.

9.15.2 Preparing the assembly.

In the **drive_assembly** file, bring up the Relations window from the Menu Manager. Click on the **Local Parameters** banner to display the **Local Parameters**.

A **Local Parameter** is a variable which you can define and then refer to within Relation expressions. You can create as many as you like. They can be text strings, real or integer numbers.

To add a new parameter in the assembly, click on the **+** button.

Enter its **Name**, select its **Type** and enter its **Value** as shown on page 16.

The value of this new parameter called **NOMINAL_DIA** will be used in your relation expressions to drive the shaft and hole diameters. Mathematically speaking, **NOMINAL_DIA** is a variable, and can be used in Relation expressions.
Enter your relations as indicated below.

You must comment your relations, so that other users can follow what is being done. Create a comment simply by preceding the text with: /*

Select dimension symbol from screen.

Create comments for each relation expression.

Test the relations by changing the value of the NOMINAL_DIA parameter, clicking OK and then re-generating the assembly model.

The model should vary as shown here: