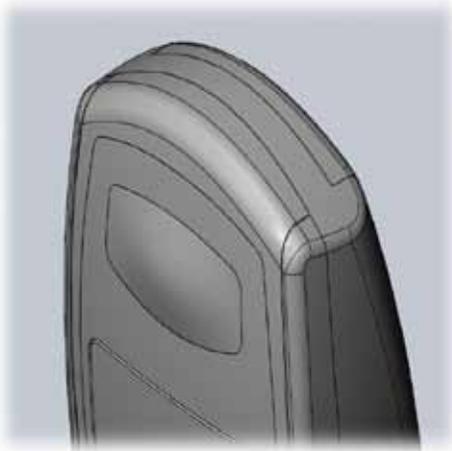# Feature Configurations in SolidWorks
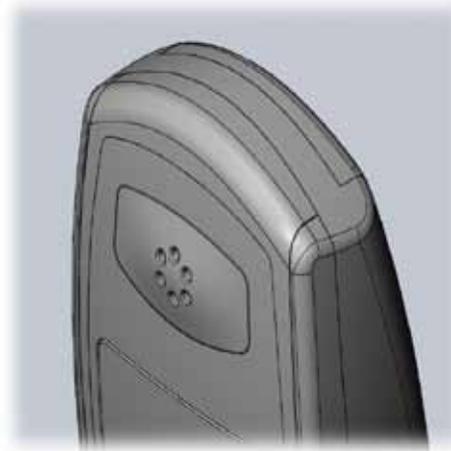
SolidWorks contains a functionality called "Configurations". It allows you to easily switch between different designs based on dimensions, materials or features. This tutorial covers how to setup feature based configurations so you can easily look and switch between different design alternates.

Default Configuration
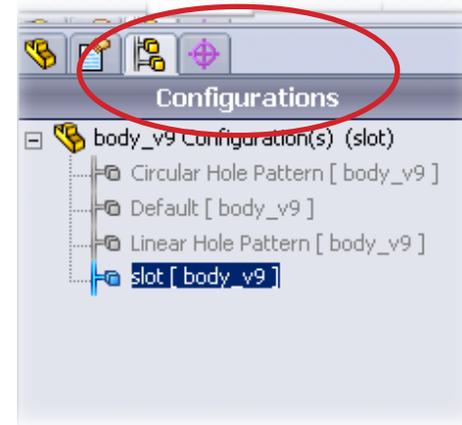

Linear Configuration

Circular Configuration

Slot Configuration

These four images show examples of a feature configuration on a speaker detail. The different configurations can easily be selected through the "Configuration" tab.
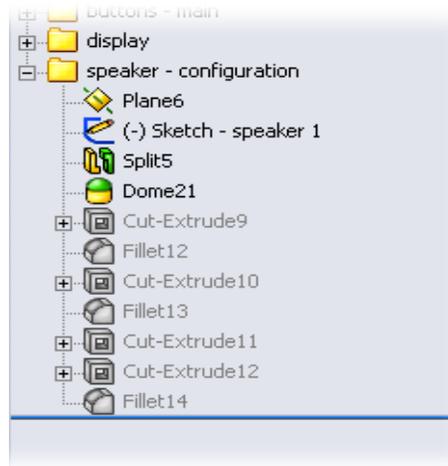
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Feature Configurations in SolidWorks

Step 1

Model each design alternative and then suppress it. The drawing tree below shows the default configuration and three alternate designs modeled but suppress.

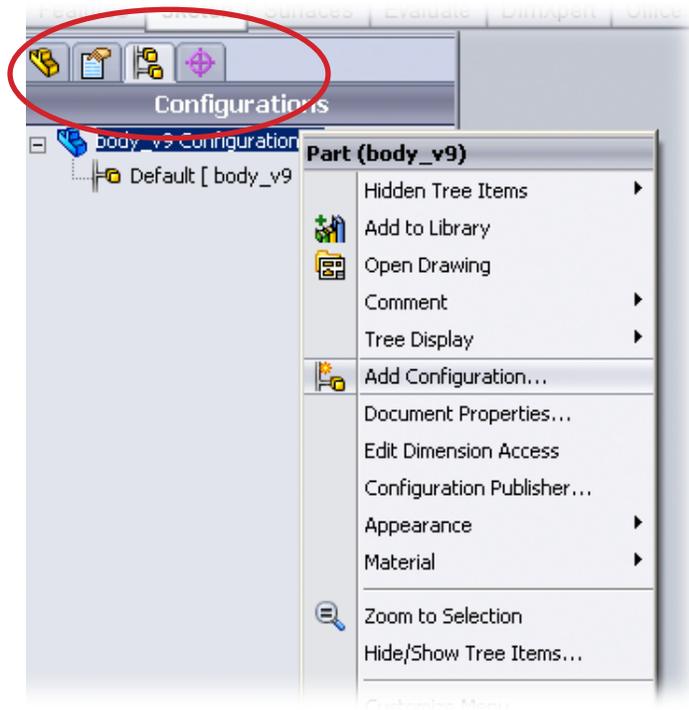*Base design with all design alternatives suppressed*

Drawing tree showing alternate designs

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Feature Configurations in SolidWorks

Step 2

Switch to the Configuration tab and right mouse click on the top item in the feature tree.
Select the Add Configuration option and type in a name.

*Right mouse click on the top item
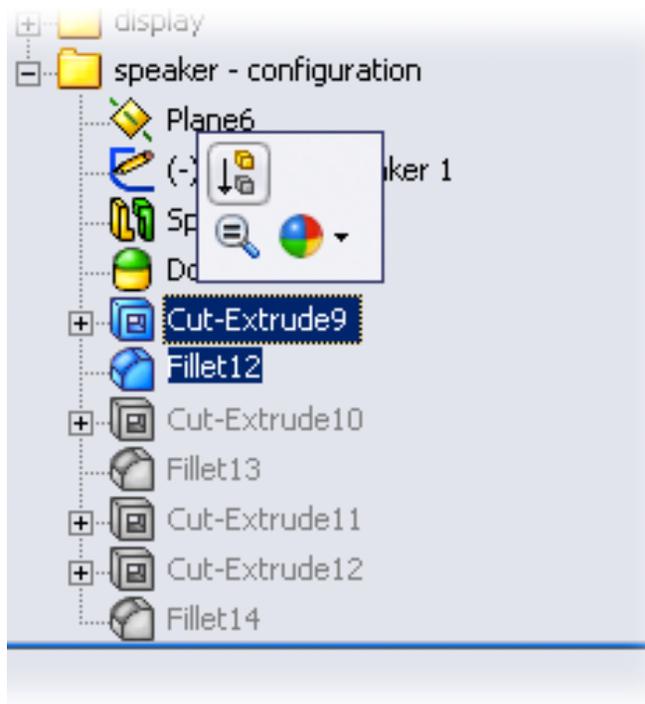to access the pop up menu*

Name the Configuration

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Feature Configurations in SolidWorks

Step 3

Switch back to the Feature tab and unsuppress the items for configuration.
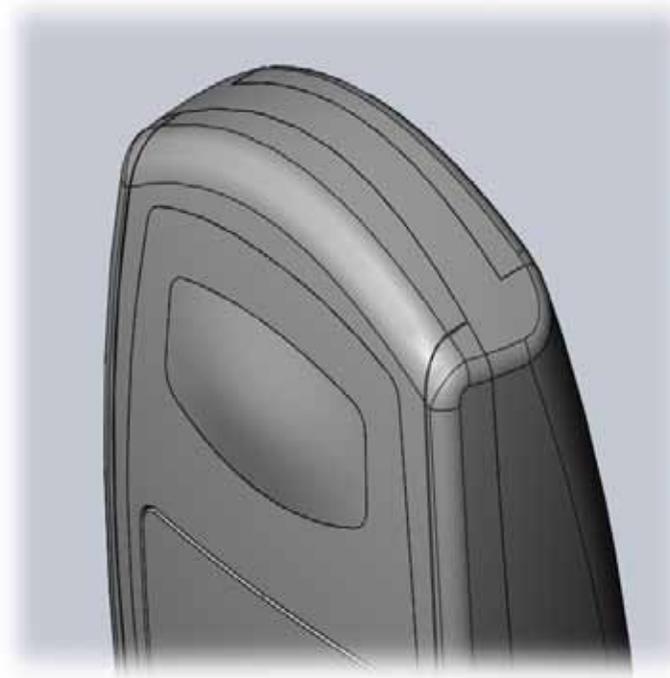

*Right mouse click on the feature
and unsuppress items*

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Feature Configurations in SolidWorks

Step 4

To add another configuration, switch back to the "Default" configuration first by double clicking on it under the Configuration tab.

Double click on the configuration to select it

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Feature Configurations in SolidWorks

Step 5

Right mouse click on the top item in the feature tree and select the Add Configuration option and type in a name. Repeat this process to add additional configurations.

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