ABOUT SOLIDWORKS CONFIGURATIONS FOR DESIGN

Prof. Cătălin Iancu, *Constantin Brâncuşi University of Târgu*-Jiu, ROMANIA

ABSTRACT: In this paperwork are presented the SOLIDWORKS Configurations features and the steps to be taken in order to use this facility for better design. There are presented the settings that have to be done for using different configurations, which allows creation of multiple variations of a part or assembly model within a single document. Also "Configurations" provide a convenient way to develop and manage families of models with different dimensions, components, or other parameters.

KEY WORDS: SOLIDWORKS, configurations, Configuration Manager, design table.

1. INTRODUCTION

In [1], it has been presented some of SOLIDWORKS advanced design modules, such as SOLIDWORKS SustainabilityXpress, which has the capability of evaluation of environmental impact of a design throughout the life cycle of a product. Thus it can be compared results from different designs to ensure a sustainable solution for the product and the environment.
In the same line of optimizing design enrolls Configurations facility, which allows create multiple variations of a part or assembly model within a single document. So "Configurations" provide the tools for developing and managing families of models with different dimensions, components, or other parameters.
To create a configuration, one can specify a name and properties, and then modify the model to create the design variations wanted.
For **part** documents, configurations allow to create families of parts with different dimensions, features, and properties, including custom properties.
For **assembly** documents, configurations allow to create simplified versions of the design by suppressing components, families of assemblies with different configurations of the components, different parameters for assembly features, different dimensions, or configuration-specific custom properties.
For **drawing** documents, it can be displayed views of the configurations created in part and assembly documents [2].

Configurations can be created using any of the following methods:
- Create configurations manually.
- Use design tables to create and manage configurations in a Microsoft Excel worksheet. Design tables can be displayed in drawings.
- Use the Modify Configurations dialog box to create and modify configurations for commonly configured parameters.
Custom properties created in design tables or the Modify Configurations dialog box are automatically added to the Configuration Specific tab in the Summary Information dialog box.

In figure 1 is presented the Configuration Manager tab, which can be accessed from Feature Manager, and where begins the creation of different configurations.

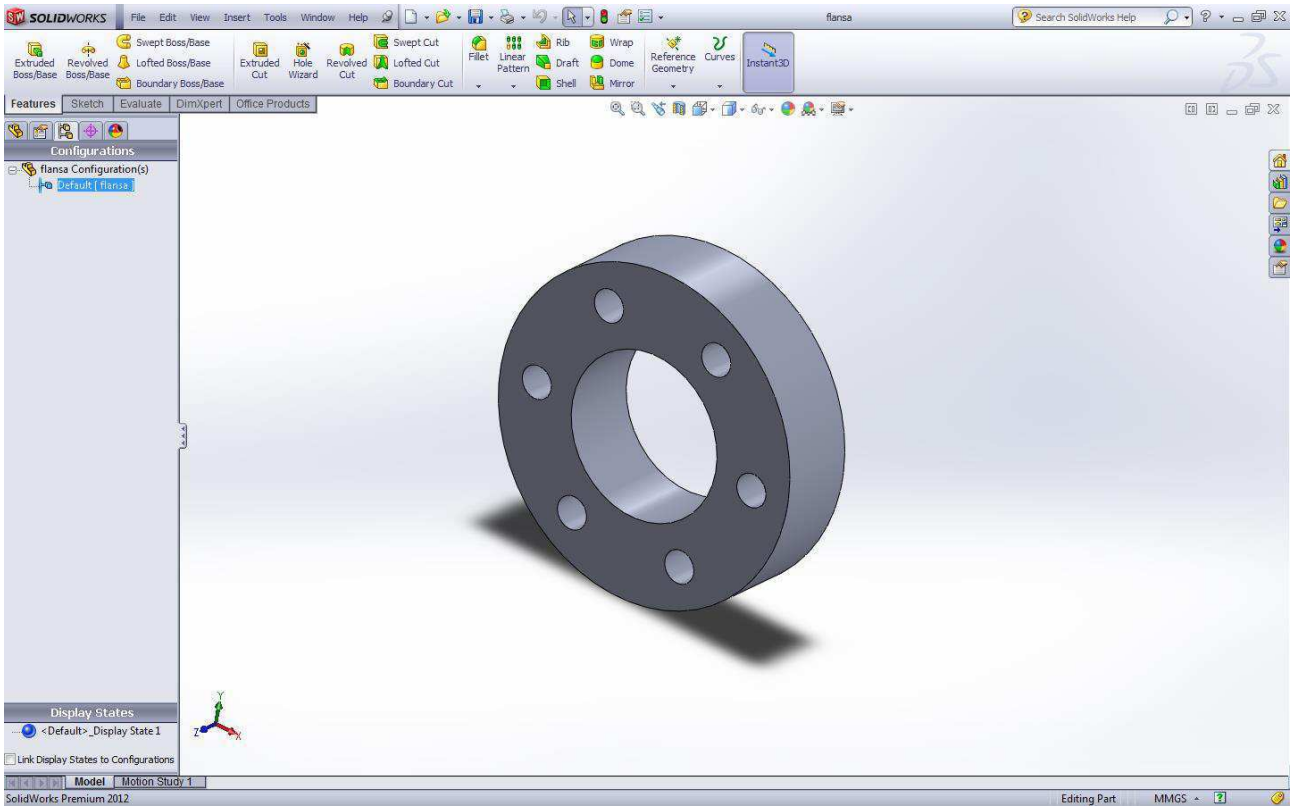

Figure 1. Configuration Manager tab

2. CREATING A CONFIGURATION MANUALLY

To create a configuration manually, you specify its properties and then modify the model to create variations in the new configuration.
To create a configuration manually one must take these steps:
- In either a part or assembly document, click the ConfigurationManager tab at the top of the FeatureManager design tree to change to the ConfigurationManager.
- In the ConfigurationManager, right-click the part or assembly name and select Add Configuration.
- In the Add Configuration PropertyManager, type a Configuration Name and specify properties for the new configuration.
- Click OK ✔.
- Click the FeatureManager design tree tab to return to the FeatureManager design tree.
- Modify the model as needed to create the design variation.
The dialog box is presented in figure 2.

3. CREATING A CONFIGURATION USING A DESIGN TABLE

A design table allows you to build multiple configurations of parts or assemblies by specifying parameters in an embedded Microsoft Excel worksheet.
When you use design tables in the SolidWorks software, it is important to format the tables properly.
To use design tables, you must have Microsoft Excel installed on your computer.
There are several different ways to insert a design table:
A - using SolidWorks software for inserting a design table automatically (figure 3). The steps are:
- In a part or assembly document, click Design Table on the Tools toolbar, or click Insert, Design Table.

- In the PropertyManager, under Source, select Auto-create.

- Set the **Edit Control** settings and **Options**, as desired.

- Click OK ✔.

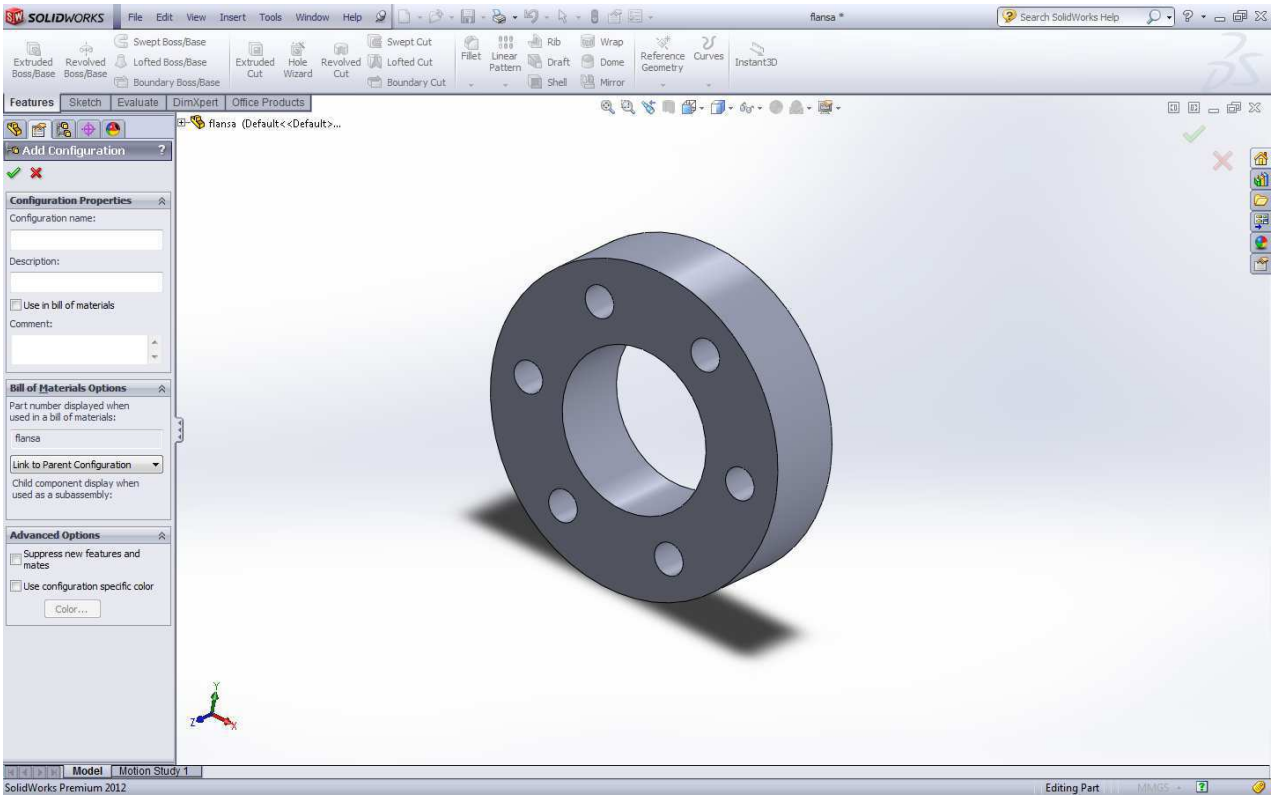

Figure 2. Creating a configuration manually

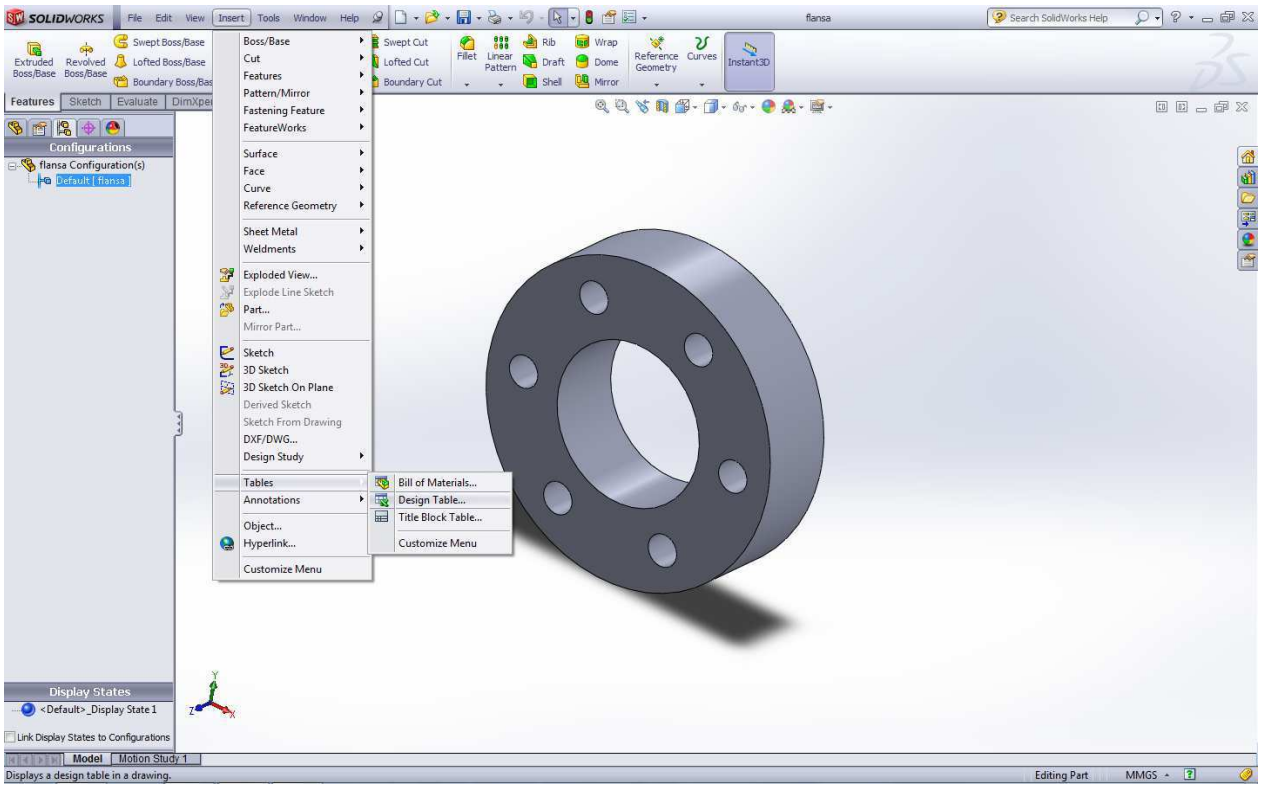

Figure 3. Creating a configuration using a design table

Depending on the settings you selected, a dialog box may appear that asks which dimensions or parameters you want to add.
An embedded worksheet appears in the window, and the SolidWorks toolbars are replaced with Excel toolbars.
Cell A1 identifies the worksheet as Design Table for: <model_name>.
Click anywhere outside of the worksheet (but in the graphics area) to close the design table.

B - inserting a blank design table;
In a part or assembly document, click Design Table on the Tools toolbar, or click Insert, Design Table.
In the PropertyManager, under Source, click Blank (figure 4).
- Set the **Edit Control** settings and **Options**, as desired.

- Click OK ✔.
Depending on the settings you selected, a dialog box may appear that asks which dimensions or parameters you want to add.
An embedded worksheet appears in the window, and the SolidWorks toolbars are replaced with Excel toolbars.
Cell A1 identifies the worksheet as Design Table for: <model_name>. Cell A3 contains the default name for the first new configuration, First Instance.
In row 2, type the parameters that you want to control. Leave cell A2 blank. Notice that cell B2 is active.
You can also enter parameters by double-clicking the feature or dimension in the graphics area or in the FeatureManager design tree. When you double-click a feature or dimension, its associated value appears in the Default row.
In column A (cells A3, A4, and so on), type the names of the configurations that you want to create. The names can include numerics, but must not include the forward slash (/) or at (@) characters.
When you finish adding information to the worksheet, click outside the table to close it.
A message lists the configurations that were created.

- Click OK ✔.
The design table is inserted in the model, and Design Table appears in the FeatureManager design tree.
You can change the configuration name in cell A3 (Default) if you want.
Type the parameter values in the worksheet cells.

C - insert an external Microsoft Excel file as a design table.
In a part or assembly document, click Design Table on the Tools toolbar, or click Insert, Design Table.
The Design Table PropertyManager appears.
Under Source click From file, then click Browse to locate the Excel file (figure 4).
To link the design table to the model, select the Link to file check box. A linked design table reads all of its information from an external Excel file.
Note that if you update a linked design table in Microsoft Excel, then open the SolidWorks model, you can choose to update either:
- the model with the design table values, or
- the design table with the model values
You can set the update options in Tools, Options, System Options, External References. Set Update out-of-date linked design tables to Model, Excel file, or Prompt.
- Set the **Edit Control** settings and **Options**, as desired.

- Click OK ✔.
An embedded worksheet appears in the window, and the SolidWorks toolbars are replaced with Excel toolbars. In figure 5 is presented such a design table.

4. USING MODIFY CONFIGURATION DIALOG

The Modify Configurations dialog box (figure 6) provides a table where you can create and modify configurations for commonly configured parameters in parts and assemblies. You can access the Modify Configurations dialog box in several ways.
To access the Modify Configurations dialog box do one of the following:

1. Right-click an item and click Configure Dimension, Configure Feature, Configure Component, or Configure Material.
2. Access a previously saved table: in the ConfigurationManager, expand Tables, right-click a saved table, and click Show Table.

Note that the Modify Configurations dialog box lists the configurations of the model in one column and the configured parameters of the selected item in other columns.
3. Add configurations and make changes as needed.

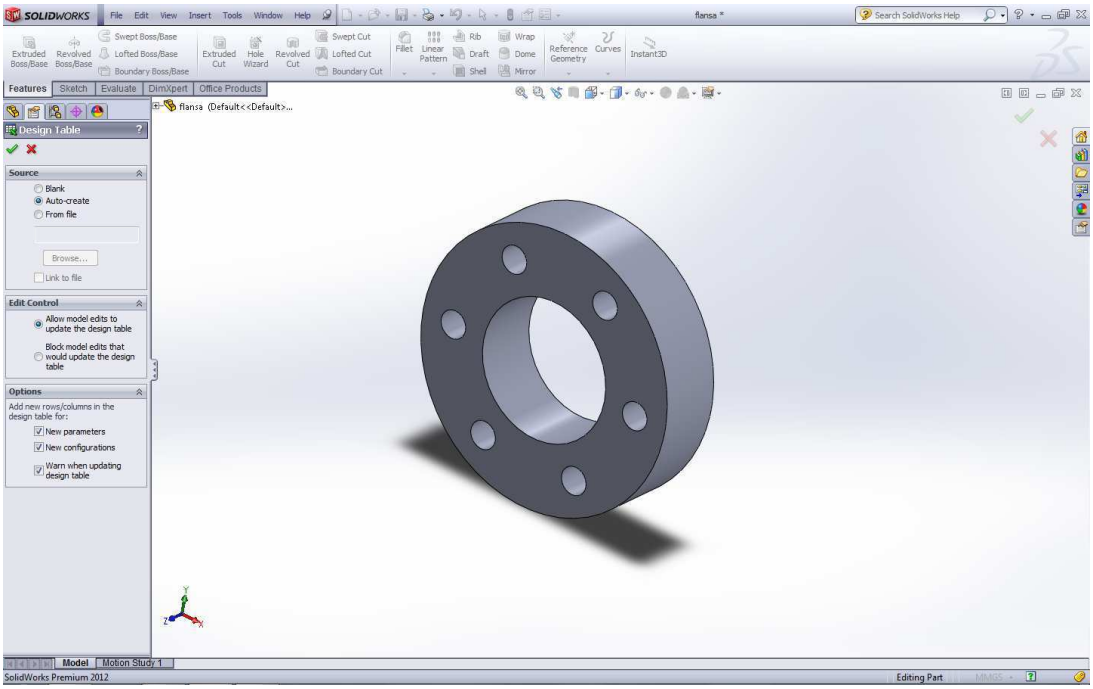

Figure 4. Inserting a blank design table or from external file

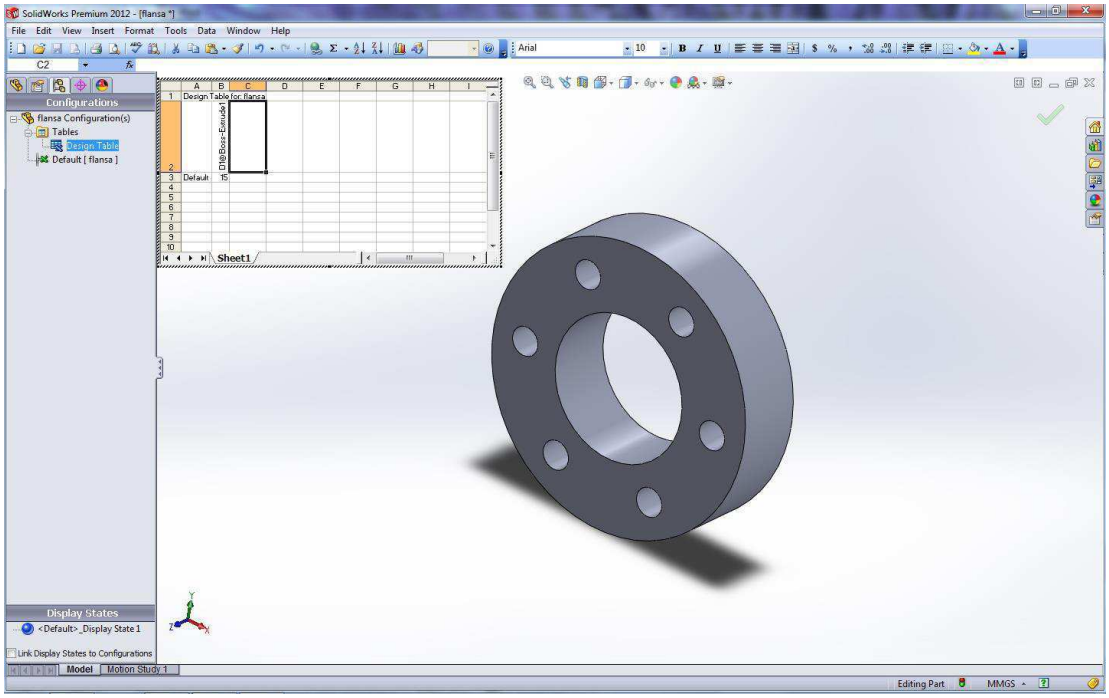

Figure 5. Editable design table (Excel sheet)

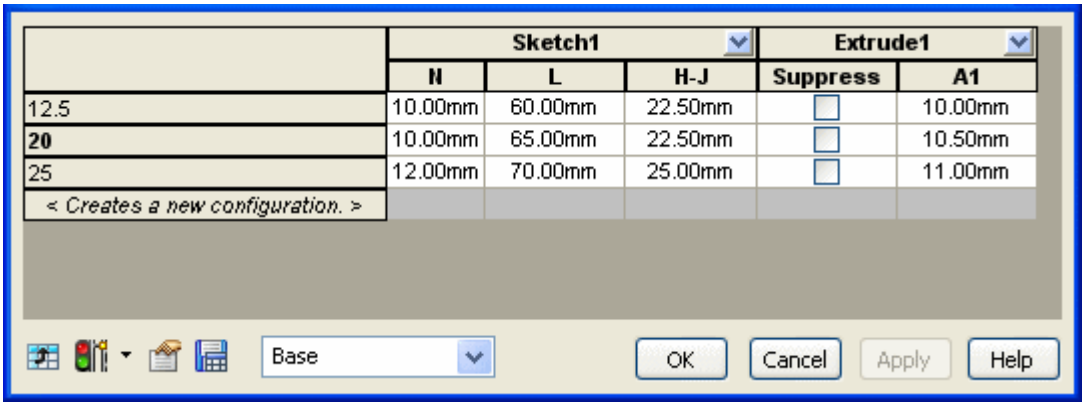

Figure 6. Modify Configuration dialog box

You can perform the following operations in the Modify Configurations dialog box:
- add, delete, and rename configurations;
- create, edit, and delete configuration-specific custom properties;
- rename features and dimensions;
- reconfigure parameters, which apply the active configuration's value to all configurations.

In parts, you can configure:
- dimensions of features and sketches;
- suppression states of features and sketches;
- material;
- custom properties.

In assemblies, you can configure:
- which configurations of components to use;
- dimensions of assembly features and mates;
- suppression states of components, assembly features, and mates;
- custom properties.

Also you can manipulate the table to:
- add and remove feature parameters;
- change which configuration is active;
- rearrange columns.

5. CONCLUSIONS

As it was presented, SOLIDWORKS Configurations facility is intended to optimizing design. Using this facility one can create multiple variations of a part or assembly model within a single document. So Configurations facility permits to develop and manage families of models with different dimensions, components, or other parameters.

Parameters for a specific configuration can be:
- Color parameter;
- Comment or User notes;
- Component part number;
- Cosmetic threads;
- Custom properties;
- Dimensions and Tolerances;
- Display states;
- End conditions for features;
- Equations associated to features;
- External sketch relations;
- Global variables;
- Lighting and cameras for visualization;
- Mass properties;
- Materials;
- Scale features;
- Sketch planes;
- Sketch relations;
- Split parts;
- Suppression state of components;
- Suppression state of features.

It is obvious that's a wide list of parameters, so the possibilities for optimizing design are very broad, so one can save time and money for a quite consuming activity – CAD.

REFERENCES

[1].Iancu C., About SolidWorks sustainability module capability, Fiability & Durability Revue, ISSN 1844-640X, pp.229-235,1/ 2014
[2]. SolidWorks Advanced Modules, Dassault Systèmes SolidWorks Corporation, Waltham, MA, USA, 2012.
[3]. Lombard, M., Solid Works Bible, Wiley, USA, ISBN 978-1-118-50840-4, 2013
[4] http://www.aboutsolidworks.com/ solidworks_tutorials.htm