Design Configuration with SolidWorks Design Table - RC Wheels

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Figure 1: Walking robot CAD, obtained from [1]

With the advancement of computing power, the popularity of Computer-Aided Design (CAD) has increased. Today, CAD is used in many industries such as racing, robotics and wind turbine designs. Compared to traditional hand-drawn diagrams, CAD brings the ability to draw and visualise complex designs quickly and efficiently.

In the biologically inspired robotics field, CAD designs many robots such as walking robots [1] (Figure 1), robotic fish [2] and underwater robots [3]. Often, there are different configurations of the same design, such as the same robotic gripper structure but with different types of servos. Instead of needing to save each configuration as a separate file, it is easier to have one main file which contains several design configurations. This can be done with the use of Design Table in SolidWorks.

In this article, a simple RC wheel will be designed using equations and global variables to allow easy change of dimensions. After that, the design table will be introduced with some basic functionality explained. The aim at the end is to have different wheels configuration by just clicking and typing a few numbers.

1 Wheel Design

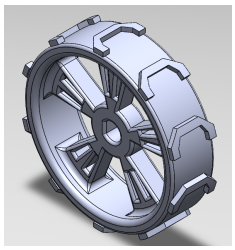

Figure 2: Simple Wheel Design

A simple wheel shown in Figure 2 will be designed using equations and global variables. Equations allow the design to be configured such that any change in the global variables will reflect the whole design without needing the user to manually change it at every feature/sketch of the part. By default, the equations tab is 'hidden'. To select it, go to 'FeatureManager Design Tree', right-click on the part, navigate to 'Hidden Tree Items', and select Equations. A folder with the symbol $\sum$ will appear along with 'Equations' text. From there, right-click 'Manage Equations'. This is where the global variable and equation are located at. A snapshot of the equation used to design the wheel is shown in Figure 3.

Global Variables		
	"Rim_Diameter"	= 30
	"Tyre_Height"	= 5
	"Center_Cap"	= 10
	"No_of_Spoke"	= 5
	"Tyre_Thickness"	= 10
	"Shaft_Diamter"	= 4
	"Shaft_D_Height"	= 3.6
	"Tread_Thickness"	= 1.5
	"Tread_Height"	= 1.5
	"Tread_Depth"	= 1
	"No_of_Tread"	= 10
	Add global variable	
Features		
Equations		
	"D4@Spoke_Sketch"	= "Center_Cap" + 5
	"D5@Spoke_Sketch"	= "Rim_Diameter" - 5

Figure 3: Equations for simple wheel design

Defining a new variable is as simple as typing the variable name in the first column and the value in the second column. Equations are automatically amended to the list when a feature such as 'Extruded Boss' and 'Extruded Cut' is added to the part. During a sketch, when the user sketches an entity or defines that entity/relation with 'Smart Dimension' they are automatically added to the equation tabs. With this, the user can create a complicated part instead of needing to change each entity/dimension individually, all can be done using the equation tab.

It is common and good practice to always define crucial parameters using the equation tabs especially when a Design Table is to be used as it will streamline the process effectively.

2 Design Table

	A	B	C	D	E
1	Design Table for: Wheel_DT				
2		$VALUE@Rim_Diameter@EQUATIONS	$VALUE@Tread_Height@EQUATIONS	$VALUE@Tread_Depth@EQUATIONS	$VALUE@No_of_Tread@EQUATIONS
3	Small Wheel, Small Tread	=30	=1.5	=1	=10
4	Small Wheel, Big Tread	=30	=2.5	=2	=10
5	Big Wheel, Small Tread	=60	=1.5	=1	=20
6	Big Wheel, Big Tread	=60	=2.5	=2	=20

Figure 4: Design Table

When there are several configurations of a particular part, it is easier to use a design table to create these configurations. To add a design table, go to 'Insert', 'Tables', and 'Excel Design Tables'. Design tables in SolidWorks are Excel spreadsheet and it has all the features of a conventional Excel spreadsheet. This allows complex equations to be evaluated easily using Excel. A screenshot of a typical design table is shown in Figure 4.

There are 3 methods to create a design table, blank, autocreate and manually link an existing spreadsheet. With all 3 methods, it is crucial to have one of the cells (A2 in this case) with the Name Box 'Family' as this tells SolidWorks that any cell right is the syntax of the parameters to be varied while anything below it is the different configuration names and values.

There is numerous naming convention that has to be followed for SolidWorks to recognise it and they can be found in SolidWokrs help webpage[1]. Alternatively, by clicking on the feature/entity the user desire, it can be added to the list automatically.

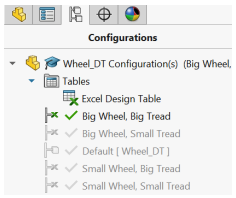

Figure 5: Design Configurations

Once it is done, save and close the design table. On the 'FeatureManager Design Tree' tab, select 'Configuration Manager'. By expanding the 'Tables' tab, an 'Excel Design Table' is seen in Figure 5. A green 'X' indicates this configuration/table is from excel. To view each configuration, simply double-click on the desired configuration. When the part is added to assemble, an option will be given to the user to select which configuration is to be used.

3 Acknowlegement

The design table and equation were learnt from online resources [4], [5]. Files can be obtained at GitHub[2]

[1] D. Grzelczyk and J. Awrejcewicz, "On the controlling of multi-legged walking robots on stable and unstable ground," in 2020-03, ISBN: 978-1-83880-229-5. DOI: 10.5772/intechopen.90208.

[2] H. El Daou, T. Salumäe, G. Toming, and M. Kruusmaa, "A bio-inspired compliant robotic fish: Design and experiments," in *2012 IEEE International Conference on Robotics and Automation*, 2012, pp. 5340–5345. DOI: 10.1109/ICRA.2012.6225321.

[3] G. Bianchi, S. Cinquemani, and F. Resta, "Bio-inspired design of an underwater robot exploiting fin undulation propulsion," *Applied Sciences*, vol. 11, no. 6, p. 2556, 2021, ISSN: 2076-3417. [Online]. Available: https://www.mdpi.com/2076-3417/11/6/2556.

[4] I. CADimensions. "Lunch & learn - design tables and configurations." (2013-03), [Online]. Available: https://www.youtube.com/watch?v=NRFYEgLz8KO&t=16s.

[5] P. Creative. "Rack and pinion design 2 - rack gear design in solidworks." (2022-08), [Online]. Available: https://www.youtube.com/watch?v=fys_u5jwNwQ.

[1] https://help.solidworks.com/2022/english/SolidWorks/sldworks/c_Design_Table_Configurations.htm?

[2] https://github.com/steveLim0928/Design_Table_Tutorial