SW Tips/Tricks

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Links, Equations, and Design Tables

This project demonstrates a simple tolerance analysis of two mating plates in an assembly. The first "pin plate" has two dowel pins pressed into each of the two holes, mating to a second "hole plate" with two holes. This project will demonstrate the classic worst case tolerance analysis where the extremes of the tolerance range for each dimension studied will be checked against each other in the context of the mating parts of the assembly. Tolerance analysis is not a trivial subject, therefore this article will simplify the task to demonstrate techniques utilizing various SolidWorks tools, without becoming engrossed in all of the details of the analysis. Also, try to think of some the other ways to solve these same problems utilizing other tools that SolidWorks provides. Remember that there are always multiple solutions to each part of the task. Some of these alternative solutions will be discussed.

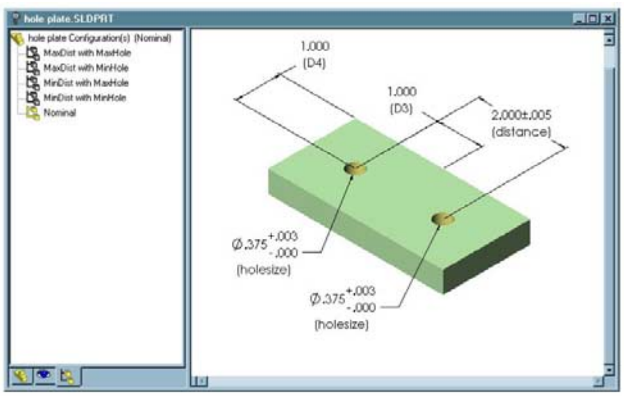
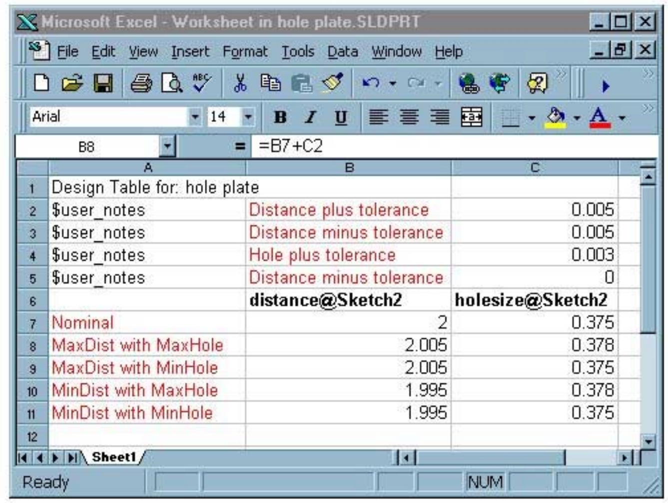


Figure 1



	A	B	C
1	Design Table for: hole plate		
2	$user_notes	Distance plus tolerance	0.005
3	$user_notes	Distance minus tolerance	0.005
4	$user_notes	Hole plus tolerance	0.003
5	$user_notes	Distance minus tolerance	0
6		distance@Sketch2	holesize@Sketch2
7	Nominal	2	0.375
8	MaxDist with MaxHole	2.005	0.378
9	MaxDist with MinHole	2.005	0.375
10	MinDist with MaxHole	1.995	0.378
11	MinDist with MinHole	1.995	0.375
12			

Figure 2

The goal is to determine if the two mating parts will fit together (when the production parts are manufactured anywhere in the tolerance range specified for each dimension). The first step will be to model the individual parts, with special attention on the way the dimensions and tolerances are applied to the model. Then the part models will be placed in an assembly and each worst case combination will be studied. For each possible combination the interference and clearance between the pins and the holes will be checked.

There are three parts and an assembly that need to be created (see Figures 1, 3, and 5). Create the "hole plate", the "pin plate", the "pin", and the assembly of these parts. The same pin is used in both holes in the "pin plate". The outside diameter tolerance of these pins will not be considered for this project. The plates are modeled as simple rectangular plates 4in x 2in x 0.5in thick. The "hole plate" is the part that contains the clearance holes for the "pins". Before dimensioning the holes in a sketch, think ahead to the assembly. Specifically consider the mates that will be applied to

continued on page 2

INSIDE THIS ISSUE

…continued from page 1

both parts. The first component in the assembly, the "pin plate", can either be fixed at the origin of the assembly, or the planes of the component can be mated coincident to the planes of the assembly (Front to Front, Top to Top, Right to Right). Now when the "hole plate" is placed in the assembly, the planes of the component can again be mated coincident to the planes of the assembly. This mating simplifies the analysis of the tolerances. Dimension the holes in both plates from the center of the plates. By doing this, it is simulating the way these components will be assembled. The "pins" will center on the holes and the alignment of the perimeter edges of the plates is a function of these pin and hole locations.

SolidWorks provides several good tools to use throughout this project. There are two holes in the hole plate that will vary in diameter. For the purposes of this project both holes will vary together. To keep the holes equal, one of several tool can be used. 1) Sketching one hole, then mirroring it to the opposite side of the part will maintain the same diameter in both locations. 2) Adding an "equal" geometric relation between each circle circumference. 3) Link Values to make two dimensions equal value. This project will use Link Values tool between the two hole diameter dimensions.

Names for each dimension are generated as the dimensions are placed in the model. These dimension names make the dimension unique. These unique dimension names are used to specify the dimensions in equations or a design table. To display these names go to the Tools pulldown menu, then pick Options, then switch to the General tab. In the upper left corner check the box for Show Dimension Names. Now when the dimensions are visible, the dimension names will also be shown in parenthesis below the value. These dimensions can be renamed by right selecting on the dimension value, then pick Properties. The Name window shows the portion of the dimension name that can be changed to

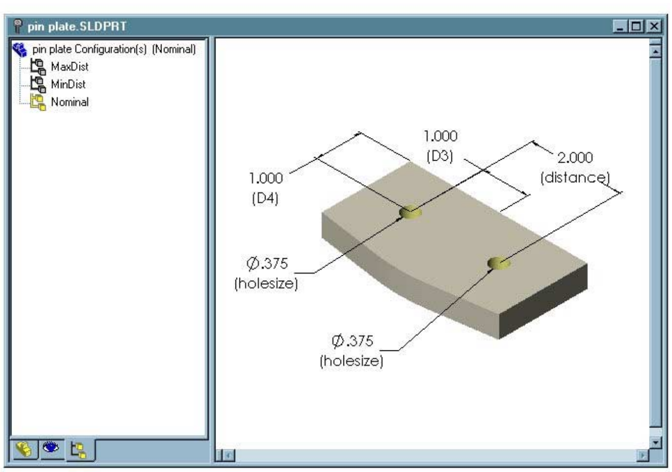
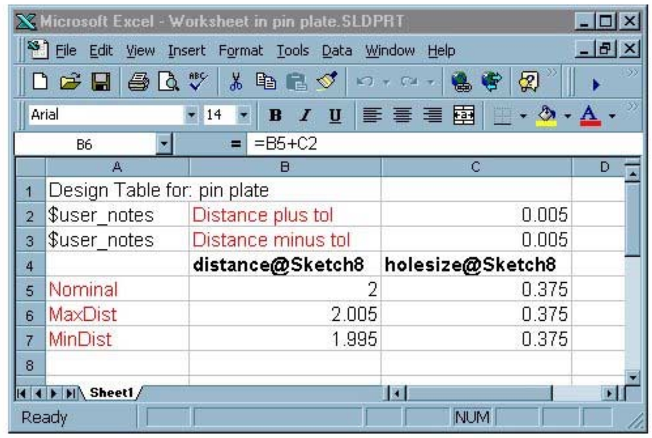


Figure 3



Microsoft Excel - Worksheet in pin plate.SLDPRT

	A	B	C
1	Design Table for: pin plate		
2	$user_notes	Distance plus tol	0.005
3	$user_notes	Distance minus tol	0.005
4		**distance@Sketch8**	**holesize@Sketch8**
5	Nominal	2	0.375
6	MaxDist	2.005	0.375
7	MinDist	1.995	0.375
8			

Figure 4

make this name user friendly. This project has several dimensions renamed.

Pin Plate:
The distance between holes → Renamed to "Distance"
The distance between centerline and left hole → Renamed to "Distance/2"

Hole Plate:
The distance between holes → Renamed to "Distance"
The distance between centerline and left hole → Renamed to "Distance/2"

An Equation is employed to keep the "distance/2" dimension equal to the "distance" dimension divided by 2. This could have been done using mirror command also. In other words by using a mirrored sketch circle for the two holes, both the equal diameters and the distances from the centerline of the part to each circle would have been maintained (mirroring the circle may have been a better choice). This could also be accomplished using a part design table. For the value of the "distance/2" dimension, place a formula in the cell to divide the "distance" dimension by 2.

A design table is basically a Microsoft Excel spreadsheet that is created to organize configurations of parts and assemblies. The part design table can control all, or a portion of, your dimension values in your model for each configuration of the part. The part design table can also control whether a feature is suppressed or unsuppressed. The real power of these design tables is that you can use the entire Excel functionality in the design tables. All functions, formatting, and tools are available.

The basic layout of a part design table is configuration name(s) are listed down the first column. The dimension name(s) and feature name(s) that will be controlled are

continued on page 3

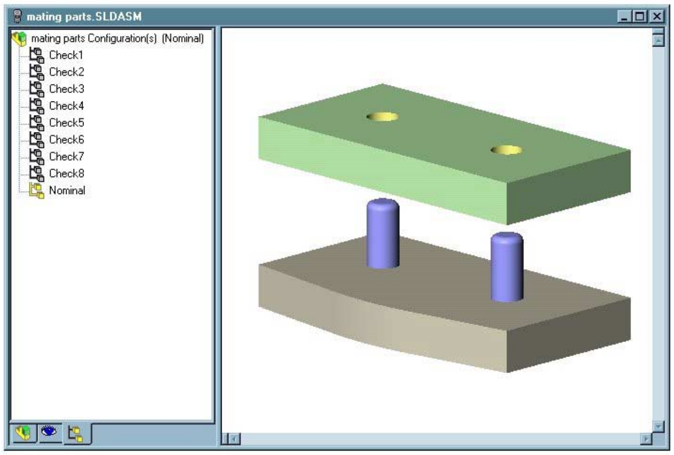


Figure 5

…continued from page 2

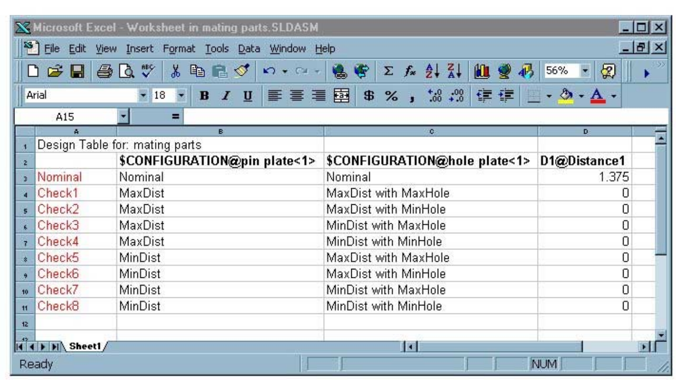
in the first row across the top. Then the other cells in the matrix are filled in with either values of dimensions, or the words "suppressed" or "unsuppressed". To put a row or column of calculations or comments that will be ignored by SolidWorks, substitute "$user notes" for the configuration name or parameter.

Part Design Tables are used to create families of configurations within a single part file. For the purposes of this project, the configurations of the part will consist of all the combinations of worst case dimensions for each part. Then in the Assembly Design Table we can create assembly configurations that in turn specify which configuration of each part will be the active configuration for each instance of the component in the assembly.

Configurations for all the combinations of the minimum and maximum dimension values were created based on the tolerance values for each dimension. Figure 2 shows the design table for the hole plate. The 2^{nd} through 5^{th} row of the design table are user to store the plus and minus values of the tolerance for each dimension (i.e. Distance plus tolerance = .005) The first configuration is located in row 7 and is named "Nominal". For the parameter "distance@sketch2", located in column B, the value is 2.000 (located in cell B7). Then the additional configurations down column A are all the possible combinations of worst case dimensions. "MaxDist with MaxHole" configuration denotes a hole plate with the maximum value of "distance@sketch2" combined with a maximum value of "holesize@sketch2". To fill in the values of the parameters, formulas are used. In cell B8, the formula used is =B7+C2. This takes the nominal value for "distance@sketch2" from cell B7 and adds to it the "Distance plus tolerance" from cell C2, yielding the value 2.005. SolidWorks will only pass the final value of the cell for the parameter. The same method is used to fill in all of the other parameter values in this design table.

Assembly Design tables control various parameters related to multiple configurations of an assembly. These are extremely powerful, controlling parameters such as the state (suppressed or unsuppressed), or the specific part configuration name that is active of each individual component instance in the assembly. The values of parameters such as distance and angle mates can also be controlled for each assembly configuration. This project creates an assembly configuration for each possible combination of part configurations. There are four different hole plate configurations and two different pin plate combination, so that adds up to 8 assembly configurations. The syntax for the parameter to vary the part configuration in the assembly is $configuration@component<instance> where component is the part name, and instance refers to the instance number of the part in the assembly. Figure 6 shows the assembly design table for this project. Row 4 delineates the first assembly configuration named "Check1". This configuration activates the "MaxDist" configuration of pin plate for instance 1, and the "MaxDist with MaxHole" configuration of instance 1 of the hole plate. Continuing this procedure for the remainder of the assembly configurations creates all the eight possible combinations of part tolerances.

Interference detection can now be performed for each assembly configuration. Switch to the Configuration Manager Tab, double click on the assembly configuration to make it active. Then pick the Tools pull down menu, the pick Interference detection. Inside the top window of the dialog box, list the components to be checked by picking the components either from the graphics area, or the Feature Manager. To check the entire assembly, pick to the assembly name at the top of the Feature Manager. Then press the check button. SolidWorks checks for interference between components and reports any interference it finds in the bottom window of the dialog box. If interference is found between it also highlights the interference in the graphics area of the model. The gold outline represents the actual interference, and the blue bounding box is an orthographic box that



	A	B	C	D
1	Design Table for: mating parts			
2		**$CONFIGURATION@pin plate<1>**	**$CONFIGURATION@hole plate<1>**	**D1@Distance1**
3	Nominal	Nominal	Nominal	1.375
4	Check1	MaxDist	MaxDist with MaxHole	0
5	Check2	MaxDist	MaxDist with MinHole	0
6	Check3	MaxDist	MinDist with MaxHole	0
7	Check4	MaxDist	MinDist with MinHole	0
8	Check5	MinDist	MaxDist with MaxHole	0
9	Check6	MinDist	MaxDist with MinHole	0
10	Check7	MinDist	MinDist with MaxHole	0
11	Check8	MinDist	MinDist with MinHole	0
12				

Figure 6

encapsulate the entire interference. The accompanying dimensions are the delta X, Y, and Z of the bounding box. Now, this info is not displaying the solution, just showing the problem. Modeling with SolidWorks does sometimes still involve work.

For this project, when the original values shown in the figures are used, there is interference in several of the assembly configurations. One solution is to increase the hole diameter in the "hole plate" to a ∅.386. All configurations of the assembly is once again checked and no interference is found.

The extent this analysis even further, either the Measure tool and/or Reference dimensions could be added to the assembly, to determine the amount of interference of the amount of clearance between two parts.

This article was written to serve as a beginning introduction to several key SolidWorks tools that can be used to perform this type of analysis. It is an example of a one dimensional tolerance analysis. This analysis can be performed several other ways, and what is efficient for one company, may not be the best procedure for the other. Good Luck, this is really cool stuff! -(O||||O)-

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