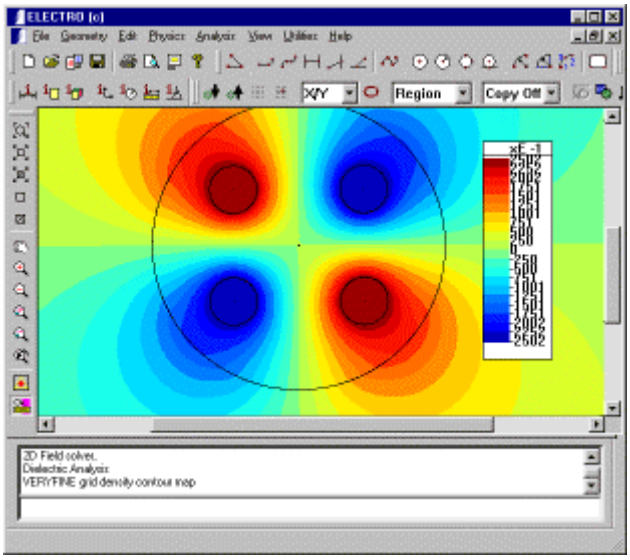
PARAMETRIC ANALYSIS

Parametric analysis is a design-optimization feature that allows you to use variables, rather than static values for specifying design parameters. You define parametric variables by type and by a series of values. You define all parametric variables in the same fashion as the non-parametric variables. Parametric analysis reduces the tedious, repetitive tasks associated with fine-tuning design parameters. For each parametric analysis, you can assign up to 16 variables.

"Page Down" or use scroll bars to read the article

INTEGRATED ENGINEERING SOFTWARE

2D Parametric Analysis



INTEGRATED
ENGINEERING SOFTWARE
www.integratedsoft.com

Table of Contents

PARAMETRIC ANALYSIS

Parametric analysis is a design-optimization feature that allows you to use variables, rather than static values for specifying design parameters. You define parametric variables by type and by a series of values. You define all parametric variables in the same fashion as the non-parametric variables. Parametric analysis reduces the tedious, repetitive tasks associated with fine-tuning design parameters. For each parametric analysis, you can assign up to 16 variables.

Parametric Variable Types

There are four types of parametric variables:

Geometry	Use a geometry variable when you want to optimize the size or position of different parts of the device.
Boundary Conditions	Use a boundary conditions variable to see the effect of different boundary conditions, including potential on a boundary and normal derivative of potential crossing a boundary.
Media	Use media to optimize materials properties, including permeability and different nonlinear curves, in regions of the model.
Sources	Use sources to see the effects of different sources on the model.

The Parametric Window Bar

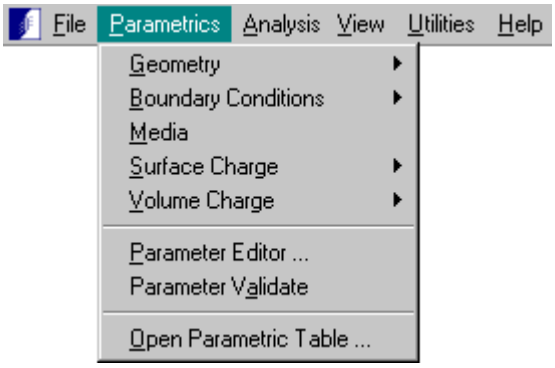
The parametric window contains five menus with commands that help create and analyze parametric design.

File menu

The Parametric File menu contains all of the Print and Save commands found in the main File menu. It does not contain the Open, Merge, Delete and Reset commands.

Parametrics menu

The Parametric menu contains the parametric variable sub-menus and commands as well as several commands editing for the parametric setup.

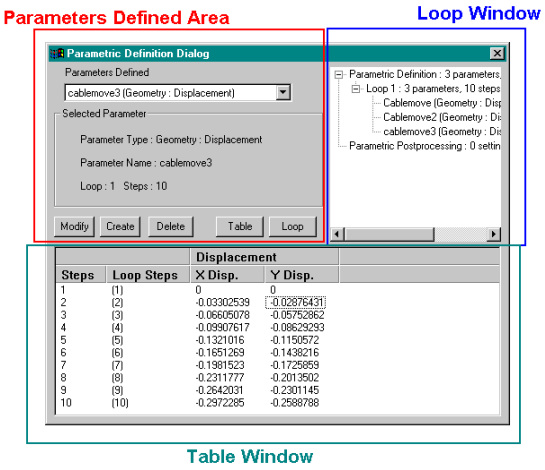


Parametric variable functions

Geometry	Contains the geometry modification commands - Displace, Rotate, Scale and Pull commands. These commands allow you to simulate geometric variations on a basic geometry.
Boundary Conditions	Contains the same commands as the main Boundary Conditions menu (Potential, dA/dn, Display and Delete).
Media	Select Media to create a material parametric variable. You can define this variable using the same Material Table that exists in the main Physics menu.
Sources	Contains the commands found in the main Physics menu.
Parameter Editor	Allows you to setup and edit parametric tables
Parameter Validate	Steps through the parametric variables
Open Parametric Table	Opens the current parametric table for editing.

The Parametric Definition Dialog is divided into three parts:

- The parameter defined section
- The Loop Window
- The Table Window



The Parameters Defined Area

Modify	Opens the Modify dialog to allow you to modify existing data.
Create	Opens the Create dialog to allow you to create a new parameter.
Delete	Opens the Modify dialog allowing you to delete existing parameters.
Table	Opens the Table window to display the step variables.
Loop	Opens the Loop window to display the Parametric Definition, Loop Structure, and Parametric Post Processing functions.

Performing a Parametric Analysis

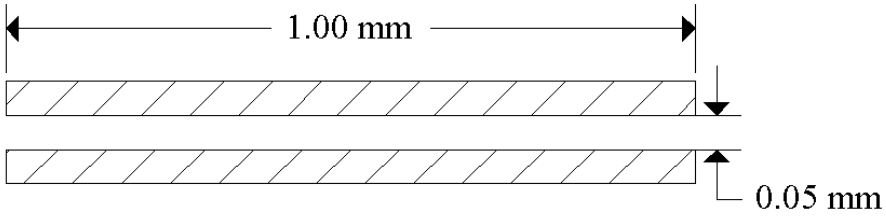
There are two methods of setting up a parametric analysis:

- One method utilizes the Parametric Editor command and the Parametric Definition dialog box.
- The other method utilizes the Parametrics menu commands and program prompts in the message area to guide you through the process.

Parametric analysis using the Parametric Editor

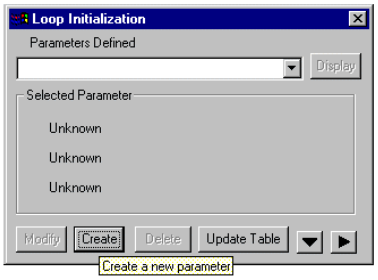
To set up a parametric analysis using the Parametric Editor command:

1) Create a basic model. Normally, this is the base design where you want to see the effect of different variables. As an example, we will vary the gap between the plates of a capacitor. The top plate is assigned 1 Volt and the bottom is grounded.

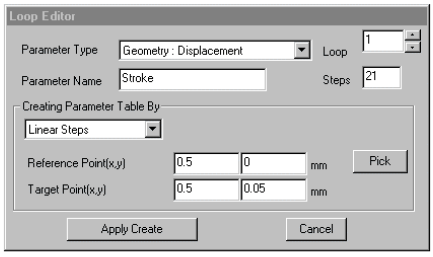


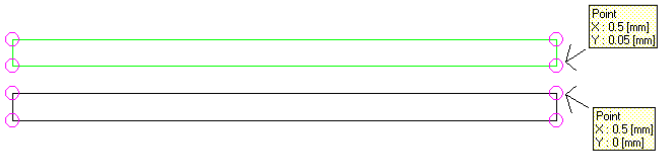
2) From the Utilities menu, select Parametric On/Off. This changes the menu bar to the parametric menu bar.

3) Select Loop Initialization from the Parametric menu to open the Loop Initialization dialog box.



4) Click the Create button on the Loop Initialization dialog box to open the Loop Editor dialog.





5) In this box, first select the Parameter Type; in this case a Geometry Displacement.

6) Every parameter must have a name (if you require more than one parametric variable, each name must have a unique first letter) in this case use “Stroke”.

7) Specify a Loop number (the program can support up to 4 nested Loops). In this case, there will only be one loop, so leave the default setting to 1.

8) Enter the number of steps (the number of different values assigned to the variable under examination) as 21.

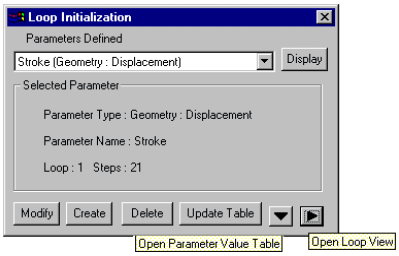
9) Make sure that the geometry selection type is set to regions. Then, using the mouse, select the top plate of the capacitor as the region that is to be displaced. (The Apply Create button will then become active but the parametric definition is not yet complete.)

10) You can have parameters vary linearly (equally spaced intervals) or non-linearly, or by a Function, or from a File. In this case, use the default Linear Steps.

11) The Reference (start) and Target (end) points for the displacement can be entered directly into the dialog, or can be selected using the mouse. To use the mouse, press the “Pick” button and select the points shown above.

12) Click the Apply Create button. The Loop Editor will close.

13) The Loop Initialization dialog will be updated with the information on the Stroke parameter and appear as shown:

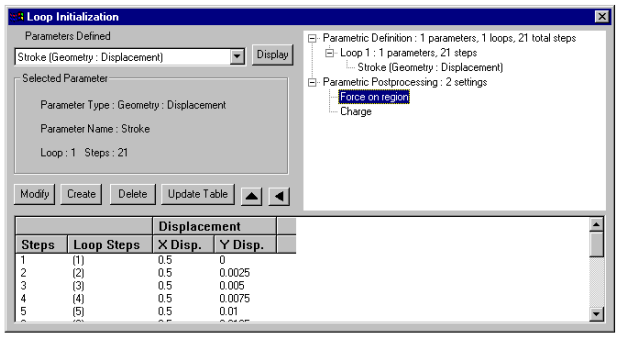


14) Before continuing, click the two arrow buttons to open the Loop View and Parameter Value Table.

15) From the Analysis menu, select Force>Region and select the bottom plate of the capacitor when prompted. Right click to end the selection.

16) From the Analysis menu, select Charge and then select the top plate when prompted. Right Click to end the selection.

17) The Loop View section of the dialog box will show the post processing you have defined. It should appear as shown below:

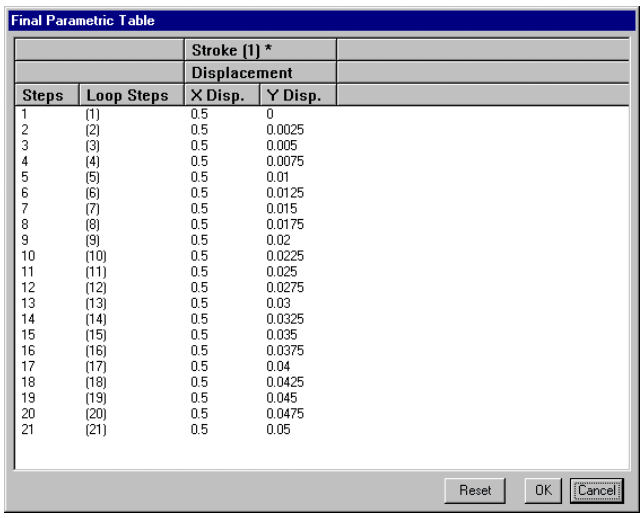


Selecting Parameters or Post Processing items in the Loop View causes the corresponding geometry to become high lighted in the main program window.

18) At this point the parametric and post processing can be modified if desired. The final definition of the parametric does not occur until you press the “Update Table” button.

19) To verify that the top plate will be displaced during the parametric run, select Parameter Validate from the Parametrics menu. This should cause the top plate to become high lighted and then move vertically up to the maximum displacement and back again to the start position.

20) Once the parametric has been validated, press the “Update Table” button on the Loop Initialization dialog box. This will update and open the Final Parametric Table which should appear as shown below:

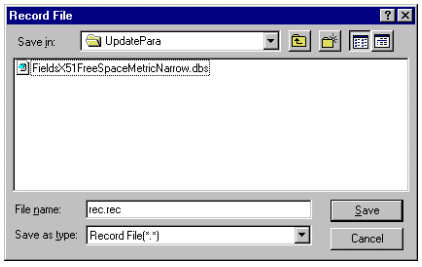


21) This table can be edited if desired. If it is correct, press the OK button to close it.

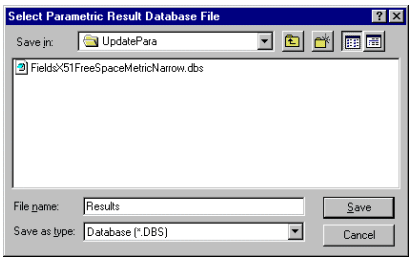
Now the parametric setup is complete. From the Utilities menu, select Parametric On/Off. This closes the parametric window and returns to the main menu bar.

To run the parametric analysis:

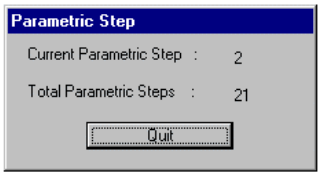
1) From the Utilities menu, select Run Parametric. A save dialog box will open to allow you to specify the name of a Record File that will record each parametric step. Enter any name you wish (such as “rec.rec”) and press the Save button.

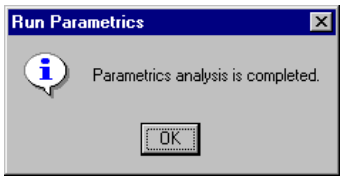


2) A second save dialog box will open to allow you to specify a database that will contain a table of the post processing results. You can specify a new name (in this case “Results”) if you do not wish to overwrite your base model.



3) Once you click the Save button, the parametric run will start. During the run, the current step number will be displayed.





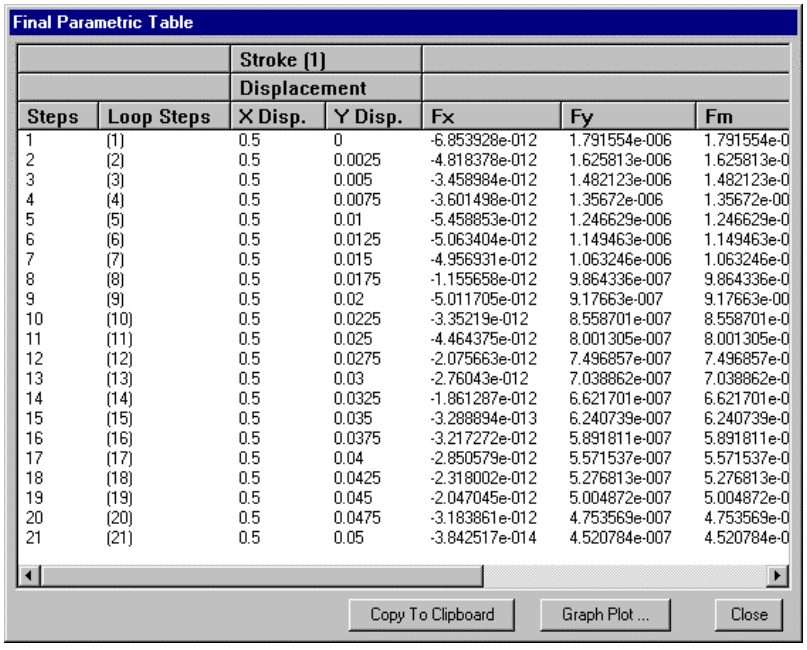
4) When the parametric run is completed, you will be informed by the dialog box.

After the solution is complete, you can find the results for each parametric step in the same directory as the original model. The results filename (8 characters in length) starts with P followed by the step number, e.g. P00000001.dbs for the first step.

Viewing the Parametric Results

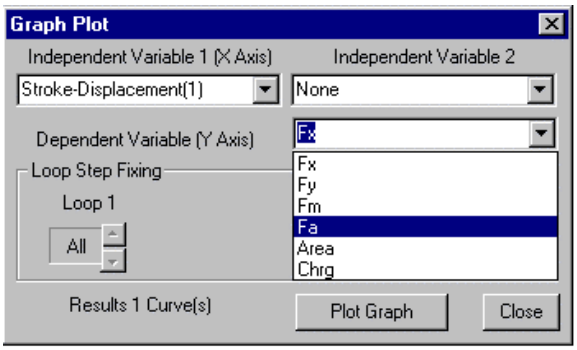
The results of the parametric run can be viewed in table or graph format by performing the following steps.

1) Select Parametric Results from the Utilities menu. This will open the Final Parametric table.



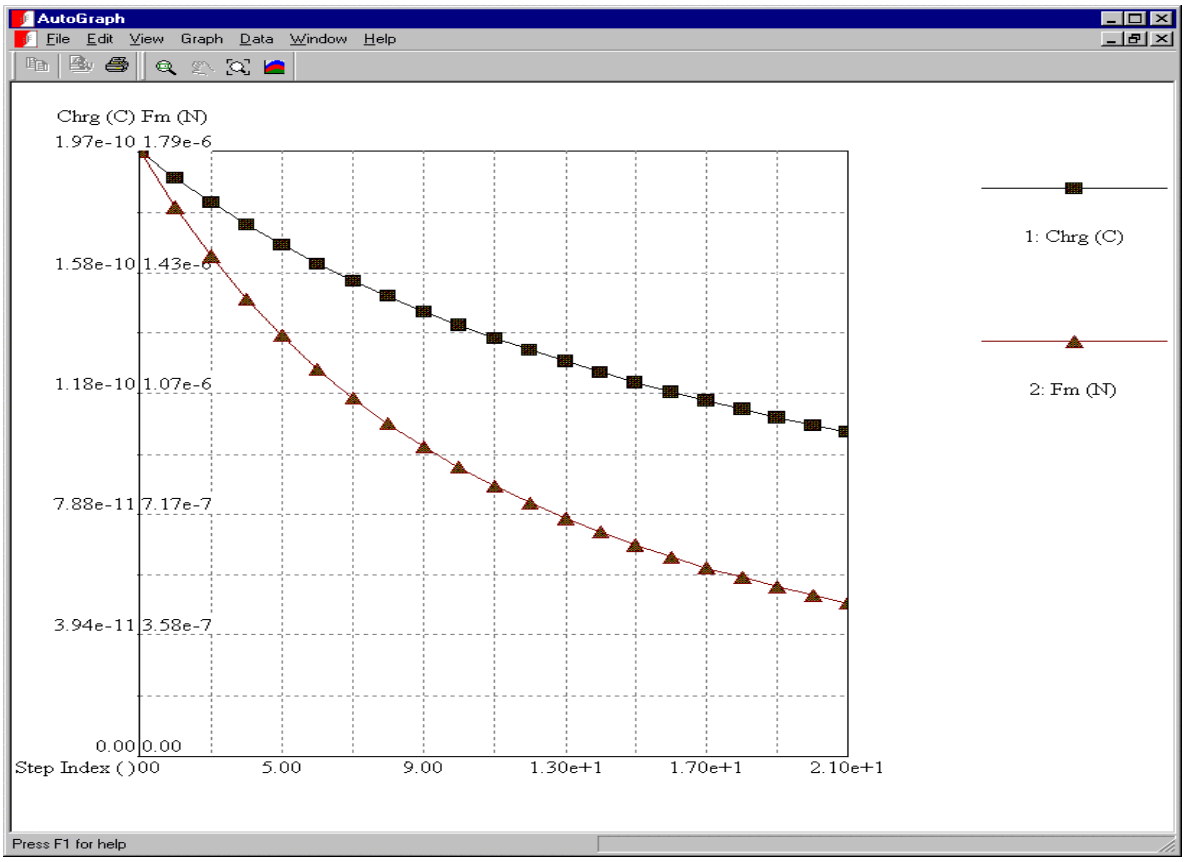
2) There is a button at the bottom of the table that allows you to copy the data to the Windows Clipboard for use by other applications.

3) Clicking the Graph Plot button will open the Plot Graph dialog box.



4) Using the drop down list, select a dependent variable to plot and click the Plot Graph button to produce a graph using the Autograph utility. Do this for each variable you wish to plot.

5) The following graph shows that the charge on the top plate decreases (approximately) inversely with the gap while the force on the bottom plate decreases (approximately) according to the inverse square of the gap. This is in close agreement with simple analytic formulas (discrepancies are due to the fact that the analytic formulas do not consider fringing effects).



This completes the example.

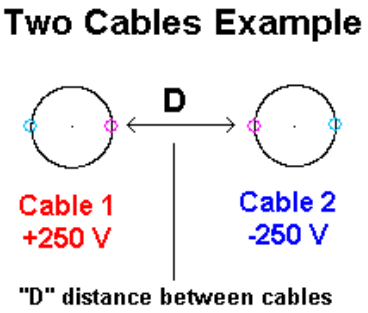
Parametric analysis using the Parametrics menu

To set up a parametric analysis using the Parametrics menu commands:

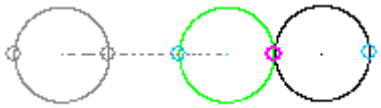
1) Create a basic model.
2) From the Utilities menu, select Parametric On/Off.
3) Select the parameter that you want to vary from the Parametrics drop down list.
4) Follow the prompts in the program message area.

We will use the displace geometry variable as an example.

In this example we have two cables, with equal voltages, but with opposite polarities. We will observe the interaction of the Voltage fields as the distance between the two decreases.



5) From the Utilities menu, select Parametric On/Off.
6) Select the Parametrics, then Geometry then Displace.
7) Enter “move” for the name for the parameter.
8) Enter 10 for the Number of displacement steps at the prompt.
9) Enter N at the prompt “will the values be imported from a file?”.
10) Enter N at the prompt “Will the values be expressed as a Function?”
11) Enter Y at the prompt “ Will the values be linearly spaced?”
12) Select Cable 1 at the prompt “Select Regions (or segments, or objects)”.
13) Select the center point of the cable as the reference point and press the right mouse button.
14) Move the outline of the cable from its original placement, to a point where the two conductors are touching, and press the right mouse button.



15) From the Parametrics menu, select Parametrics validate, and watch Cable 1 move towards cable 2.

The physical movement of cable 1 to cable 2 is now defined.

To assign the post processing functions to the parametric:

1) Select Analysis from the Analysis menu.
2) The model will solve.
3) Select Field Results from the Analysis menu.
4) Set the Field Analysis Results to:

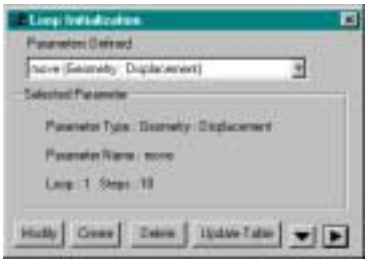
   Voltage
   Contours
   Medium density
   31 Bands
   Select the solid option

5) Press Apply Settings.
6) Close the Field Analysis Dialog box.

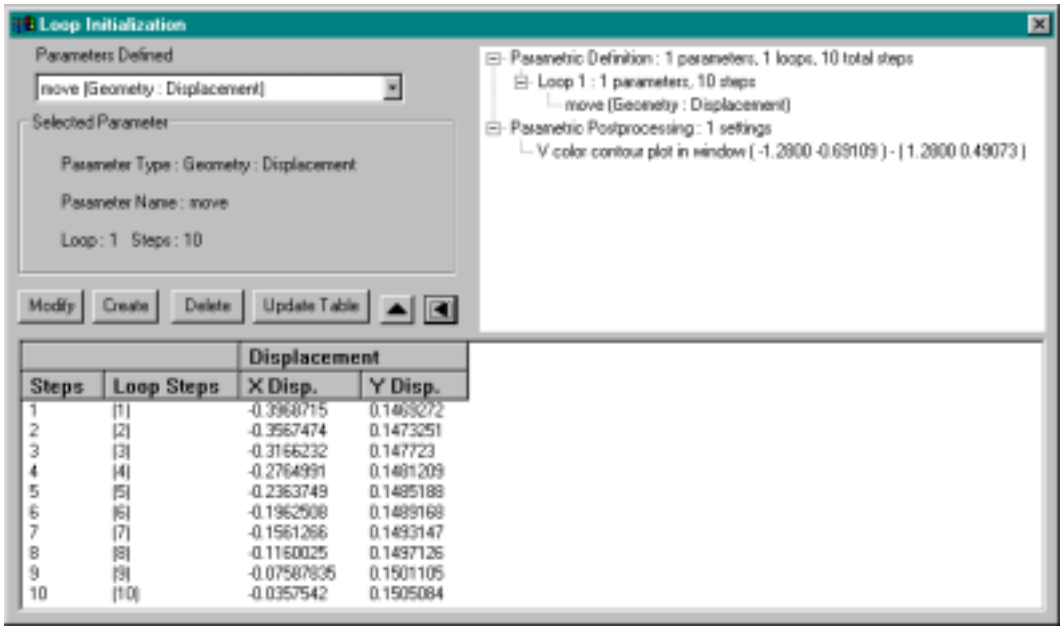
This adds the post processing option for the example

To View the Parametric table and definitions:

1) Select Loop Initialization from the Parametrics menu.
2) Press the down arrow and right arrow buttons.



3) The Dialog opens up to allow you to view the steps and parameters involved in the parametric analysis.



		Displacement	
Steps	Loop Steps	X Disp.	Y Disp.
1	{1}	-0.3968715	0.1493272
2	{2}	-0.3567474	0.1473251
3	{3}	-0.3166232	0.147723
4	{4}	-0.2764991	0.1481209
5	{5}	-0.2363749	0.1485188
6	{6}	-0.1962508	0.1489168
7	{7}	-0.1561266	0.1493147
8	{8}	-0.1160025	0.1497126
9	{9}	-0.07587835	0.1501105
10	{10}	-0.0357542	0.1505084

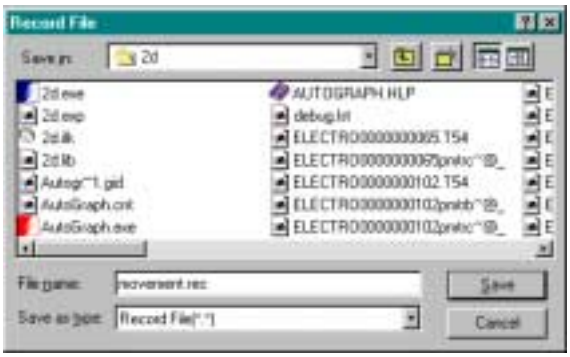
4) At this point if required, you could edit the parametric analysis.

5) Close the Dialog box.

6) Select Parametrics On/Off from the Utilities menu.

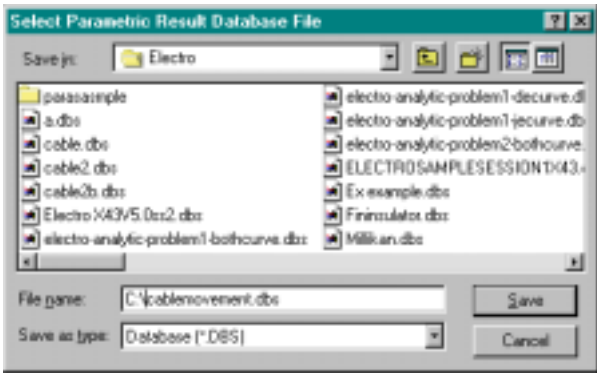
7) Select Run Parametric from the Utilities menu.

8) The record file dialog box opens



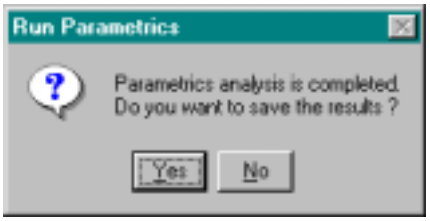
9) Enter movement.rec as the record file and press save.

10) The result database file dialog opens



11) Enter cablemovement.dbs as the file name and press save.

The program runs the parametric analysis, and displays the dialog box, when complete.



12) Press Yes and enter a file name for the completed parametric results.

Viewing the Parametric Results

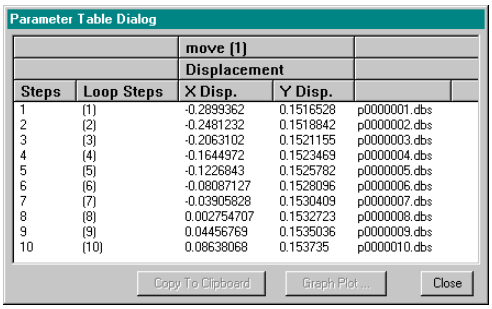
In the Cable sample, we had generated the results for the voltage contour plots at 10 positions of Cable 1 vs. Cable 2. The Parametrics function created 10 databases for us, each containing the results for that particular step.

1) To view the results for a particular step:

2) Select Parametric Results from the Utilities menu.

3) This opens the Parameter Table dialog.

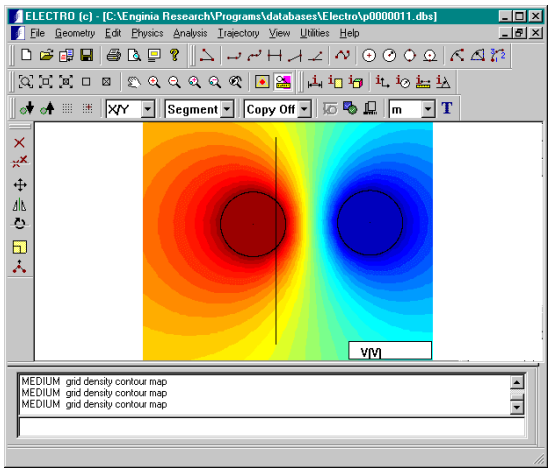
4) In this case the Parameter Table dialog shows you the cross reference between each step and the database for it's results.



5) To view the results at a particular step, select File, Open from the main menu of the 2D program.

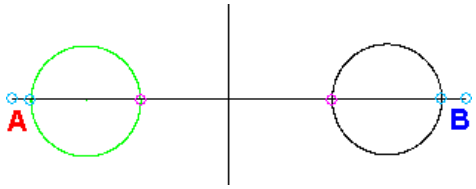
6) Select any one of the listed Databases.

7) When the database opens, the plot for that step is displayed.



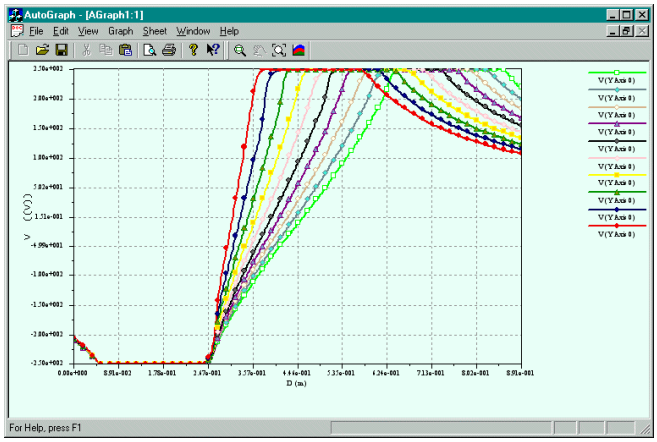
8) This method of displaying the results is used for Contours, Arrows, Loci, and Streamline Plots. For Graphs along a segment or line, the results are produced a little differently.

9) If we added an arbitrary segment running through the centers of each of the cables, we could graph the voltage on that line at the various points of Cable 1's movement.



When you select Run Parametrics from the Utilities menu, the Parametric analysis opens the Autograph utility and sends the data to it.

You can see that there is a graph plotted for each step of the cable's movement.

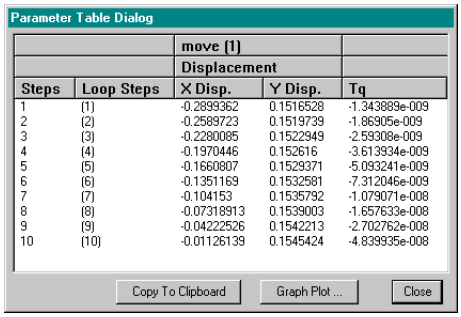


Single value Results are items that produce one number for each step of the analysis. They are:

- Force
- Torque
- Flux Linkage
- Induced Voltage

They will depend on the type of program that you have, (electrostatic, magnetostatic, etc.)

The results for a Torque analysis on the cable are displayed below.

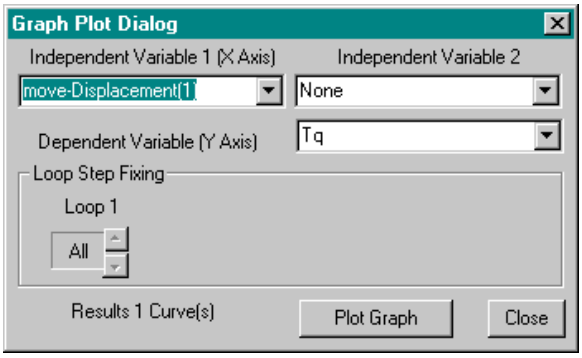


Parameter Table Dialog

		move (1)		
		Displacement		
Steps	Loop Steps	X Disp.	Y Disp.	Tq
1	(1)	-0.2899362	0.1516528	-1.343889e-009
2	(2)	-0.2589723	0.1519739	-1.86905e-009
3	(3)	-0.2280085	0.1522949	-2.59308e-009
4	(4)	-0.1970446	0.152616	-3.613934e-009
5	(5)	-0.1660807	0.1529371	-5.093241e-009
6	(6)	-0.1351169	0.1532581	-7.312046e-009
7	(7)	-0.104153	0.1535792	-1.079071e-008
8	(8)	-0.07318913	0.1539003	-1.657633e-008
9	(9)	-0.04222526	0.1542213	-2.702762e-008
10	(10)	-0.01126139	0.1545424	-4.839935e-008

Copy To Clipboard | Graph Plot ... | Close

1) To plot a graph of the torque values vs. cable displacement, press Graph Plot. This brings up the Graph Plot dialog.



2) Select the Plot Graph button and the plot of the Torque is displayed in the AutoGraph utility.

Please Note: You may have to format the graph view ranges in order to display the graph as shown.