

The Focus



A Publication for ANSYS Users

Contents

Feature Articles

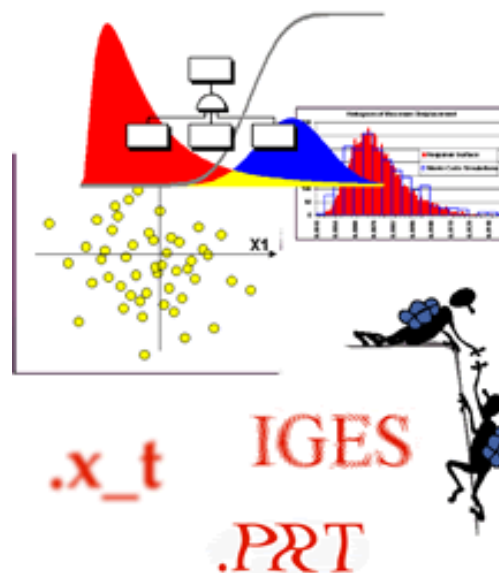
- [Probabilistic Design System](#)
- [Enhanced Technical Support](#)
- [~Heal Method](#)

On the Web

- [ANSYS Offers Probabilistic Design Seminars](#)
- [PADT Seminar: Advanced Analysis Applications: Impact, Metal Forming, HF EMAG, and MEMS](#)

Resources

- [PADT Support: How can we help?](#)
- [Upcoming Training at PADT](#)
- [About *The Focus*](#)
 - [The Focus Library](#)
 - [Contributor Information](#)
 - [Subscribe / Unsubscribe](#)
 - [Legal Disclaimer](#)
- [Next in *The Focus*](#)



ANSYS
Support Distributor

© 2002, by Phoenix Analysis & Design Technologies, Inc. All rights reserved.

The Focus



A Publication for ANSYS Users

Probabilistic Design System

by [Rod Scholl](#)

Overview

At some point, most ANSYS users have had to vary some parameter in their model to investigate impact on results. Perhaps you've done this through the GUI and multiple runs or maybe you've written a macro that Do Loops through a few input values. ANSYS Probabilistic Design System (PDS) automates the process.



Using simple menu picks (or commands) you can specify many input variables and their variation in statistical terms (Gaussian, Weibul, etc.) and then ANSYS manages the many runs that sample the terrain. You can choose the sampling method (Random, Monte Carlo, etc.). Finally ANSYS builds an HTML report (via the Automatic Report Generator) that graphs the specified output variables. It will even evaluate correlations between inputs and outputs! This is the best application of the Report Generator that I've encountered.

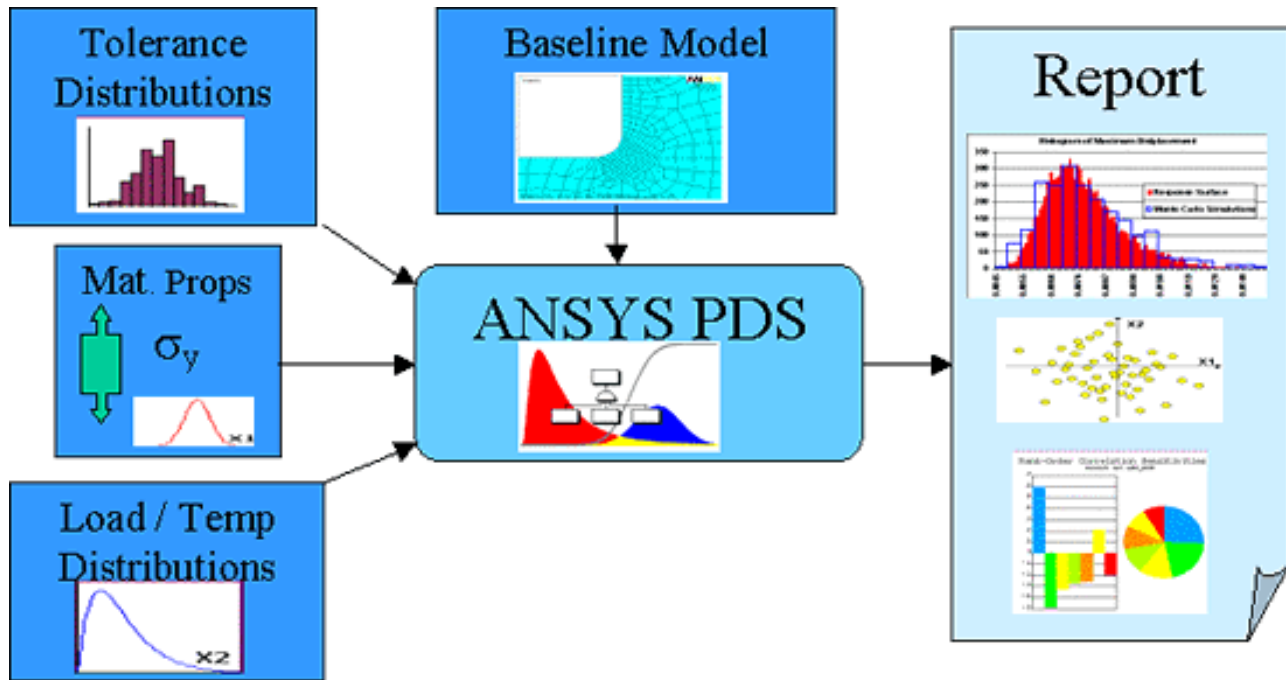
An Ideal-World Case Study

Your materials person gives you Gaussian distributions for your properties. Your dimensions are specified with tolerances as well as distributions based on SPC generated data at the vendor. (This is why this is titled Ideal-World case.) You also receive boundary conditions that are in terms of means and standard deviations assuming a Gaussian distribution. You set up the model as usual with a single case, but then specify inputs statistically in the PDS module. ANSYS runs the model 1,000 times (I've seen people do as much as 40,000) and summarizes the results in a report. You can then provide a distribution of peak stress for your lifing person. Unless you're the lifing person, and then you can specify a probability of life rather than a discrete number of cycles!

The Focus

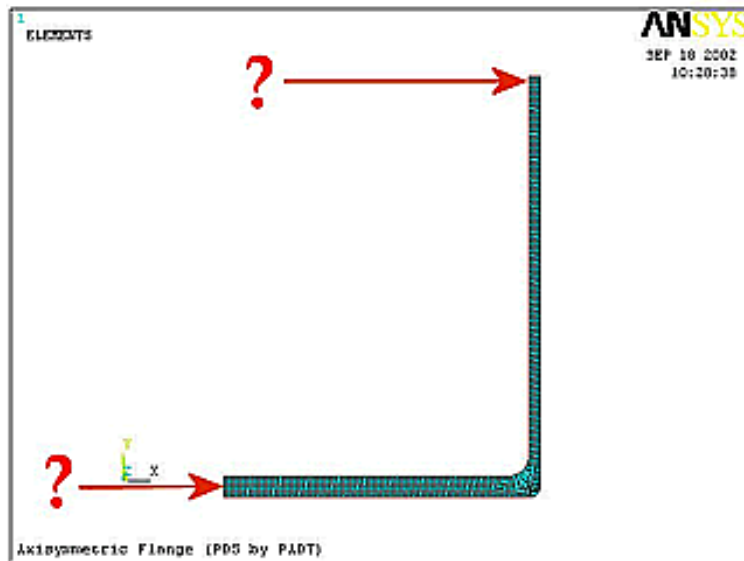


A Publication for ANSYS Users



A Real-World Case Study

You are given the model of a flange with fixed dimensions and material properties. However, the deflections are unknown at the inner and outer radii. Not unusually, you must know/characterize the stress before the deflections are finalized.



The Focus



A Publication for ANSYS Users

The Procedure

1. Build a database that solves for a single case and verify results.
2. Change the database to have the BC s applied using variables.
3. Specify statistical distributions of the input variables using PDS.
4. Specify desired output variables using PDS.
5. Instruct PDS to solve the model multiple times.
6. Review the report generated by PDS.

1. Build a database that solves for a single case

You should cut and paste this text to a file then name it `FLANGE . INP`. You can then read in the file *into a clean database* using Utility Menu > File > Read Input From&.

```

/PREP7
/title,Axisymmetric Flange (PDS by PADT)
UIMP,1,EX, , ,11.13948e6, !Sets Material Properties
UIMP,1,NUXY, , ,.3326,
K,1,6,2 !Draws Outline
k,2,6,0
k,3,4,5,0
k,4,4,5,.1
k,5,5,95,.1
k,6,5,95,2,
LSTR, 2, 3 !Makes Lines
LSTR, 2, 1
LSTR, 1, 6
LSTR, 5, 6
LSTR, 5, 4
LSTR, 4, 3
LFILLT,1,2,.05, ,
LFILLT,5,4,0.1, ,
al,5,8,4,3,2,7,1,6 !Makes Areas
et,1,82 !Meshes
keyopt,1,3,1
esize,.0175
amesh,all

/solu
dl,6,,ux,.05 !Apply Deflection Loads
d,346,,.025,,,,ux
d,2,,0,,,,uy
solve
  
```

The Focus

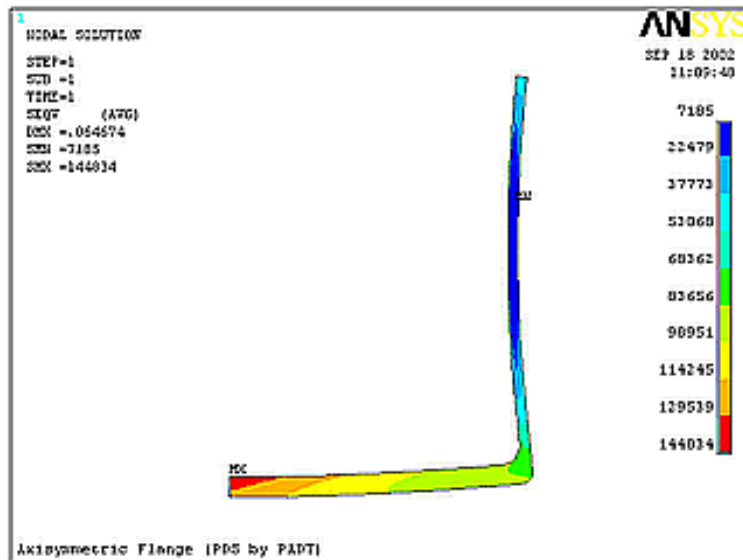


A Publication for ANSYS Users

```
/post1
PLNSOL,s,eqv
!sel,s,,,6 !Store Output Variables
nsl,s,1
nsort,s,eqv
*get,INNERSTRS,sort,0,max
```

```
!sel,s,,,8
nsl,s,1
nsort,s,eqv
*get,CORNSTRS,sort,0,max
allsel
```

They should look as follows:



2. Change the database to have the deflections applied as variables

Original	Modified
<pre>/solu d1,6,,ux,.05 !Apply Deflection Loads d,346,,.025,,ux d,2,,0,,uy solve</pre>	<pre>/solu INNERDF=.05 OUTERDF=.025 d1,6,,ux,INNERDF !Apply Deflection Loads d,346,,OUTERDF,,ux d,2,,0,,uy solve</pre>

You should be able to rerun your macro and get the same results!!!

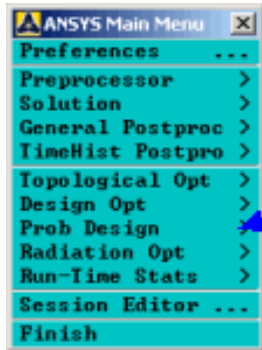
The Focus



A Publication for ANSYS Users

3. Specify Statistical Distribution of the Input Variables Using PDS

Open the Probabilistic Design module:



Specify Random Input Variables:

The following table represents the data shown in the 'Define a Random Variable' dialog's parameter list:

CORNSTRS
INNERDF
INNERSTRS
OUTERDF

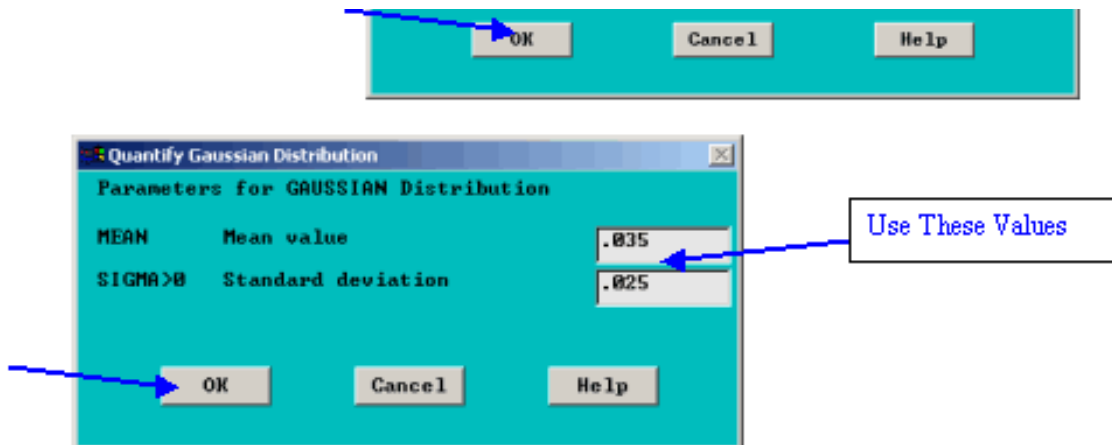
The following table represents the data shown in the 'Define a Random Variable' dialog's distribution type list:

Gauss	GAUSS
Trunc Gauss	IGAU
Lognormal 1	LOG1
Lognormal 2	LOG2
Triangular	TRIA
Uniform	UNIF
Exponential	EXPO
Beta	BETA
Gamma	GAMA
Weibull	WEIB

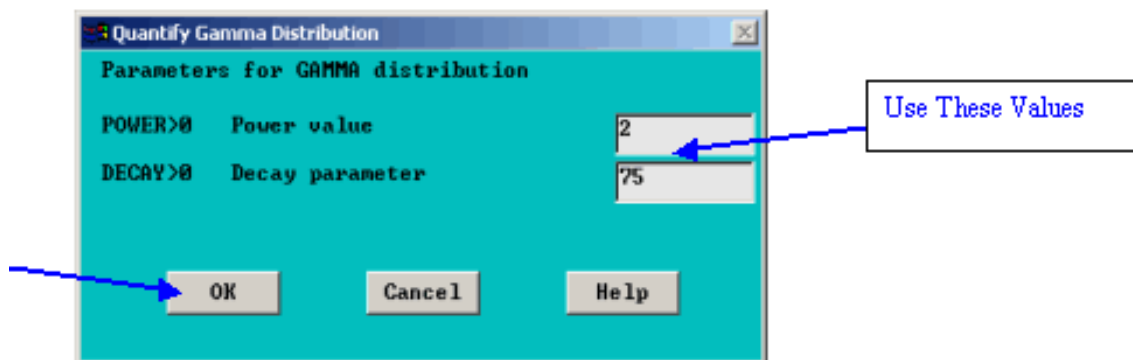
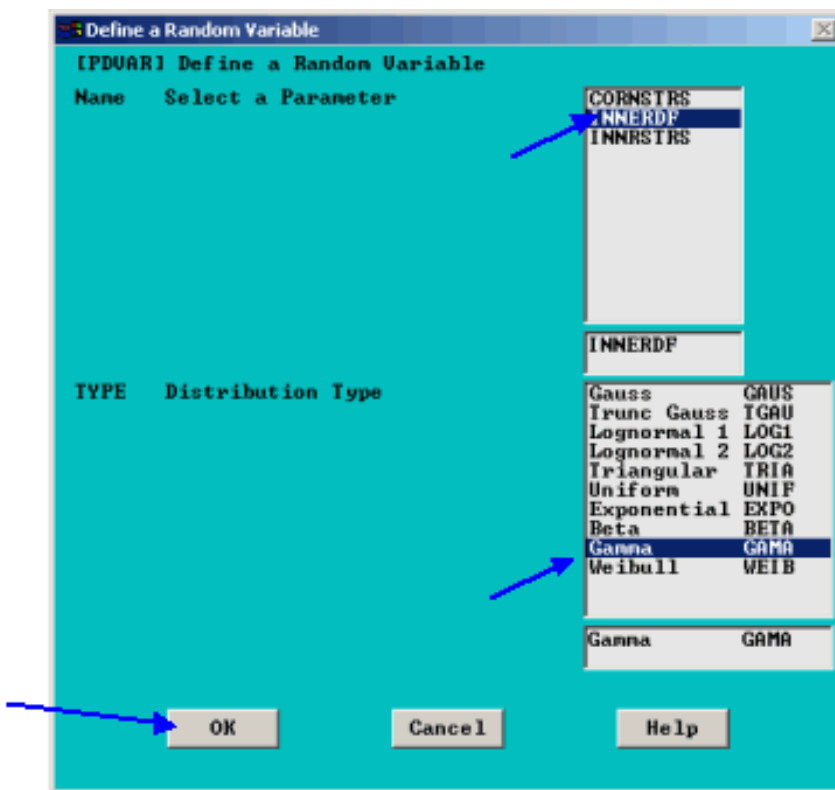
The Focus



A Publication for ANSYS Users



Do the same for the variable INNERDF. Rather than Guassian, choose Gamma:



The Focus



A Publication for ANSYS Users

4. Specify desired output variables

Note that we had to define these *first* using *GET s at the end of our original macro. PDS will not do this for you!

The figure illustrates the steps to define random output parameters in the software. It consists of four sequential screenshots:

- Prob Design Menu:** The 'Random Output...' option is highlighted in the 'Prob Design' menu.
- Random Output Parameters Dialog:** The 'Add...' button is highlighted, indicating the next step to define parameters.
- Define a Random Output Parameter Dialog (Step 1):** The 'CORNSTRS' parameter is selected from the list.
- Define a Random Output Parameter Dialog (Step 2):** Both 'INNERDF' and 'INNERSTRS' parameters are selected. A callout box points to 'INNERSTRS' with the text: "Add INNERSTRS also."

The Focus

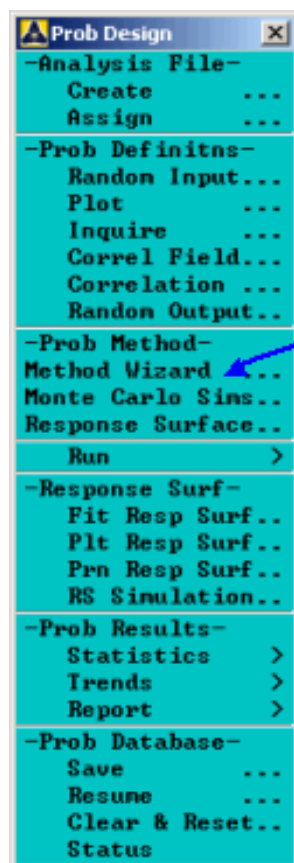


A Publication for ANSYS Users

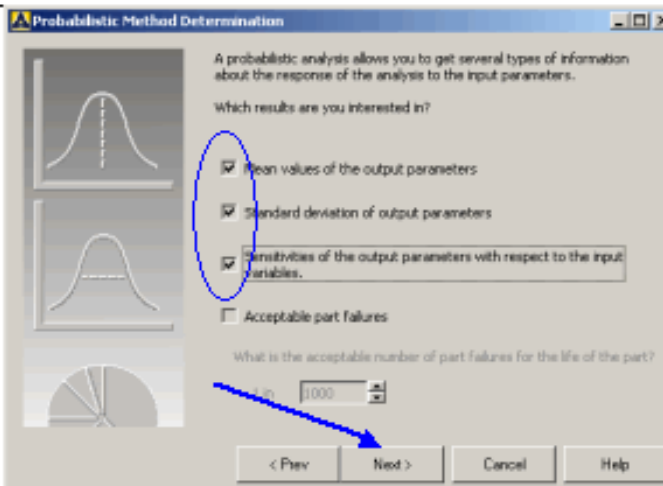
5. Instruct PDS to solve the model multiple times

PDS is designed to distribute these runs over multiple computers in your company. It will compile all the results back on your own PC. This distributed computing is not too difficult to set up but for this example we will run them all on our own machine.

Use the Method Wizard&



- Skip past the first screen with the clock hitting NEXT
- Accept default of No by hitting NEXT
- Select the 1st three boxes as shown below then hit NEXT



- Skip past the last screen by hitting FINISH

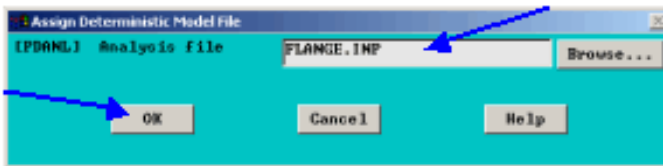
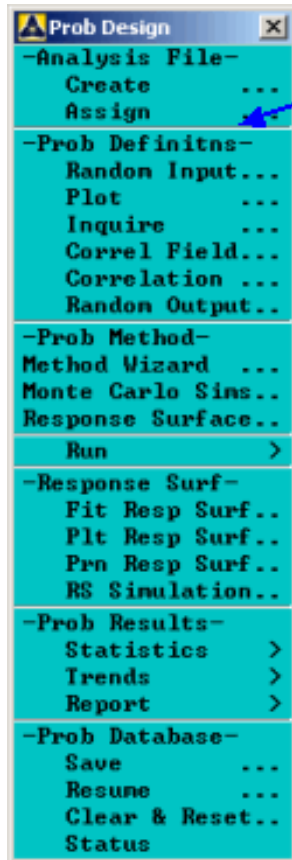
There are several other dials to turn, but these defaults will work in our case. The other bells and whistles are fairly self-explanatory; just browse around someday when you are bored.

You must Assign to ANSYS PDS what macro file to execute. This is the input file we named FLANGE . INP and then modified to have variables control the deflection.

The Focus



A Publication for ANSYS Users



DON'T FORGET THIS NEXT STEP!!!!

You must comment out (using the !) the variable definitions in your macro. Or else, PDS will run the same version 100 times!

Original	Modified
/solu INNERDF=.05 OUTERDF=.025	/solu ! INNERDF=.05 ! OUTERDF=.025

(Or you could have just deleted the lines of code...)

(Don't forget to save the file!)

Another Note – if you used any /CLEAR's in a macro – you must get rid of them.

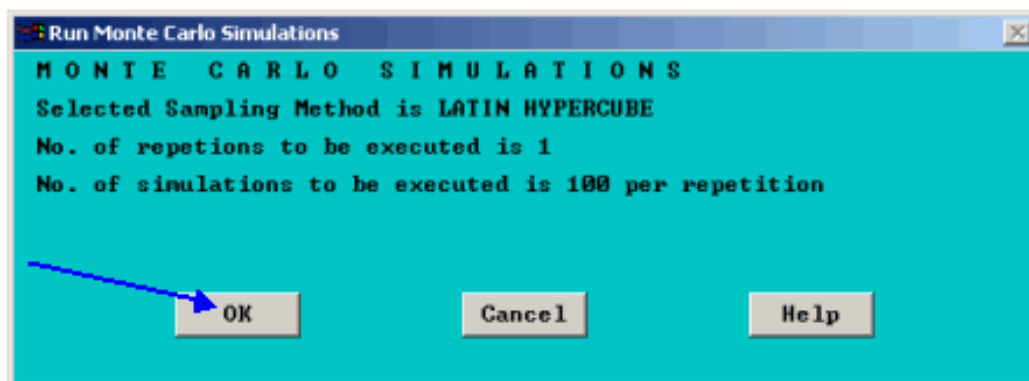
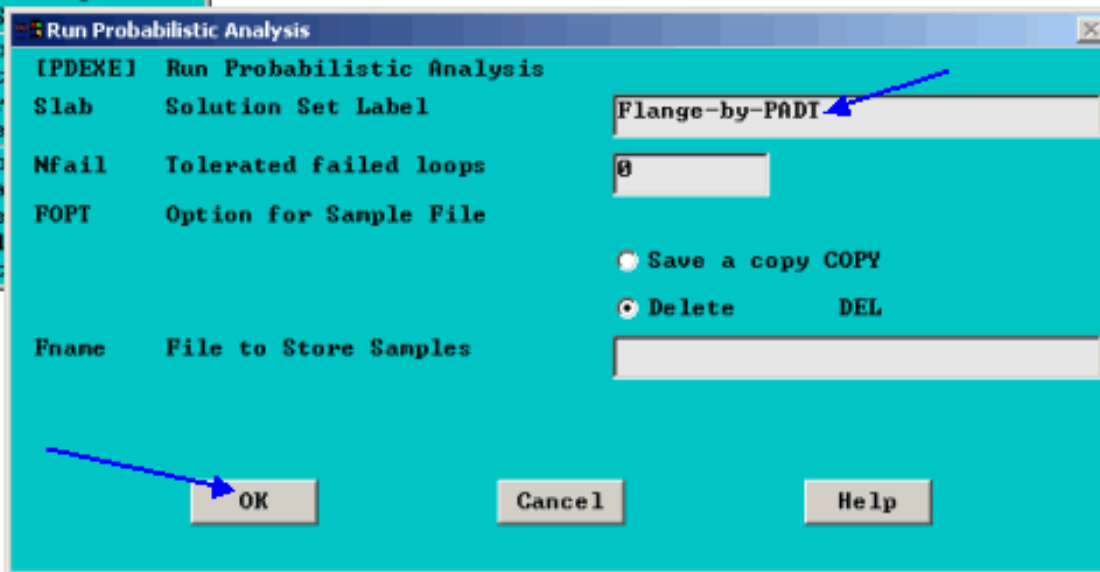
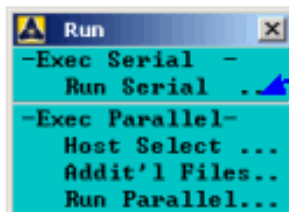
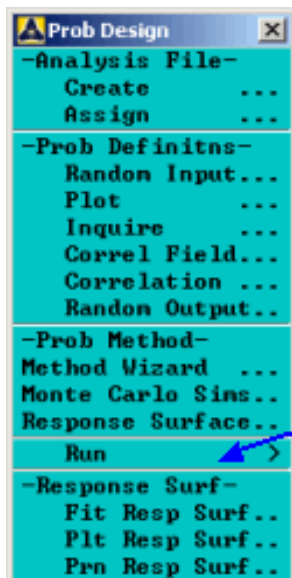
Yet Another Note – if you used any RESUME's in a Macro you must preface it with PARSAV and PARRES to help ANSYS track the PDS variables.

Now you are ready to go. Just a few more clicks&

The Focus



A Publication for ANSYS Users



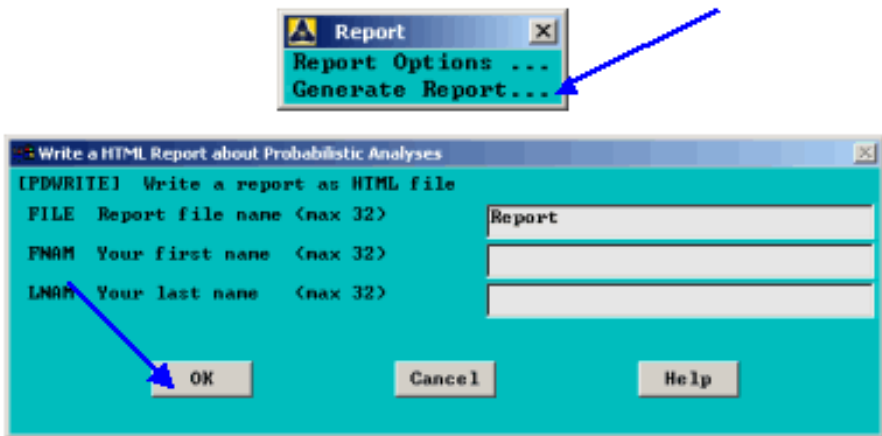
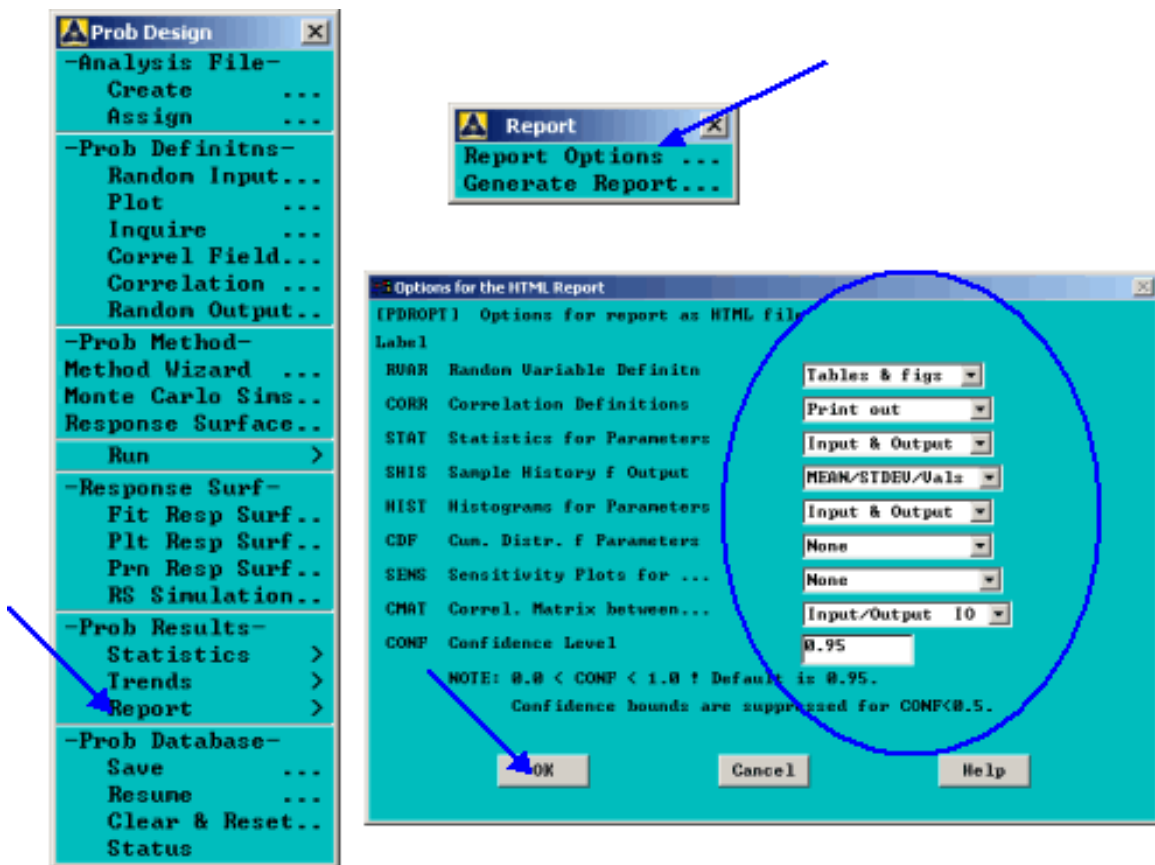
The Focus



A Publication for ANSYS Users

6. Generate Report

Use whatever settings you like, I chose these just for kicks&



You can view the entire content of automatically-generated [HTML report](#) at your leisure. Also, there are other files stored (such as parameter history for each run, etc.) which are not addressed here. See the ANSYS documentation for details.



A Publication for ANSYS Users

Acknowledgement

Thanks to Stefan Reh, from whom s presentation I swiped some graphics!



The Focus



A Publication for ANSYS Users

Enhanced Technical Support

by [Rod Scholl](#)

A part of my day I most enjoy is supporting Enhanced Technical Support agreements. We provide this service to a host of customers under other names like Mentoring , Customization , etc. I suppose I enjoy it, because I usually get to help people in a minor jam. I hear Tow Truck drivers have similar experiences.

Are you ever working on ANSYS and think, It would save me a lot of effort if only someone would stop by my office and&

- &help me get this thing to converge
- &help me get results by 3 PM today
- &help me modify this macro
- &write a generic macro-then I can pick it up and tweak it from there
- &teach me how to use this new feature
- &help me check for errors
- &put another set of eyes on this, just to be sure



Many of us have resources like that in our own companies (and maybe you re one of them). Of course, these folks are usually quite popular and often it is hard to get their time. I feel lucky working at an ANSYS reseller, because I always have an expert nearby to help with my own ANSYS analyses.

Daniel (Karate Kid) Larusso: *Yeah, but you knew karate.*
Mr. Miyagi: *Hmph&someone always know more.*

Some customers have enough users that we make regular visits, drop by offices and sit with people. Other customers purchase a chunk of time, like 10 hours, and we use it over several visits, or phone calls as the demand arises. The best benefit of all this is that Enhanced Technical Support not only helps you achieve your goal, it prioritizes *your* learning. We want to make you the expert.



The Focus



A Publication for ANSYS Users

Combining your understanding of your analysis goals, process, and skills with PADT's knowledge of ANSYS, analysis methods, and automation, allows us to realize huge time-savings. Often a 15-minute discussion, explanation, script debugging, macro-writing, etc. will save someone an afternoon of irritation. [Set up this service](#) before you run into trouble then we're only a phone call away!

The Focus



A Publication for ANSYS Users

~Heal Method

by [Rod Scholl](#)

There have been some changes in the relationship between [ANSYS, Inc.](#) and CADfix ([Transcendata](#)). Some results of these changes are:

- *CADfix For ANSYS* is no more.
- Automatic Geometry Healer (AGM) is a new product sold by [ANSYS, Inc.](#)
- CADfix is sold by [Transcendata](#).
- *CADfix for ANSYS* customers will receive licensing for both CADfix and the Automatic Geometry Healer (AGM) through the end of their TECS agreement. Contact your ANSYS ASD for details.



The Automatic Geometry Healer (AGM) is invoked by a single command ~HEAL. There is no menu selection, but should be in ANSYS 7.0. When ~HEAL is issued, ANSYS will import the file and attempt to repair using the AGM. The AGM is based on CADfix for ANSYS technology, but it is all run in batch mode. There is no user interaction.

~HEAL, Name, Extension, Path, Method

The key to the ~HEAL command is in the Method option. You have the choice of either Method=ANSYS or Method=Cadfix. If your geometry is being imported via a connection product (Pro/E, Catia, UG, Para, Sat, Ideas, etc.) then you will use Method=ANSYS. If your file is an .IGES file, you will use Method=Cadfix.

Method=ANSYS

Upon issuing the ~HEAL command with the Method set to ANSYS, the file will be imported using your connection product *for which you need a license*. Then the resulting .anf file is processed by the Automatic Geometry Healer (AGM), *for*

The Focus



A Publication for ANSYS Users

which you need a license. NOTE: You must have CADfix 5.0 installed locally, even though you *don t* need a license for it.

Method=Cadfix

Upon issuing the ~HEAL command with the Method set to Cadfix, the file will be repaired using the Automatic Geometry Healer (AGM), *for which you need a license.* NOTE: You must have CADfix 5.0 installed locally, *for which you need a license!*

I have tested the Method=ANSYS on a Pro/E model. The AGM was unable to remove slivers, sew surfaces, repair poor CAD-tolerancing problems, etc; albeit was a fairly poor model. Perhaps it would also not have been repairable with CADfix for ANSYS. The nice part was that without any options to set, it only takes one try with the AGM to fail! The bad part was that it took several hours for the repair and several more hours to read in the geometry; albeit it was a somewhat complex 3D model. I have not tested the Method=Cadfix option.

Recommendations

If you use a connection product (Pro/E, Catia, UG, Para, Sat, Ideas, etc.), we recommend that you test out your class of CAD-generated geometry using the AGM (~HEAL). If you do not have success, you may want to evaluate a seat of CADfix from TranscenData.

If you do not use a connection product and simply use .IGES, hopefully your geometry comes in clean. If it does not (big surprise), then you will need a CADfix license to repair geometry, regardless of whether you use AGM. If you can take advantage of the automation provided by AGM, compared to the learning curve to operate CADfix alone, you may want to purchase a seat of AGM as well.

The Focus



A Publication for ANSYS Users

About *The Focus*

The Focus is a periodic electronic publication published by PADT, aimed at the general ANSYS user. The goal of the feature articles is to inform users of the capabilities ANSYS offers and to provide useful tips and hints on using these products more effectively. *The Focus* may be freely redistributed in its entirety. For administrative questions, please contact [Rod Scholl](#) at PADT.

The Focus Library

All past issues of *The Focus* are maintained in an online [library](#), which can be searched in a variety of different ways.

Contributor Information

Please don't hesitate to send in a contribution! Articles and information helpful to ANSYS users are very much welcomed and appreciated. We encourage you to send your contributions via e-mail to [Rod Scholl](#).

Subscribe / Unsubscribe

To subscribe to or unsubscribe from *The Focus*, please visit the PADT e-Publication [subscriptions](#) management page.

Legal Disclaimer

Phoenix Analysis and Design Technologies (PADT) makes no representations about the suitability of the information contained in these documents and related graphics for any purpose. All such document and related graphics are provided as is without warranty of any kind and are subject to change without notice. The entire risk arising out of their use remains with the recipient. In no event, including inaccurate information, shall PADT be liable for any direct, consequential, incidental, special, punitive or other damages whatsoever (including without limitation, damages for loss of business information), even if PADT has been advised of the possibility of such damages.

The views expressed in *The Focus* are solely those of PADT and are not necessarily those of ANSYS, Inc.