Calculating Reaction Forces in a Spectrum Analysis in ANSYS Mechanical
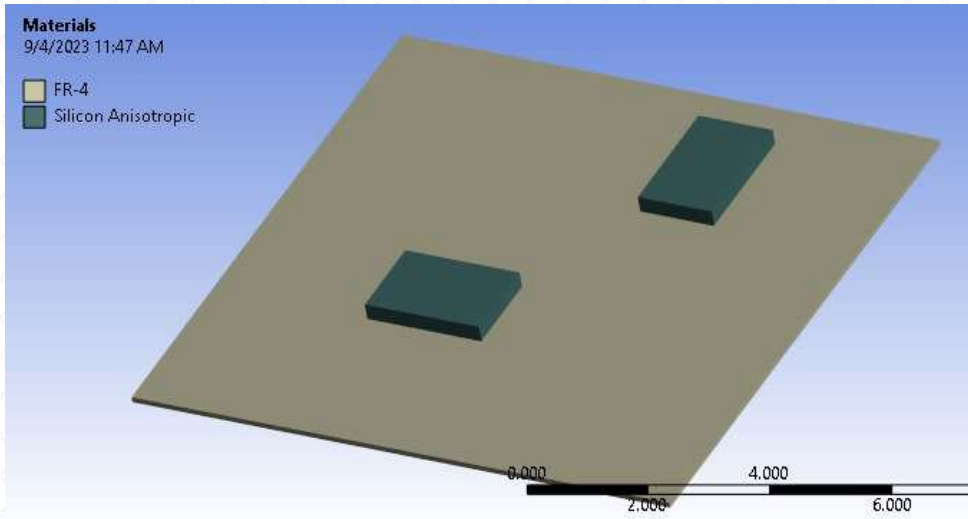
Scripting Solutions for Reaction Forces: Background

- In some physics environments, not all types of entities are selectable for calculating force reactions with the Force Reaction Probe.
- In an old blog post from 2011, we showed ANSYS users how to overcome such limitations using APDL commands in Ansys Mechanical for the case of Random Vibration environments
- But a Spectrum Analysis isn't the same thing as a Random Vibration Analysis (although loosely related, the underlying calculations are different)
- If users try to apply the earlier solution to a Spectrum Analysis, they will get incorrect results. Simply applying a standard APDL 'FSUM' is not enough in these situations
- In this blog post, we show users what's going on and how to retrieve reaction forces in a Spectrum Analysis 'correctly'
- We also provide a script for generating a response spectrum and performing the analysis, as this is a sometimes time-consuming intermediate step for beginners
- Finally, we compare the results of the Spectrum Analysis to those of an equivalent Structural Transient Analysis and show that they are quite close

Test Case

- For this discussion, we'll use a simple example in ANSYS 2023R1 (included with this post)
- The model consists of a single PCB board made of FR-4 material and two anisotropic silicon dies
- The board is fixed at all four corners and subjected to a half-sine shock pulse with a 50G peak acceleration lasting for 11 ms.

10" x 10" x 0.1" PCB

Dynamic Loading

- The test model is subjected to a 50 g half-sine pulse to its "base" (the four corners of the board)
- We'd like to know the maximum force response under the two TTL components when subjecting the entire board to this loading
- There are two ways we can do this in ANSYS. We can perform a structural transient analysis OR we could perform a spectrum analysis
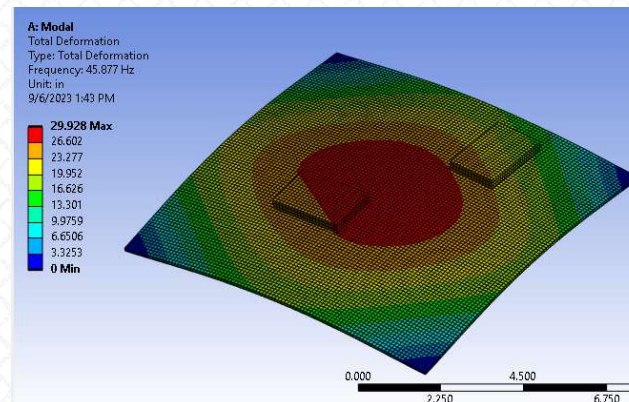

Half-Sine Acceleration (in in/s^2)

Modal Analysis

- Ultimately, we want to show users what they need to do to get accurate reaction force data in a spectrum anaysis in ANSYS Ansys Mechanical
- As a benchmark for accuracy, we'll compare those reaction forces to those of a transient structural analysis
- If we compared the spectrum results to a full transient analysis, we'd need a lot of modes for accuracy –since that's beyond the scope of this article, we'll instead compare the spectrum results to a mode-superposition trasient anlaysis (while ignoring the important question of how many modes we need to calculate for an 'accurate' solution)
- So, our first step is to perform a modal analysis. Since the half-sine pulse has a half-frequency of 1/0.011/2 = 45.45 Hz, we want to extract modes which cover this range
- In this example, we're extracting 10 modes*

Mode	✔ Frequency [Hz]
1.	45.877
2.	106.22
3.	114.73
4.	150.26
5.	256.63
6.	290.21
7.	336.93
8.	358.81
9.	477.17
10.	502.69

A: Modal
Total Deformation
Type: Total Deformation
Frequency: 45.877 Hz
Unit: in
9/6/2023 1:43 PM

29.928 Max
26.602
23.277
19.952
16.626
13.301
9.9759
6.6506
3.3253
0 Min

0.000   4.500
2.250   6.750

*Mode 1 is 45.88 Hz ≈ 45.45 Hz. As an interesing exercise, users should reduce the number of extracted modes in this example and test the accuracy again

Transient Analysis

- Next, we perform a mode-superposition-based transient structural anlaysis.
- This is generally the most costly type of structural dynamic analysis, but should also be the most accurate, and so this solution will provide a good comparison
- We'll choose a global damping ratio of 0.05 (Q factor of 10) for this study
- We'll also follow the documentation guidelines for choosing the numerical timestep (0.011/20)

Transient (C5)
- Initial Conditions
  - Modal (Modal)
- Analysis Settings
- Acceleration
- Solution (C6)
  - Solution Information
  - Directional Deformation
  - Force Reaction

Details of "Modal (Modal)"

Definition	
Modal Environment	Modal
Pre-Stress Environment	None

17.9.1. Time Step Size

The time step size (Δt) is selected in the following way. If data is from a full transient analysis (**ANTYPE**,TRANS with **TRNOPT**,FULL):

Δt = input time step size (input as *DTIME* on **RESP** command)

or if no input is provided:

$$\Delta t = \frac{1}{20 f_{max}} \qquad (17–128)$$

| Damping Controls	
Eqv. Damping Ratio From Modal	No
☐ Damping Ratio	5.e-002
Stiffness Coefficient Define By	Direct Input
☐ Stiffness Coefficient	0.
☐ Mass Coefficient	0.
Numerical Damping	Program Controlled
Numerical Damping Value	0.005

Transient Analysis

- Define two load steps. We want load step 1 to cover the half-sine pulse, and load step 2 to carry on to 0.022s (2 ms) with no loading (to roughly correspond to one full cycle of the fundamental natural frequency)
- We will be looking for maximum overall response (absolute magnitude) of z-component quantities, and we expect such extrema to occur within this first period
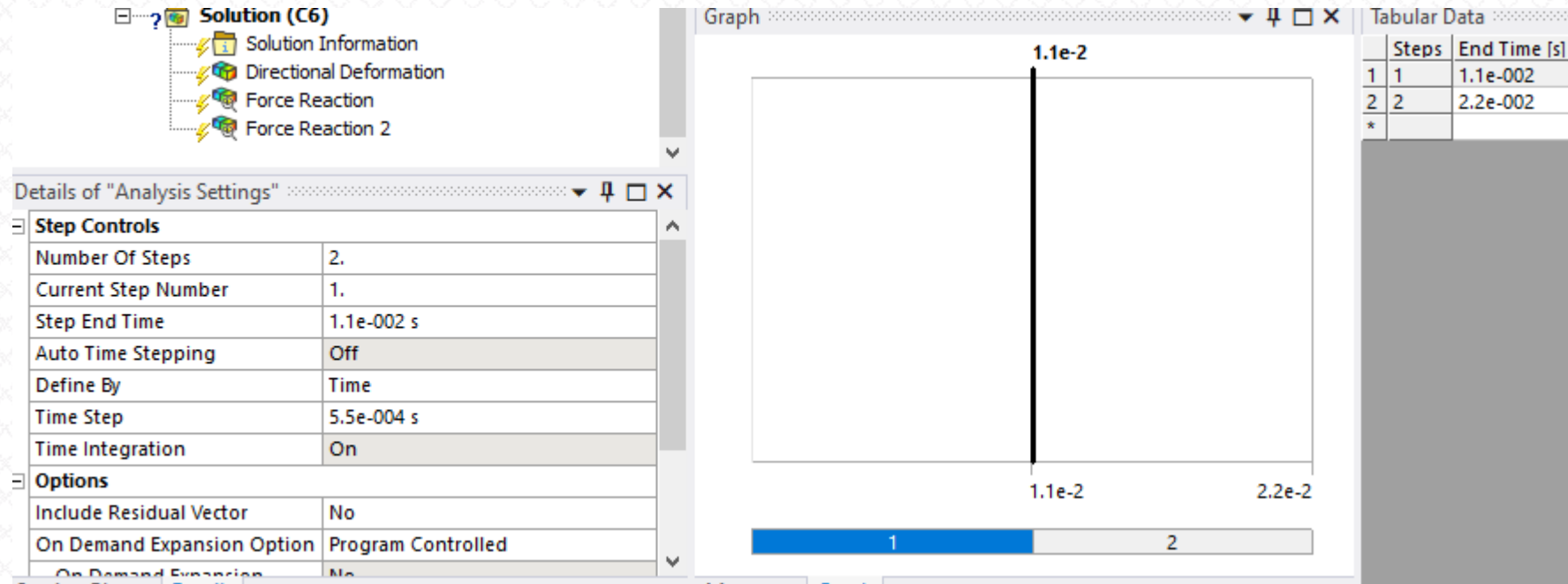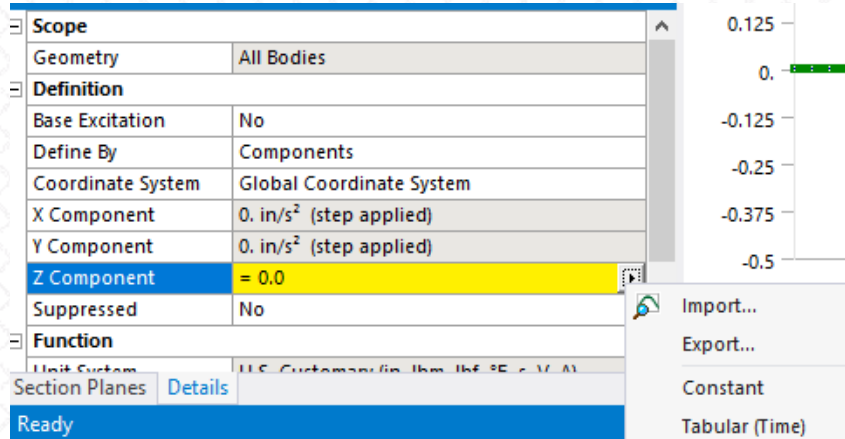
Transient Analysis

- Define the half-sine pulse as an acceleration load using the 'function' option
- But first, define angle units of radians. Degrees are the default units, and will also work fine, but be aware that the angle units on the function control will not change once it has been selected
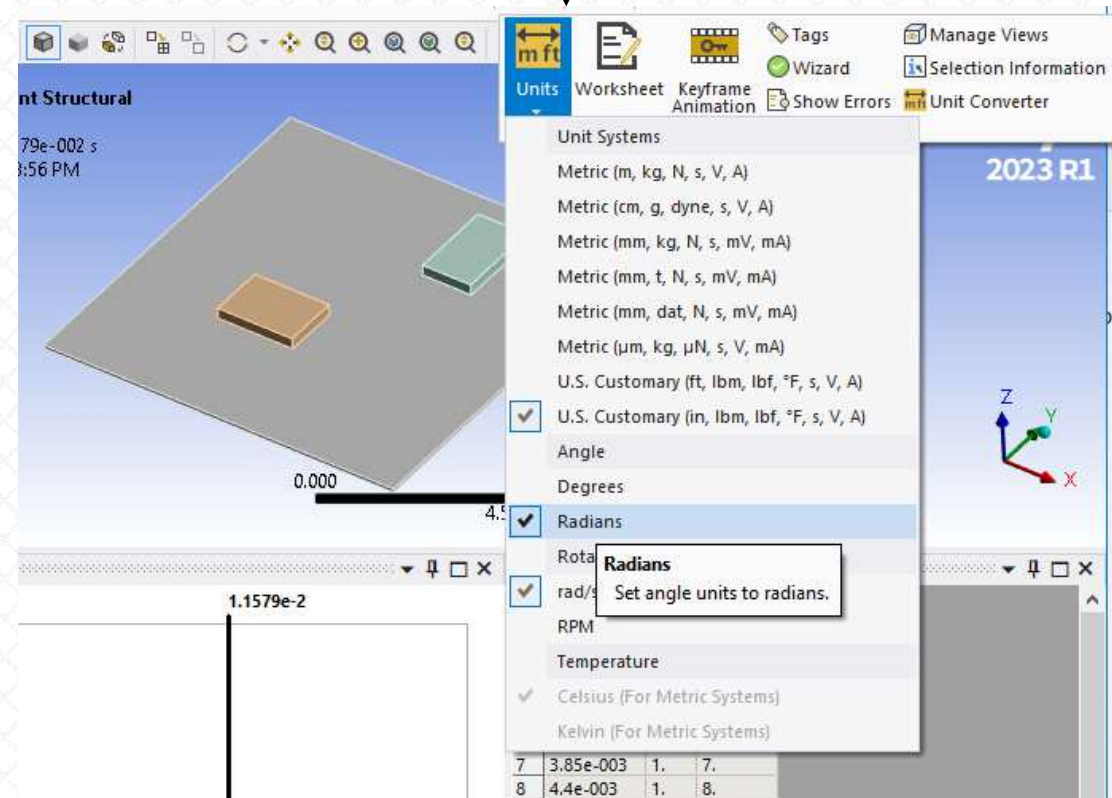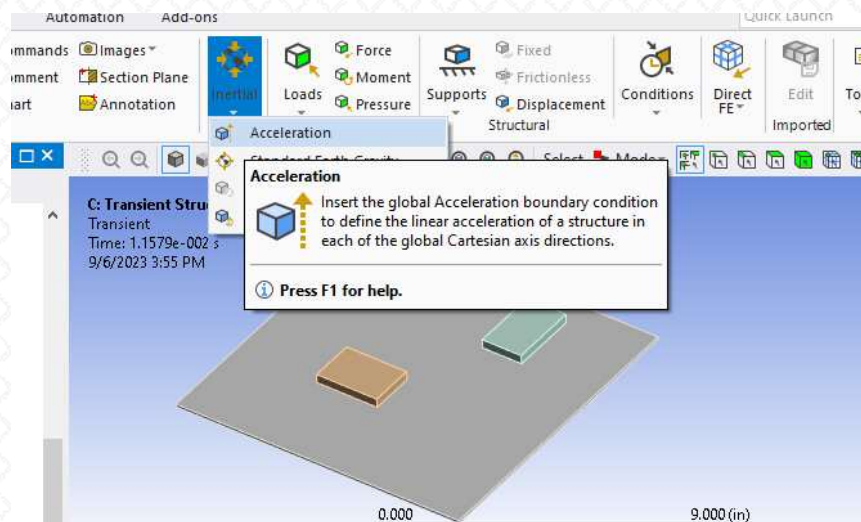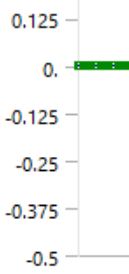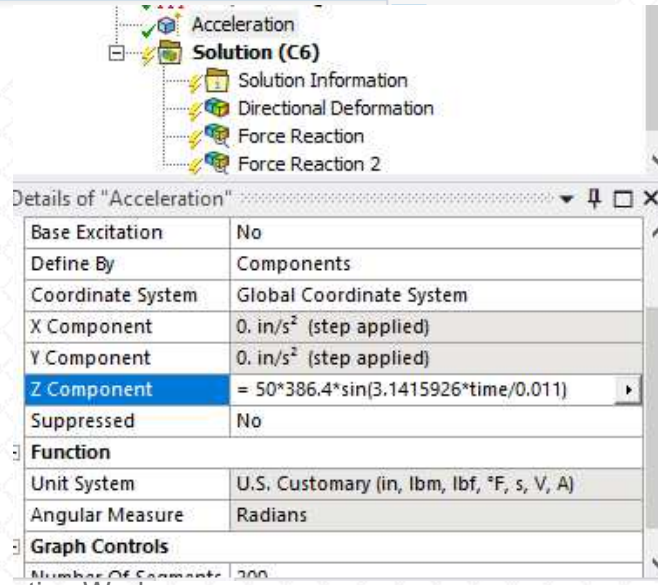
Transient Analysis

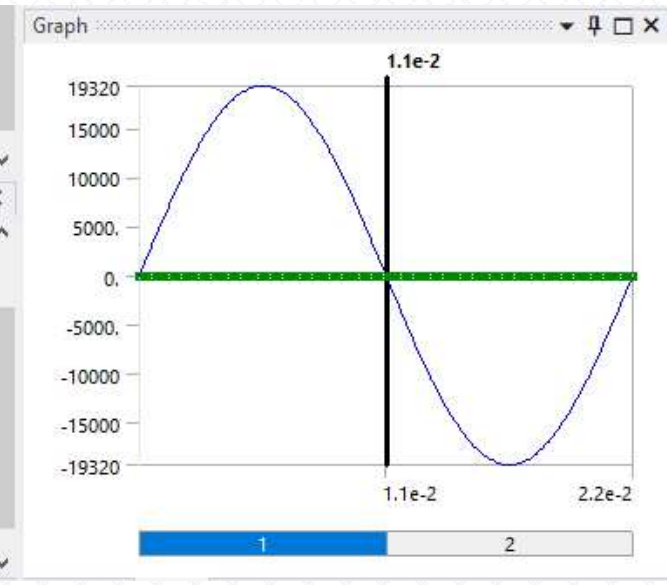- Define the half-sine pulse as an acceleration load using the 'function' option in the z-direction

- for angle units of radians, use the equation: 50*386.4*sin(3.141526*time/0.011)
- for angle units of degrees, use the equation: 50*386.4*sin(180*time/0.011)

Transient Analysis

- Deactivate all loading at load step 2 (select the cells shown below in "Tabular Data", right-click->Activate/Deactivate this step!)
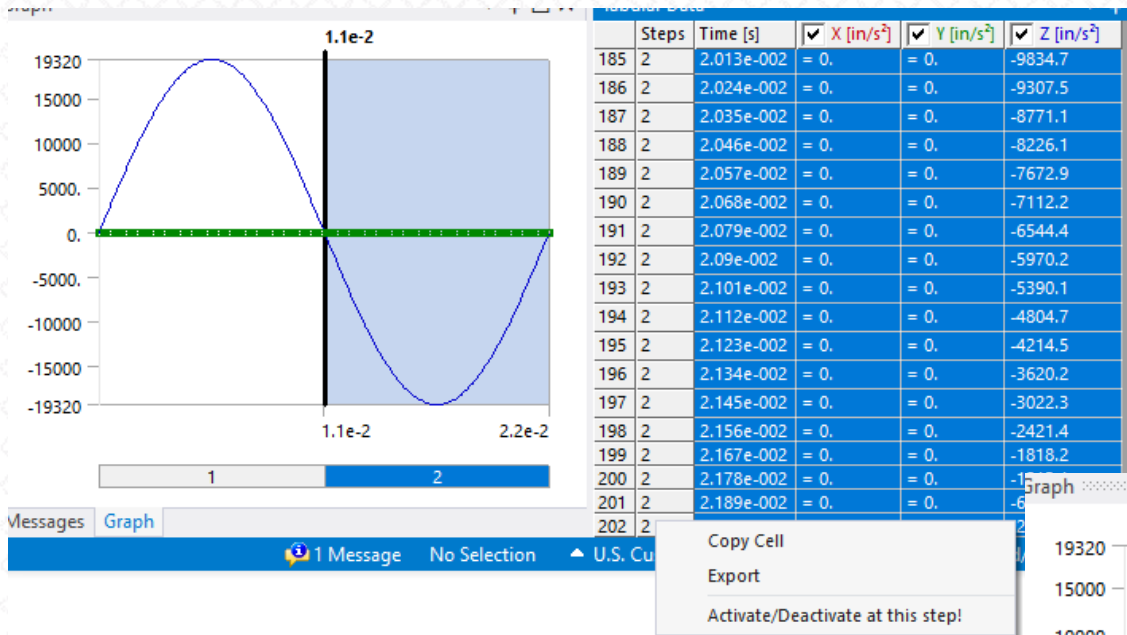
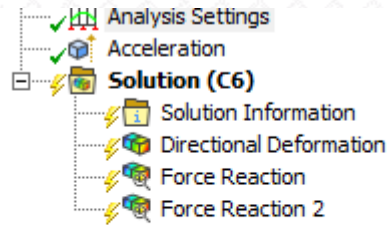- if successful, you should now have a half-sine pulsed acceleration over the interval 0≤t≤0.011s, while the solution end time t = 0.022s
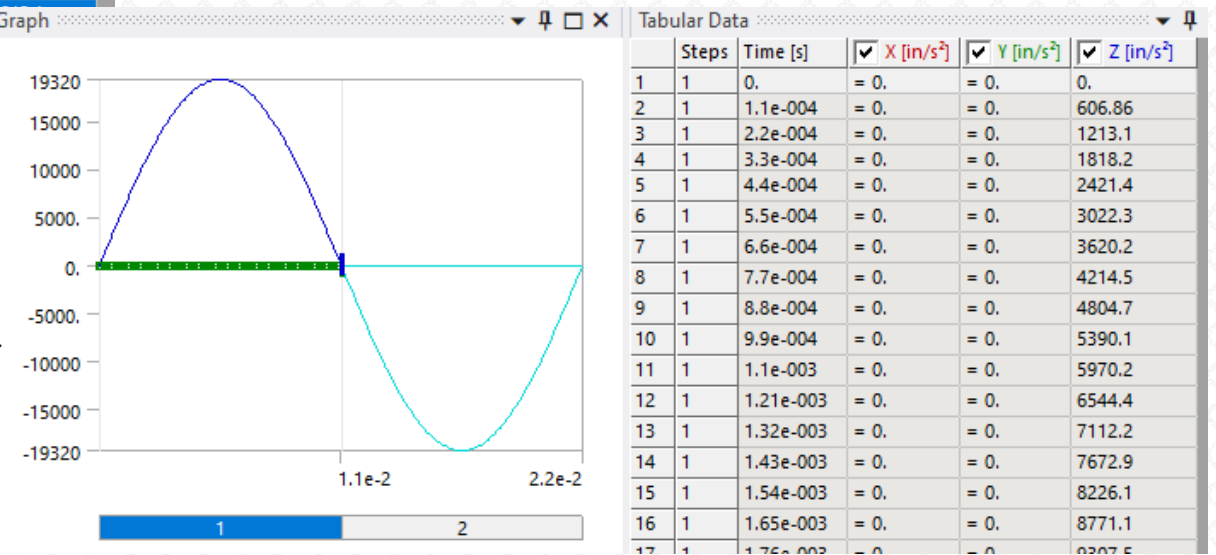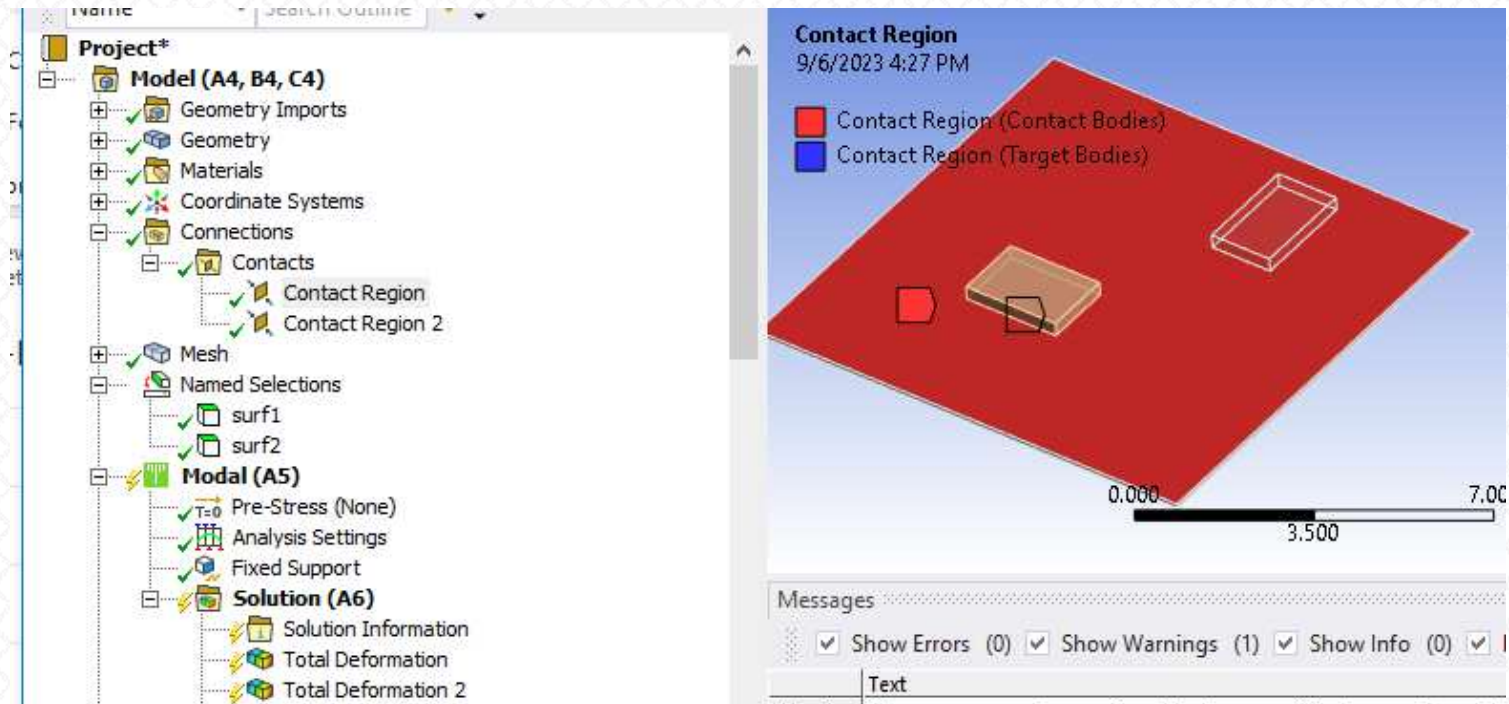
Transient Analysis

- With Analysis Settings->Output Controls->Contact Data and Nodal Forces set to "Yes", insert a force reaction probe for each of the contact regions shown below (we want to track the reaction forces at the TTL component interfaces)

Transient Analysis

- Run the anlaysis. Since this is a mode-superposition-based solution with only 10 modes, it should run quite quickly (in under a minute on most laptops)
- Once complete, you should see the overall displacement solution shown below
- Note that the largest absolute value response occurs at the end of the pulse (0.011s) as one would expect (lower right), but it occurs as a minimum (negative) displacement
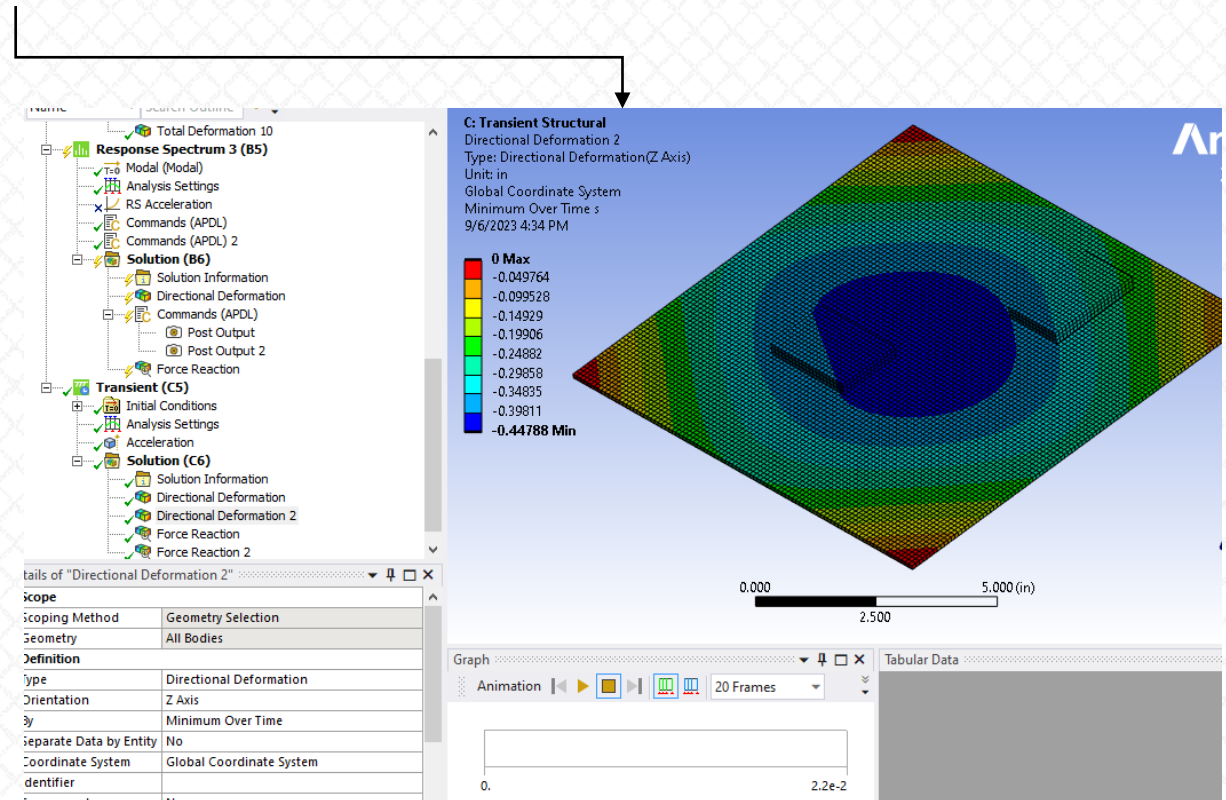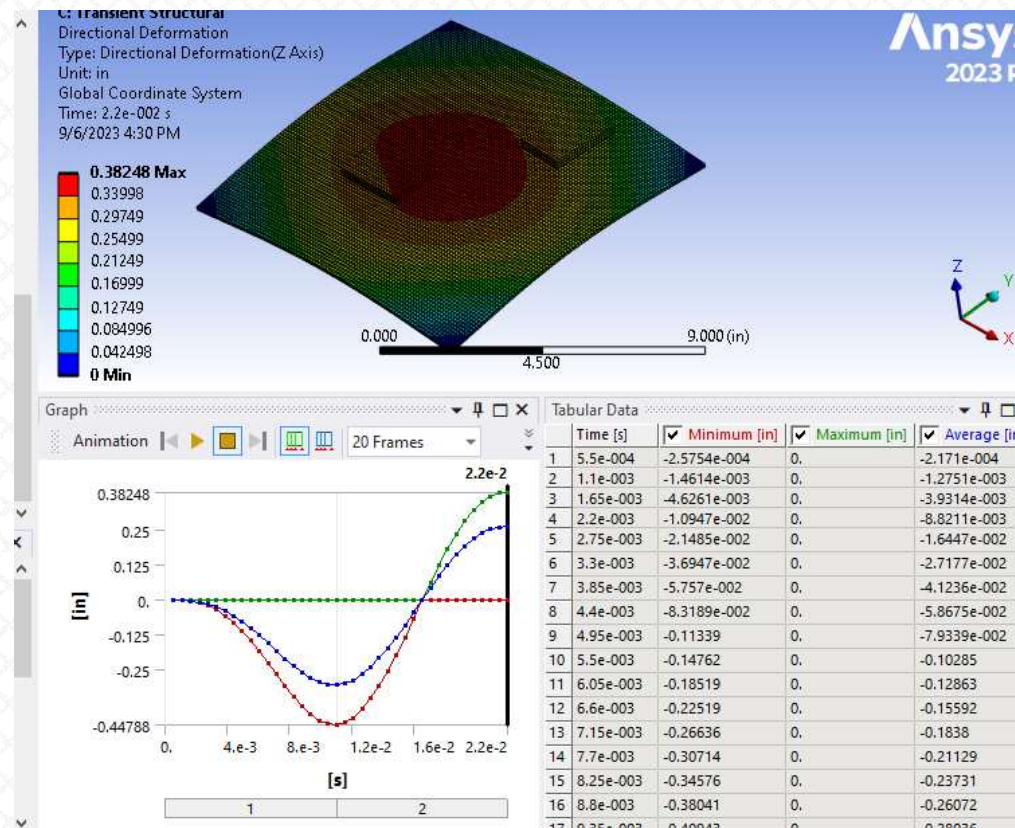
Transient Analysis

- Review the two Force Reaction probe results. Recall that these are the results we'd like to compare in a spectrum analysis (we're interesed in the maximum absolute values of the z-component.
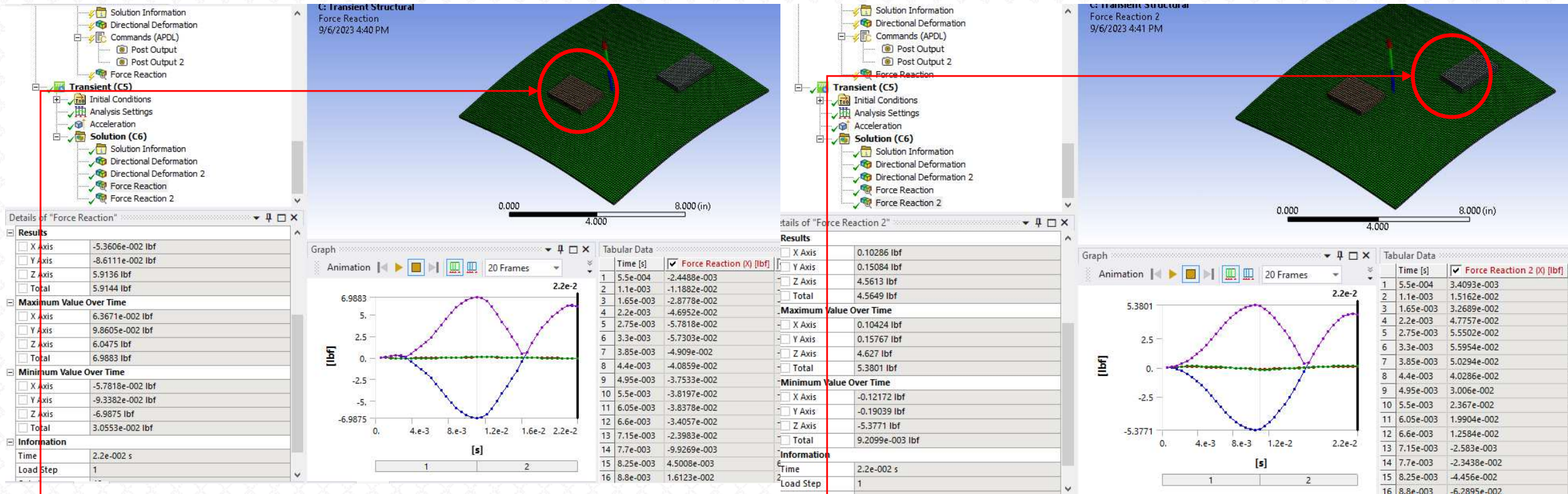
$|F_{zmax}|$ = 6.9875 lbf @ Contact Region 1 (0.011s)

$|F_{zmax}|$ = 5.3771 lbf @ Contact Region 2 (0.011s)

Spectrum Analysis

- We'd now like to perform an equivalent spectrum analysis of this model and compare the TTL component interface (z-component) reaction loads to those of the structural transient analysis.
- This article does not attempt to cover the Response Spectrum Method* in detail, but we'll summarize some highlights.
- First, it is a modal techique to estimate the maximum overall response (the nuances of this definition may all be treated within the technique) of a structure to a transient loading event of finite duration (a "shock")
- One result is produced for all model degrees of freedom, and this result is meant to convey the maximum overall response of the structure over all time.
- The input to this type of analysis is a Shock Response Spectrum (SRS) –not the time-history signal itself. Therefore, SRS generation is an intermediate step
- Excellent primers on this technique, and SRS generation may be found here, here, here, and here
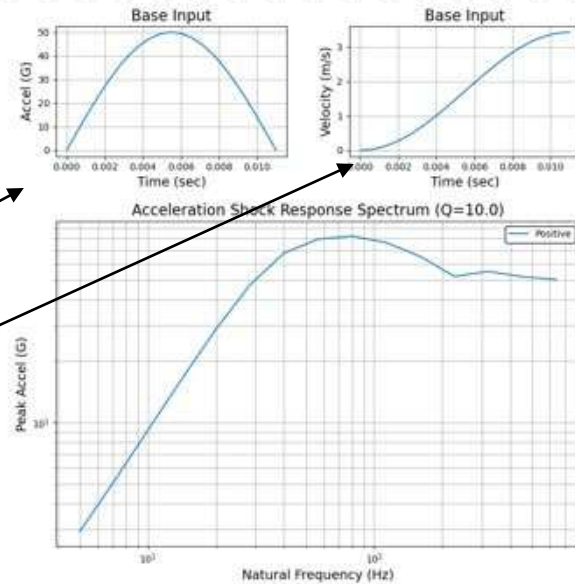
*this technique is unfortunately named, as it has little if anything to do with 'spectral analysis'. However, googling 'Response Spectrum Method' will usually produce the desired results

Spectrum Analysis

- There is no shortage of software for generating SRS curves from time-history signals
- Although MATLAB doesn't appear to have any built-in functions for doing this, users have written code freely available on MathWorks File Exchange (for example, see here*)
- At PADT, we've been quite content with a Python version of this code, which users may freely download here (this is all open source code, but be sure to read the license information)
- After appyling the above code to our signal (50 G half-sine lasting 11 ms), we obtain the spectrum shown below

- acceleration time-history
- velocity time-history

Freq (Hz)	Accel (in/s^2)
5.00	1129.33
7.07	1987.15
10.00	3546.30
14.14	6336.88
20.00	11094.32
28.28	18290.59
40.00	26204.87
56.57	30641.23
80.00	31760.09
113.14	29480.61
160.00	25211.04
226.27	20140.94
320.00	21284.62
452.55	20060.52
640.00	19438.84

- We've saved this spectrum in the Ansys Mechanical example model accompanying this article

*all of the free code we're linking to appears to be based on David O Smallwood's An Improved Recursive Formula For Calculating Shock Response Spectra
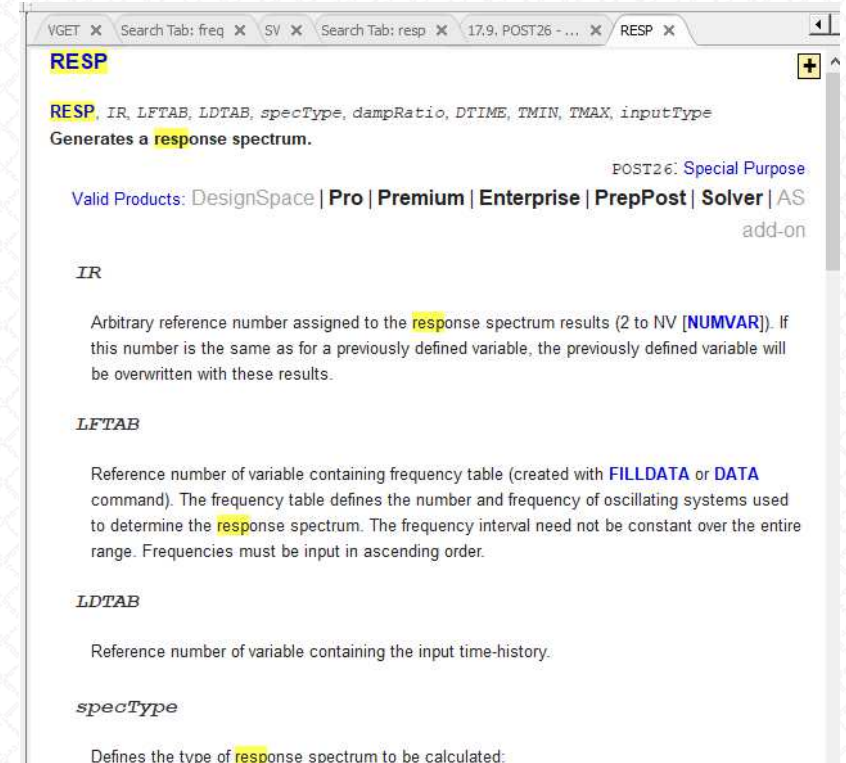
Spectrum Analysis

- Perhaps less widely known today is the fact that ANSYS offers its own functionality for generating shock spectra
- This is an APDL command called 'RESP'. And in the spirit of this blog, we're providing this solution to users of ANSYS as well (for more, search for 'RESP' in the documentation. A screenshot is shown below right)
- Because it's an APDL command, it is possible to simply embed a command object which invokes this directly within a Ansys Mechanical model, thus eliminating an intermediate step (generating the SRS)
- This is what we'll do next. Before introducing the code, we'd like to extend our thanks to Dave Looman of ANSYS, Inc., who first demonstrated the usage of this command to the world. Our script is based on his.
- For his latest iteration, see here (screenshot below)

How to create a shock response spectrum from time history data with APDL commands

REPLY   👍 0   September 16, 2020 at 5:28 pm

Dave Looman
Ansys Employee

In the input below the POST26 RESP command is used to compute the shock response spectrum for a 20g triangular acceleration of 10 milli-sec duration. The time-history includes time after the shock load for the lower frequencies to reach their peak displacement value. The input assumes that a modal analysis has been done and expanded.n! After a modal analysis...npi=acos(-1)namplitude=20 ! pulse amplitude (20 g)nwidth=0.01 ! pulse width (10 milli-secs)n n*dim,acel,table,4,1nacel(1,0)=0.0, 0.5*width, width, 5.0*widthnacel(1,1)=0.0, amplitude, 0.0, 0.0n n/axlab,x,TIME n/axlab,y,ACEL (g)n*vplot,acel(1,0),acel(1,1) ! verify acceleration inputnThe table array, acel, can now be used as input to the RESP command in the time-history post-processor.n/post26nstore,alloc,10 ! allocate the number of frequencies to be solved forn-vput,acel(1,0),1,0 ! variable 1 is time valuesnvput,acel(1,1),2,0 ! variable 2 is acceleration values-ndata,5,1,10,1,Frequncy ! variable 5 is frequency list (I love this old command!) n(10F5.1)n10.0 20.0 30.0 40.0 50.0 60.0 70.0 80.0 90.0 100.0n n! Use parameters for RESP command input to make more readablenSpec=6 ! variable to contain spectrum outputnFreq=5 ! variable with frequencies to be solved fornInp_Acel=2 ! variable with acceleration inputnOut_Type=3 ! create acceleration spectrumnDamp=0.05 ! damping (ratio to critical) to be usednTinc=0.0005 ! integration time step, 20 pts per cycle at 100 hznInp_Type=1 ! acceleration input, new at 14.0n nresp,Spec,Freq,Inp_Acel,Out_Type,Damp,Tinc,,,Inp_Type n n/title,Response Spectrum for %amplitude%g, %width*1000% m-sec shock load (5% Damping)n/axlab,x,Frequency (Hz)n/axlab,y,Peak Acceleration (g)nxvar,Freq ! x axis is frequencynplvar,Spec ! plot acceleration spectrumn n

RESP

RESP, IR, LFTAB, LDTAB, specType, dampRatio, DTIME, TMIN, TMAX, inputType
Generates a response spectrum.

POST26: Special Purpose

Valid Products: DesignSpace | Pro | Premium | Enterprise | PrepPost | Solver | AS
add-on

IR

Arbitrary reference number assigned to the response spectrum results (2 to NV [NUMVAR]). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with these results.

LFTAB

Reference number of variable containing frequency table (created with FILLDATA or DATA command). The frequency table defines the number and frequency of oscillating systems used to determine the response spectrum. The frequency interval need not be constant over the entire range. Frequencies must be input in ascending order.
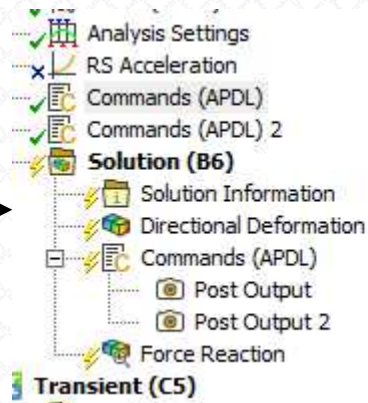
LDTAB

Reference number of variable containing the input time-history.

specType

Defines the type of response spectrum to be calculated:

Spectrum Analysis

- In our example, we've input the acceleration SRS generated in Python (see slide 15), but this is suppressed for the time being. We'll use this purely for comparison purposes later
- We've inserted three command objects. The first re-executes the 'srss' command (specifying the spectrum combination method). The reason for this is that the default behavior of this command is to sum the static components of the total dynamic forces only. We want instead to compare the calculated total dynamic reaction force (including the inertial component) to that calculated in the transient structural solution earlier.
- The second calculates the SRS from the half-sine pulse, writes the spectrum to a file in the solution folder, and performs the spectrum analysis

Spectrum Analysis

- The third command object is the post-processing macro to get the reaction forces at the TTL component interfaces
- For this purpose, we've made named selections for the bottom surfaces of each, called 'surf1' and 'surf2'

Spectrum Analysis

- Readers of the Focus blog will be familiar with use of the FSUM commnad to retrieve otherwise unavailable reaction forces in Ansys
- What's new here, however, is the use of the 'mcom' file.
- This MUST be input prior to the calculation of reaction forces
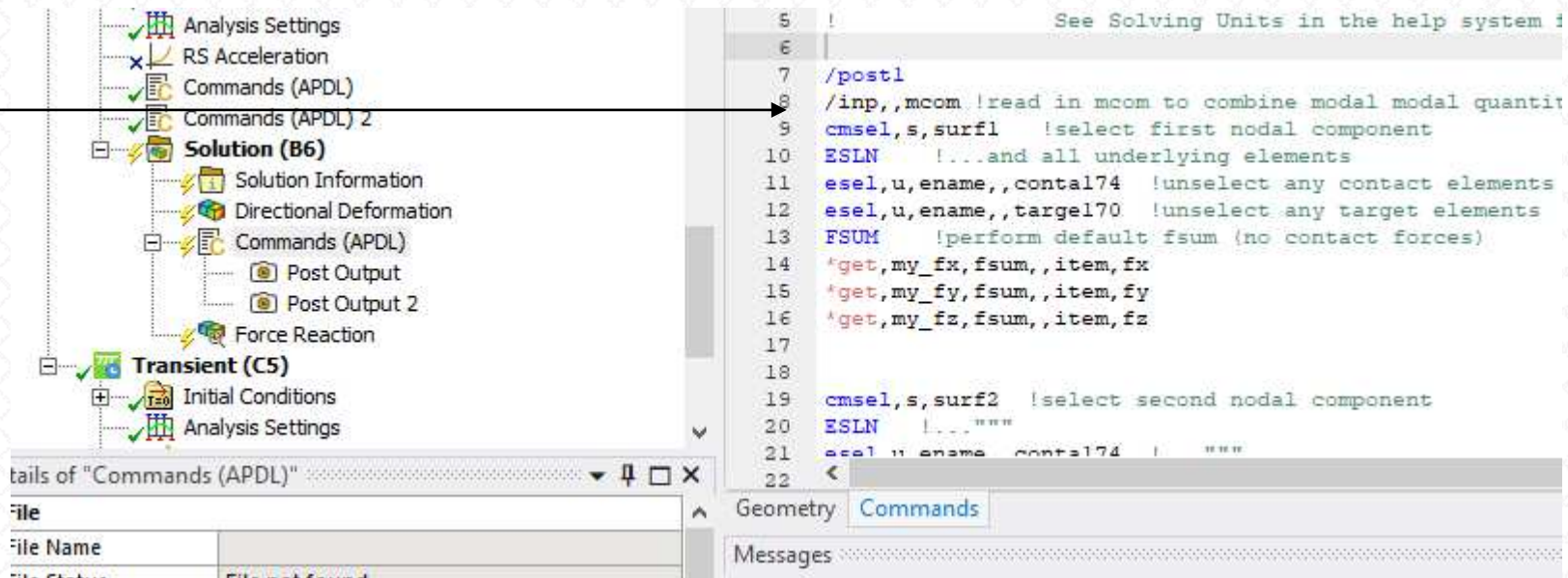
Spectrum Analysis

- The mcom file contains the APDL load case operations which perform the load combinations defined in the spectrum analysis (in this case, the SRSS load combinations)

```
1    /COM,ANSYS RELEASE 2023 R1         BUILD 23.1       UP20221128        11:00:58
2    /COM, file.mcom
3    LCOPER,ZERO
4    FORCE,TOTAL       ! Combine modal static plus inertial forces
5    FILE, 'D:\techsupport\adam_fletcher\spectrum_2021R1_files\dp0\SYS\MECH\file', rst
6    LCDEFI,1,        1,        1
7    LCFACT,1,   0.146776E-01
8    LCASE,1
9    LCOPER,SQUARE
10   LCDEFI,1,        1,        2
11   LCFACT,1,   0.620824E-04
12   LCOPER,ADD,1,MULT,1
13   LCDEFI,1,        1,        3
14   LCFACT,1,  -0.521098E-04
15   LCOPER,ADD,1,MULT,1
16   LCDEFI,1,        1,        4
17   LCFACT,1,   0.504641E-04
18   LCOPER,ADD,1,MULT,1
19   LCDEFI,1,        1,        5
20   LCFACT,1,   0.211288E-04
21   LCOPER,ADD,1,MULT,1
22   LCDEFI,1,        1,        6
23   LCFACT,1,   0.609118E-04
24   LCOPER,ADD,1,MULT,1
25   LCDEFI,1,        1,        7
26   LCFACT,1,   0.277481E-05
27   LCOPER,ADD,1,MULT,1
28   LCDEFI,1,        1,        8
29   LCFACT,1,   0.389557E-05
30   LCOPER,ADD,1,MULT,1
31   LCDEFI,1,        1,        9
32   LCFACT,1,   0.234025E-05
33   LCOPER,ADD,1,MULT,1
34   LCDEFI,1,        1,       10
35   LCFACT,1,  -0.238067E-05
36   LCOPER,ADD,1,MULT,1
37   LCOPER,SQRT
```

- The mcom file multiplies each eigenvector by it's mode coefficient, squares it, sums them all up, and then takes the square root

$$\{r_i\} = \omega_i^m A_i \{\phi_i\} \qquad (15\text{--}175)$$

15.7.6.4. Square Root of the Sum of the Squares (SRSS) Method

The SRSS method, accessed with the **SRSS** command, is taken from the NRC Regulatory Guide([41]). For this case, Equation 15–180 reduces to:

$$R_a = \left(\sum_{i=1}^{N} (R_i)^2 \right)^{\frac{1}{2}} \qquad (15\text{--}186)$$
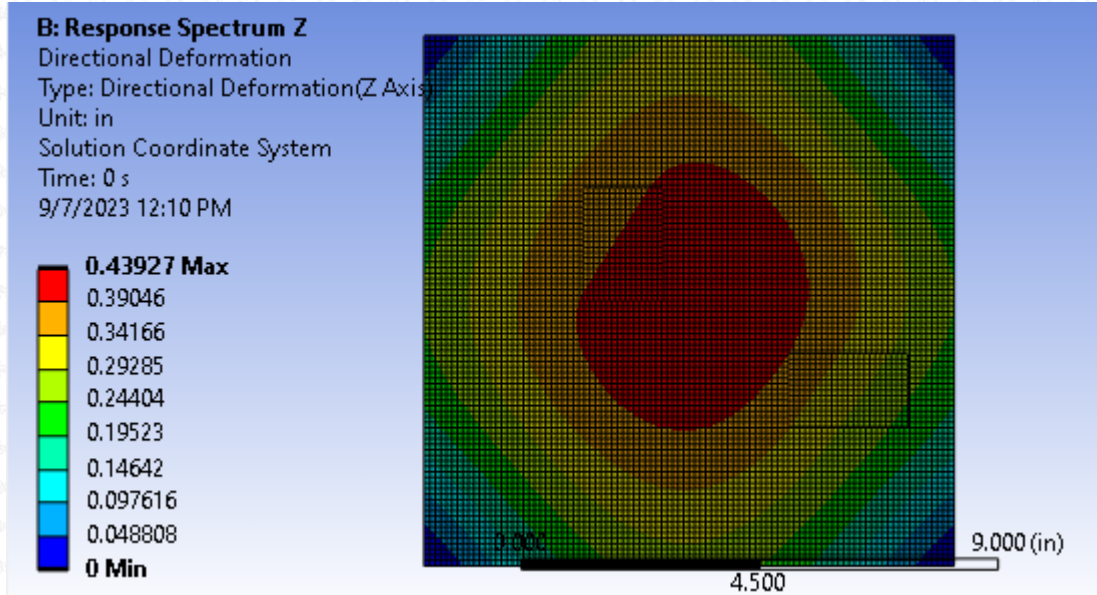
Spectrum Analysis

- Now, ANSYS automatically does this in a spectrum analysis (as you can verify by looking at the end of the ds.dat input file as shown here). But notice that it's only storing reaction forces for one 'compnames' variable. This corresponds to the fixed support. This ensures that we'll get the correct reaction forces for the fixed support, but only for that entity
- To clarify: most result quantities WILL get stored correctly (displacement, for example), but only the reaction force for the fixed support will be correct. Any other FSUM's executed downstream will be incorrect unless the mcom file is read again
- This means that we are actually running the mcom file twice (!)
- Once for stored quantities and the support reactions, and again for any additional 'FSUM's we may want to execute

```
solve
freq
fini
*list,,mcom
/copy,file,mcom,,Displacement,mcom
/post1
/inp,,mcom
/nopr
*dim,_compnames,string,128,1
*dim,_centroids,array,1,3
_compnames(1,1) = '_CM52_FIXEDSU'
_centroids(1,1) = 5
_centroids(1,2) = 5
_centroids(1,3) = 0.05
*do,iIndex,1,1
        allsel,all
        cmsel,s,_compnames(1,iIndex)
        spoint,,_centroids(iIndex,1), _centroids(iIndex,2),_centroids
(iIndex,3)
        fsum
        *get,reaction_%iIndex%_1,fsum,0,item,fx
        *get,reaction_%iIndex%_2,fsum,0,item,fy
        *get,reaction_%iIndex%_3,fsum,0,item,fz
        *get,reaction_%iIndex%_4,fsum,0,item,mx
        *get,reaction_%iIndex%_5,fsum,0,item,my
        *get,reaction_%iIndex%_6,fsum,0,item,mz
*enddo
spoint,,0.,0.,0.
allsel,all
```

- This corresponds to the fixed support

Spectrum Analysis

- Here are the results. Compare them to the transient results on slides 12 and 13.
- These results are about as close as one could expect from two different approximate numerical techniques
- Keep in mind that, in general, the transient results should be slightly more accurate.

$|F_{zmax}| = 6.909$ lbf @ Contact Region 1

$|F_{zmax}| = 5.209$ lbf @ Contact Region 2

Spectrum Analysis

- Finally, to compare spectrum results with macro-generated SRS to those with the spectrum calculated with Python (see slide 15), suppress the second macro (see below) and unsuppress the spectrum load in the tree outline

|F

|F

Spectrum Analysis

- And here are those results. The two spectra differ slightly (the Python spectum was calculated at different frequencies)

$|F_{zmax}|$ = 7.00 lbf @ Contact Region 1

$|F_{zmax}|$ = 5.284 lbf @ Contact Region 2

Spectrum Analysis

- Finally, open the Solver Files Directory and review ANSYS-generated SRS (in file "spectrum.csv")
- Compare it to the one we calculated with Python (slide 15)...
- The differences come mainly from the fact that the two curves are taken at different frequencies
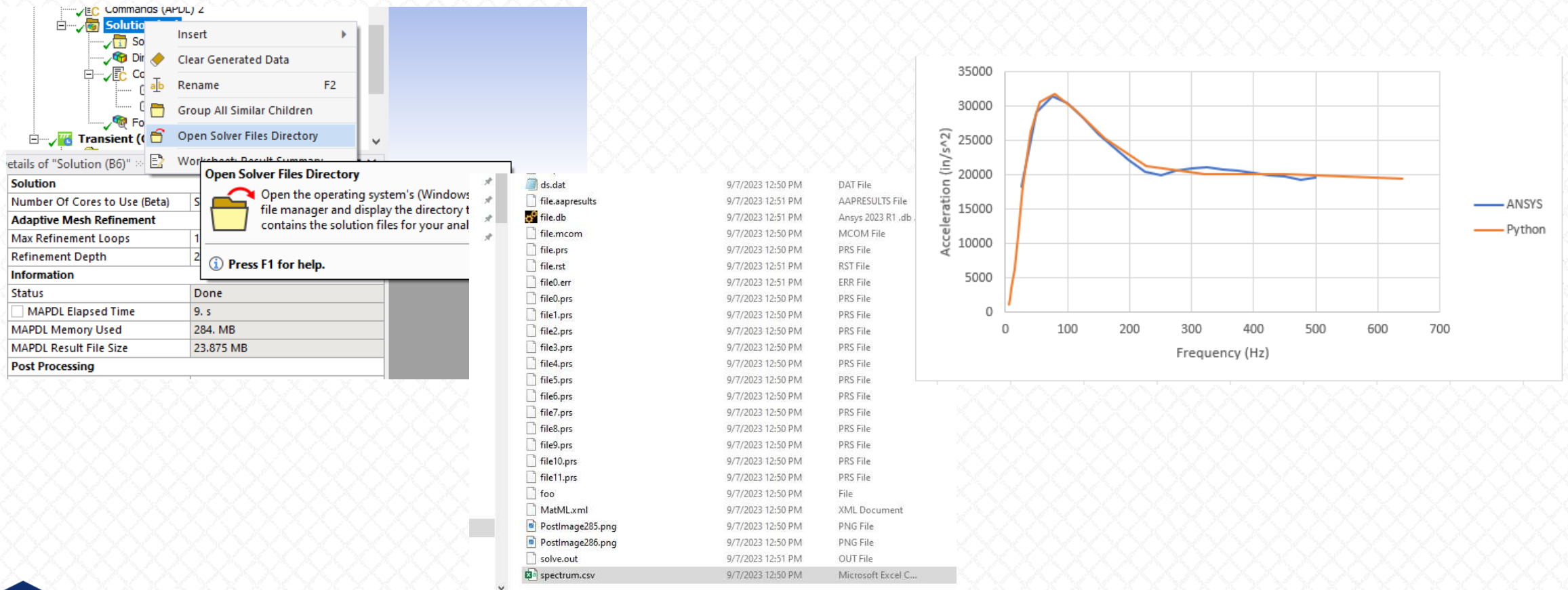
Conclusions

- The Response Spectrum Method of calculating structural responses to shock loads is often confusing to novice ANSYS users
- Even users who understand the procedure well are often suspicious of the result accuracy
- In this article, we demonsrate that this method produces responses which compare very well to models with a single, well-defined damping (results will, of course, be different for models with more complicated damping)
- We also show users how to overcome the shortcomings of the Force Reaction Probe (which will only provide results for boundary conditions)
- Finally, we provide a scipt in the form of a Mechanical Commands object to automate the generation of an SRS from a time-history signal and perform the response spectrum analysis in Mechanical