

Calculating Reaction Forces in a Spectrum Analysis in ANSYS Mechanical



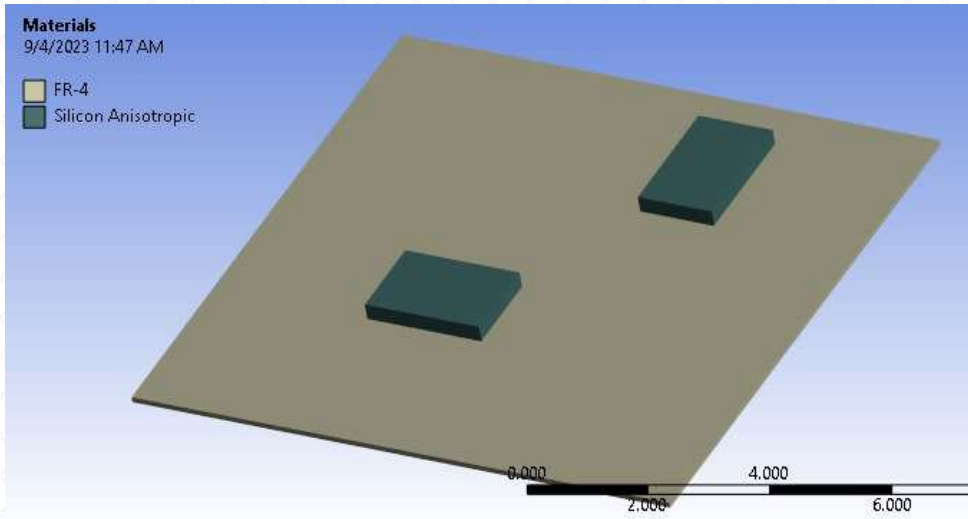
Scripting Solutions for Reaction Forces: Background

- 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

Total mass of model ~ 0.8 lbm

Properties	
Volume	11.884 in ³
Mass	0.82336 lbm
Scale Factor Value	1.

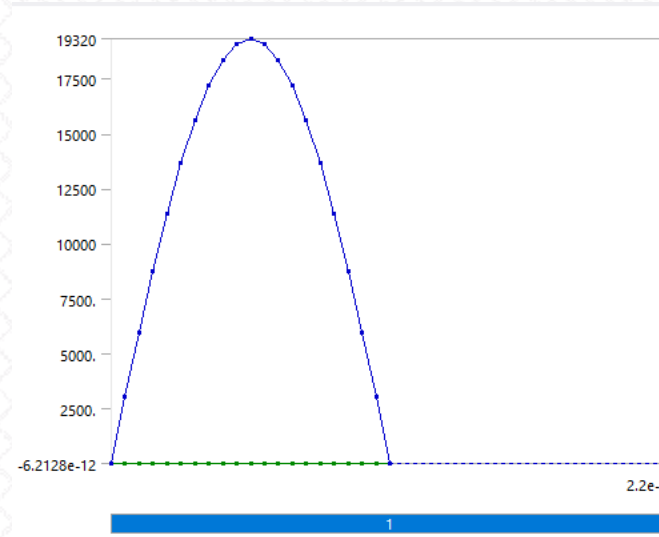
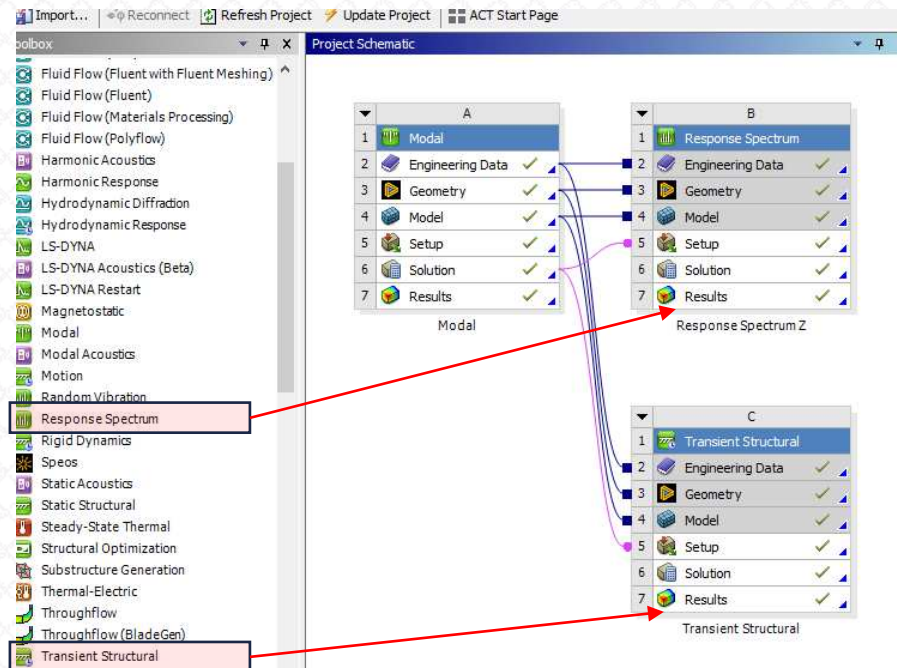
The image shows the ANSYS software interface. On the left is the Project Outline tree, listing Model (A4, B4), Modal (A5), Response Spectrum (B5), and Solution (B6). The main window displays a 3D model of the PCB with four corners fixed, indicated by black lines and the text "four edges fixed (load application)". A blue arrow points upwards from the center of the board, labeled "Spectrum Acceleration in Z". A scale bar at the bottom of the model shows 0.000, 3.000, and 6.000 (in). Below the model is a graph of the response spectrum, showing a half-sine pulse. A table of data is also visible.

Frequency (Hz)	Acceleration (g)
1	5
2	7
3	10
4	14
5	20
6	28



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



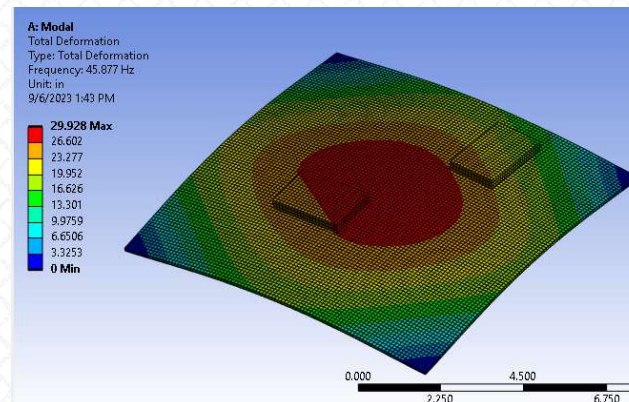
Steps	Time [s]	X [in/s ²]	Y [in/s ²]	Z [in/s ²]
1	0.	0.	0.	0.
2	5.5e-004	0.	0.	3022.3
3	1.1e-003	0.	0.	5970.2
4	1.65e-003	0.	0.	8771.1
5	2.2e-003	0.	0.	11356
6	2.75e-003	0.	0.	13661
7	3.3e-003	0.	0.	15630
8	3.85e-003	0.	0.	17214
9	4.4e-003	0.	0.	18374
10	4.95e-003	0.	0.	19082
11	5.5e-003	0.	0.	19320
12	6.05e-003	0.	0.	19082
13	6.6e-003	0.	0.	18374
14	7.15e-003	0.	0.	17214
15	7.7e-003	0.	0.	15630
16	8.25e-003	0.	0.	13661
17	8.8e-003	0.	0.	11356
18	9.35e-003	0.	0.	8771.1
19	9.9e-003	0.	0.	5970.2
20	1.045e-002	0.	0.	3022.3
21	1.1e-002	0.	0.	-6.2128e-012
22	2.2e-002	= 0.	= 0.	= -6.2128e-012

Half-Sine Acceleration (in in/s²)

Modal Analysis

- Ultimately, we want to show users what they need to do to get accurate reaction force data in a spectrum analysis 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 transient analysis (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	<input checked="" type="checkbox"/> 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



*Mode 1 is 45.88 Hz \approx 45.45 Hz. As an interesting 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 analysis.
- 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)

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)

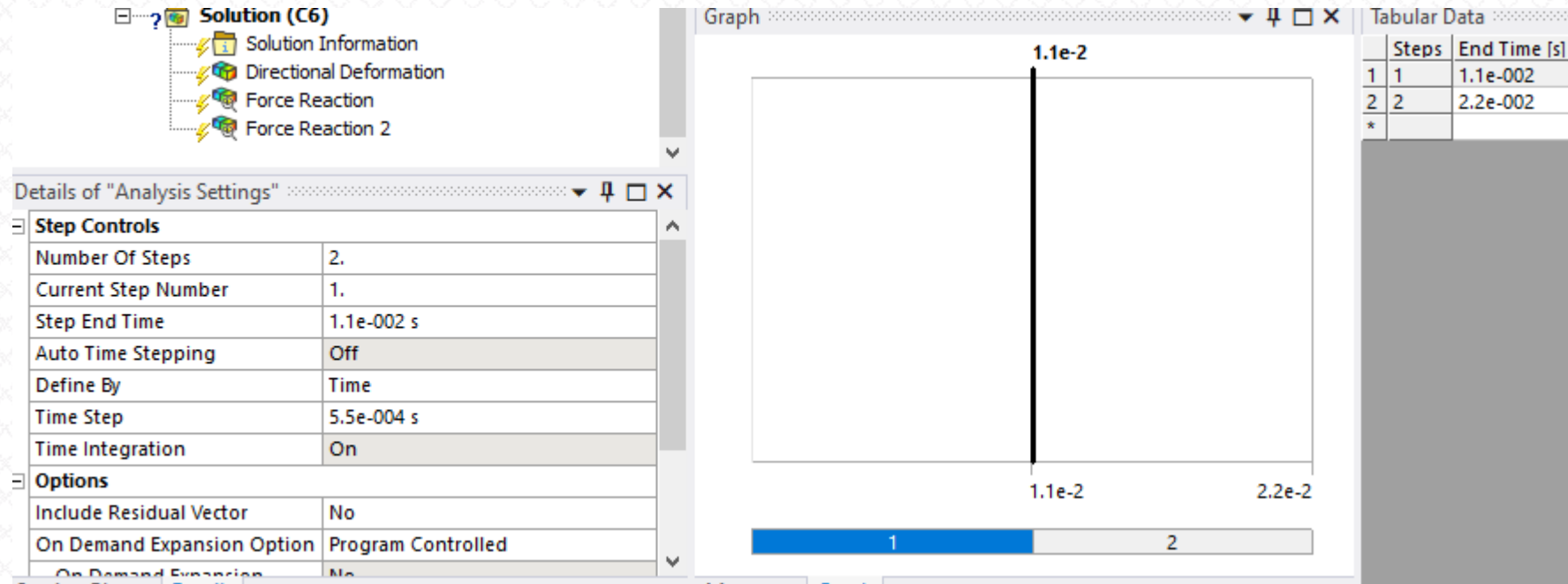
or if no input is provided:

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

Damping Controls	
Eqv. Damping Ratio From Modal	No
<input type="checkbox"/> Damping Ratio	5.e-002
Stiffness Coefficient Define By	Direct Input
<input type="checkbox"/> Stiffness Coefficient	0.
<input type="checkbox"/> Mass Coefficient	0.
Numerical Damping	Program Controlled
Numerical Damping Value	0.005

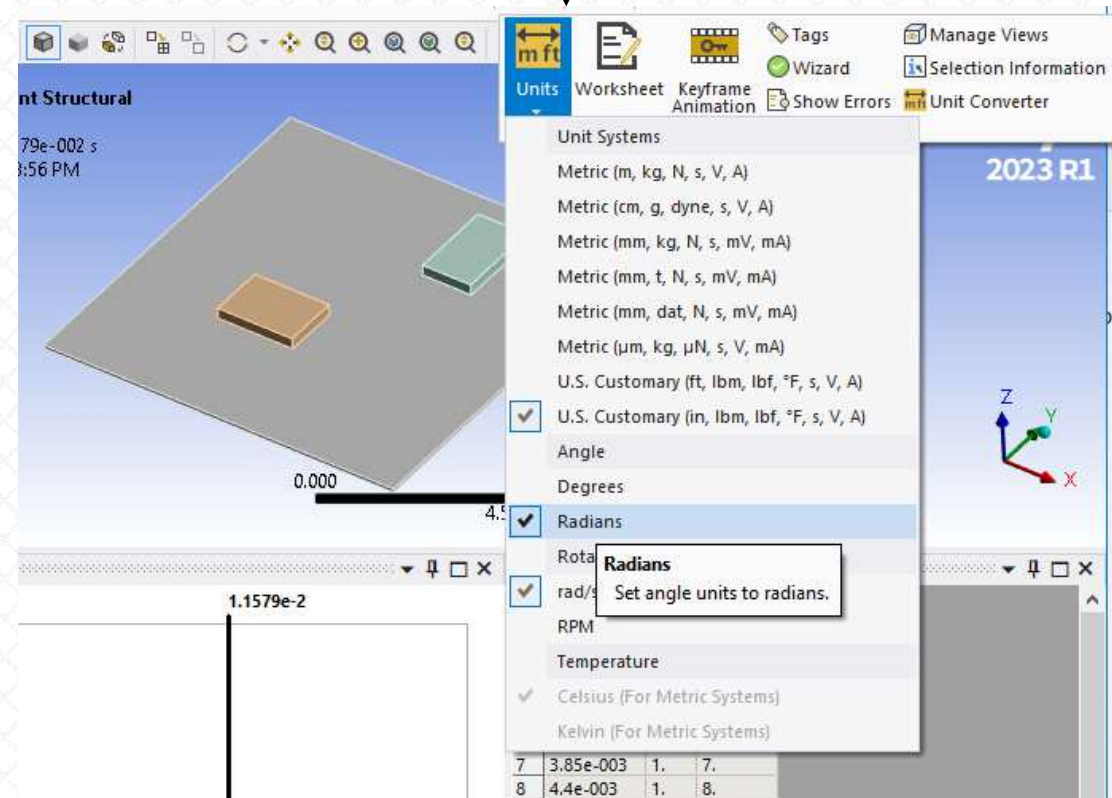
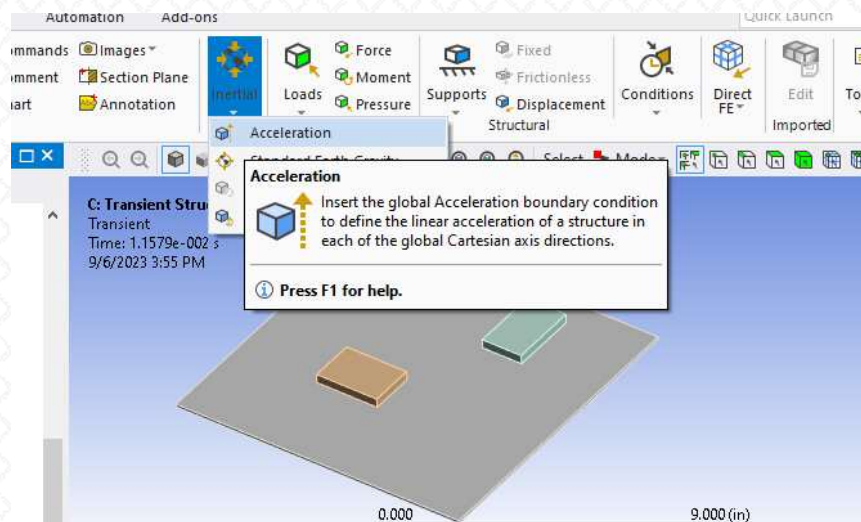
Transient Analysis

- 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



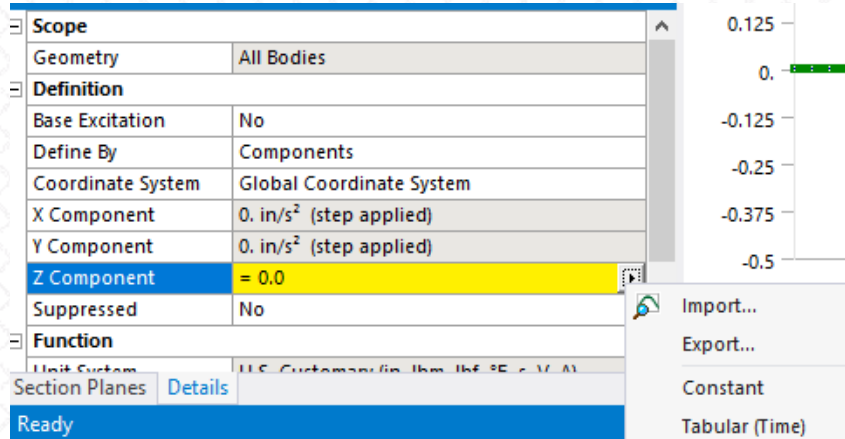
Transient Analysis

- 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

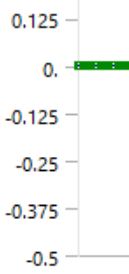


Transient Analysis

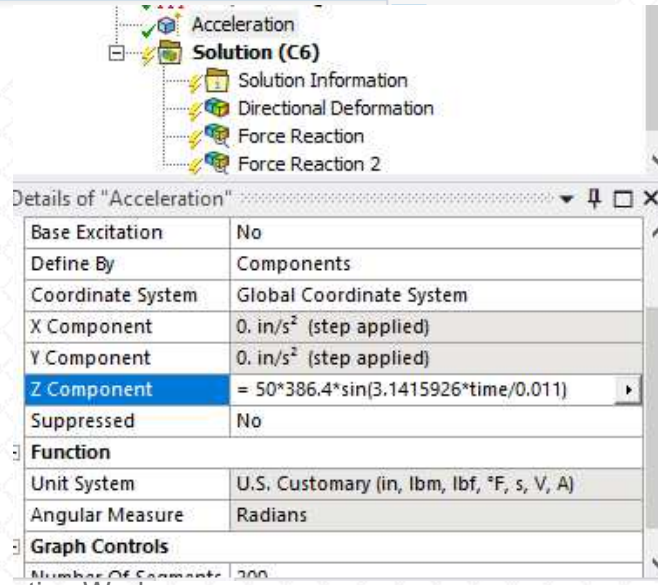
- Define the half-sine pulse as an acceleration load using the 'function' option in the z-direction



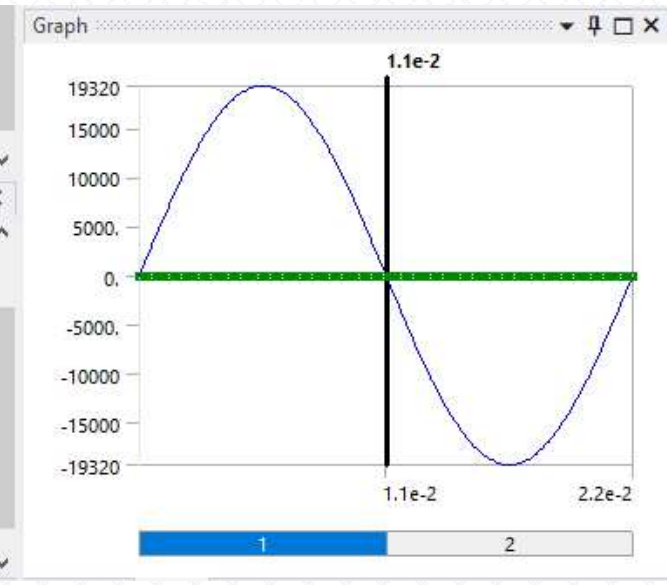
Scope
 Geometry: All Bodies
Definition
 Base Excitation: No
 Define By: Components
 Coordinate System: Global Coordinate System
 X Component: 0. in/s² (step applied)
 Y Component: 0. in/s² (step applied)
 Z Component: = 0.0
 Suppressed: No
Function
 Unit System: U.S. Customary (in, lbf, °F, s, V, A)



- for angle units of radians, use the equation:
 $50 \cdot 386.4 \cdot \sin(3.141526 \cdot \text{time} / 0.011)$
- for angle units of degrees, use the equation:
 $50 \cdot 386.4 \cdot \sin(180 \cdot \text{time} / 0.011)$



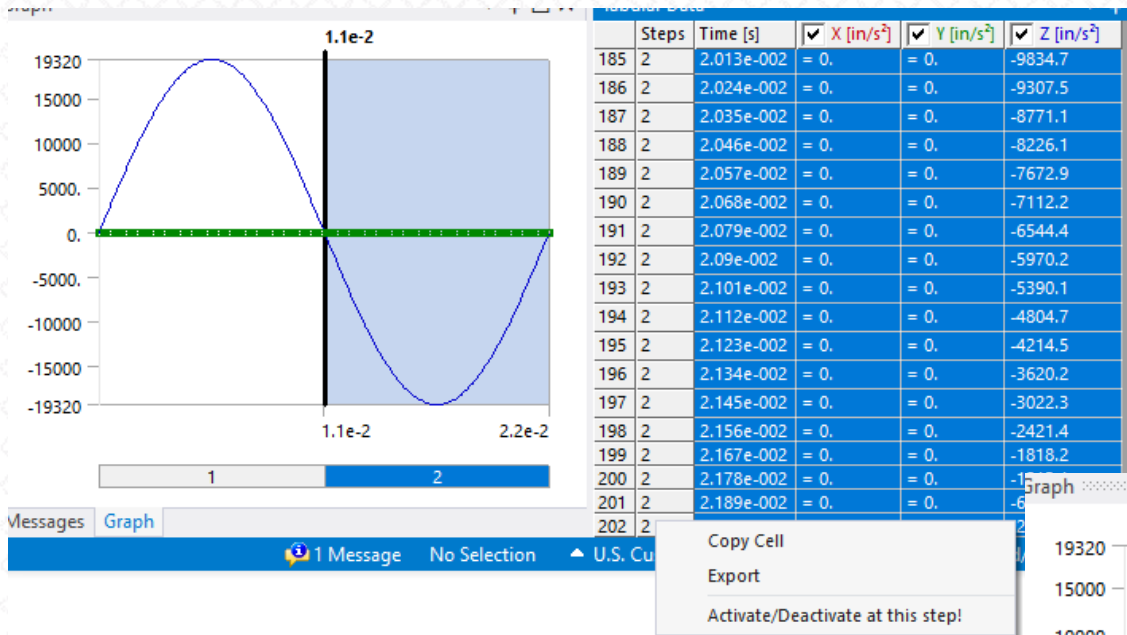
Details of "Acceleration"
 Base Excitation: No
 Define By: Components
 Coordinate System: Global Coordinate System
 X Component: 0. in/s² (step applied)
 Y Component: 0. in/s² (step applied)
 Z Component: = 50*386.4*sin(3.141526*time/0.011)
 Suppressed: No
Function
 Unit System: U.S. Customary (in, lbf, °F, s, V, A)
 Angular Measure: Radians



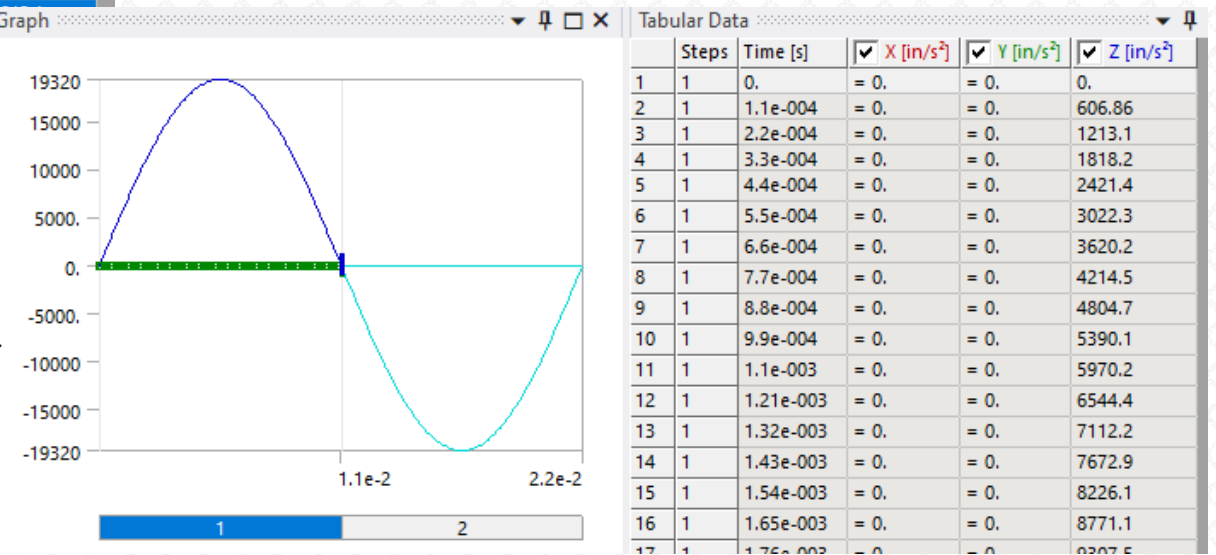
Steps	Time [s]	X [in/s ²]	Y [in/s ²]	Z [in/s ²]
1	0.	= 0.	= 0.	0.
2	1.1e-004	= 0.	= 0.	606.86
3	2.2e-004	= 0.	= 0.	1213.1
4	3.3e-004	= 0.	= 0.	1818.2
5	4.4e-004	= 0.	= 0.	2421.4
6	5.5e-004	= 0.	= 0.	3022.3
7	6.6e-004	= 0.	= 0.	3620.2
8	7.7e-004	= 0.	= 0.	4214.5
9	8.8e-004	= 0.	= 0.	4804.7
10	9.9e-004	= 0.	= 0.	5390.1
11	1.1e-003	= 0.	= 0.	5970.2
12	1.21e-003	= 0.	= 0.	6544.4
13	1.32e-003	= 0.	= 0.	7112.2
14	1.43e-003	= 0.	= 0.	7672.9
15	1.54e-003	= 0.	= 0.	8226.1
16	1.65e-003	= 0.	= 0.	8771.1
17	1.76e-003	= 0.	= 0.	9307.5

Transient Analysis

- Deactivate all loading at load step 2 (select the cells shown below in “Tabular Data”, right-click->Activate/Deactivate this step!)

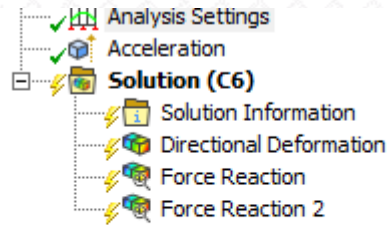


- if successful, you should now have a half-sine pulsed acceleration over the interval $0 \leq t \leq 0.011s$, while the solution end time $t = 0.022s$



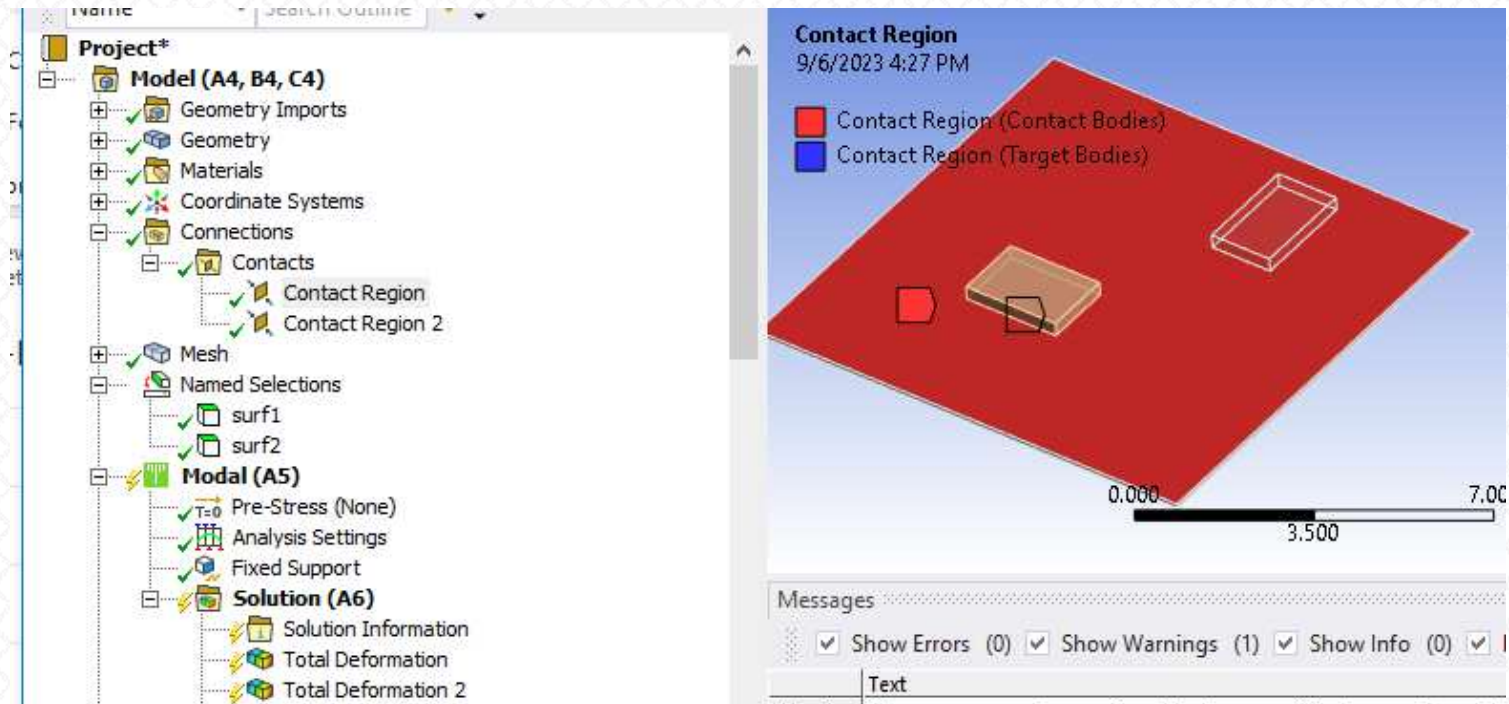
Transient Analysis

- 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)



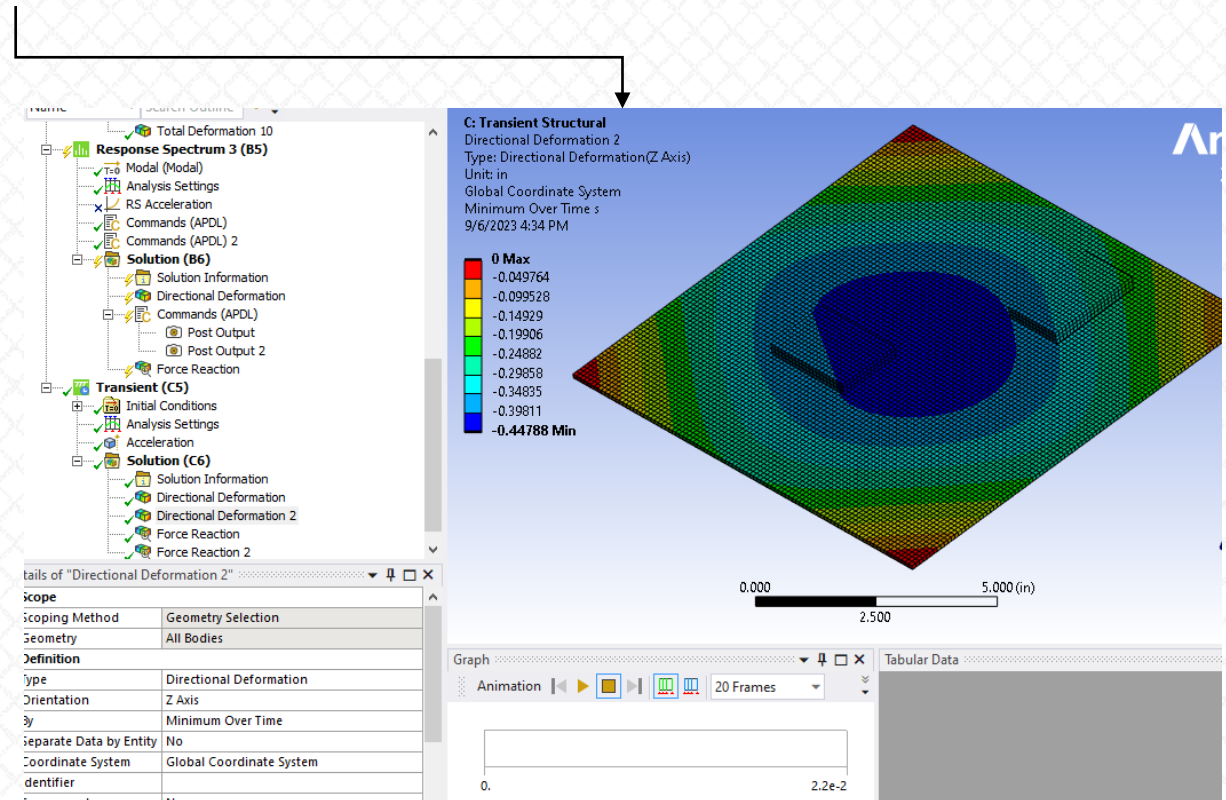
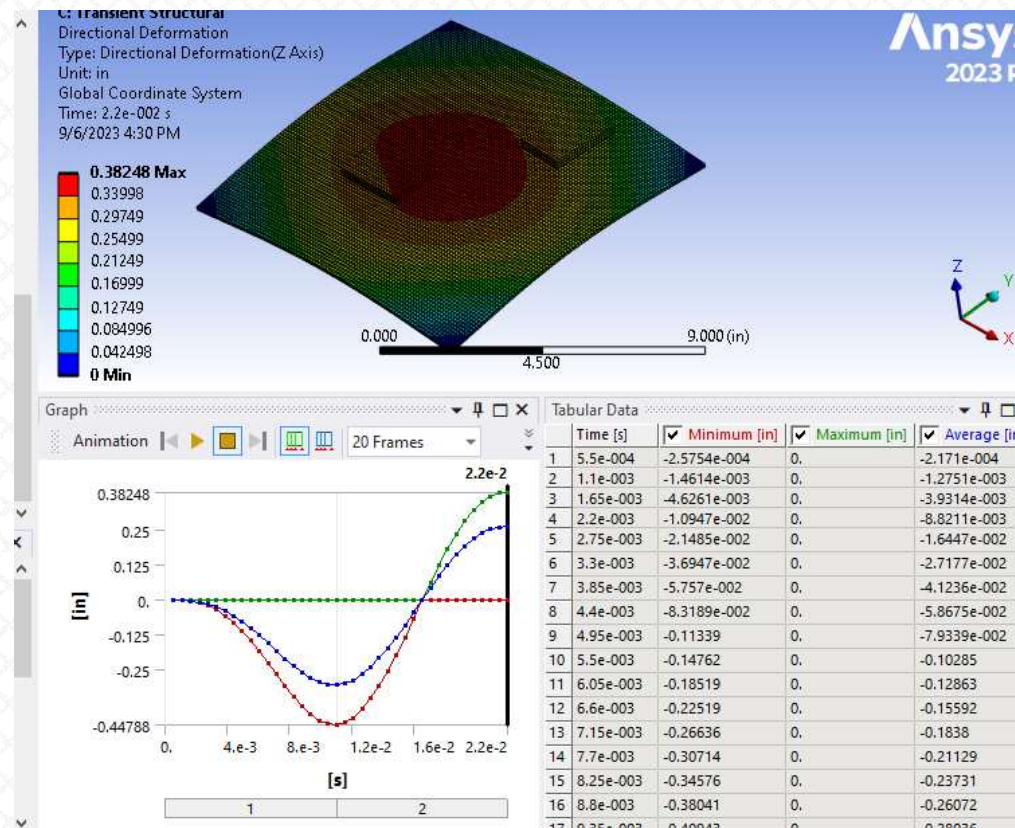
Details of "Analysis Settings"

Mode Selection Method	None
Output Controls	
Stress	Yes
Back Stress	No
Strain	Yes
Contact Data	Yes
Nodal Forces	Yes
Volume and Energy	Yes
Euler Angles	Yes
Calculate Reactions	Yes
Calculate Velocity and Acceler...	Yes



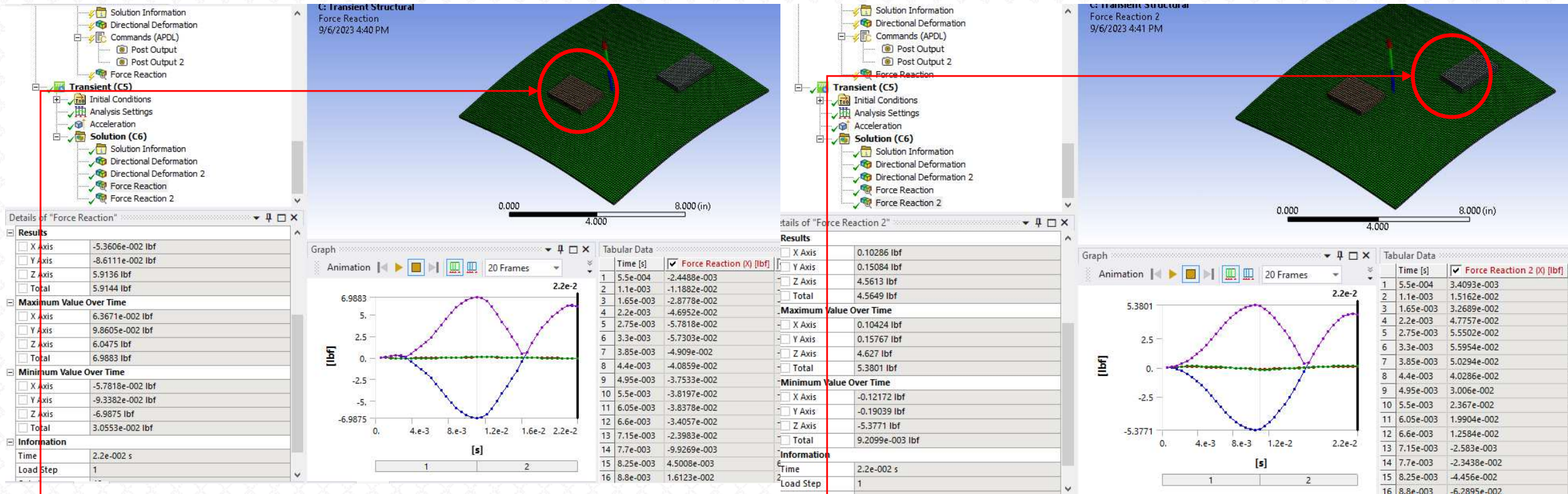
Transient Analysis

- Run the analysis. 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 interested in the maximum absolute values of the z-component).



$|F_{zmax}| = 6.9875 \text{ lbf @ Contact Region 1 (0.011s)}$

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

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 technique 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](#)

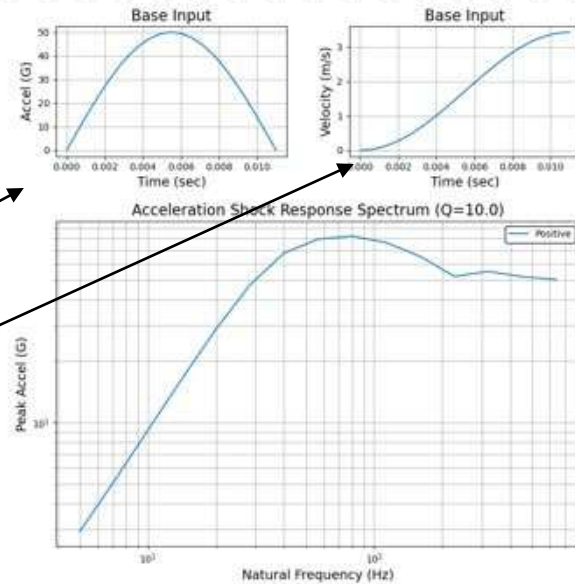
*this technique is unfortunately named, as it has little if anything to do with 'spectral analysis'. However, googling 'Response Spectrum Method' will usually produce the desired results



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 applying 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



- 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

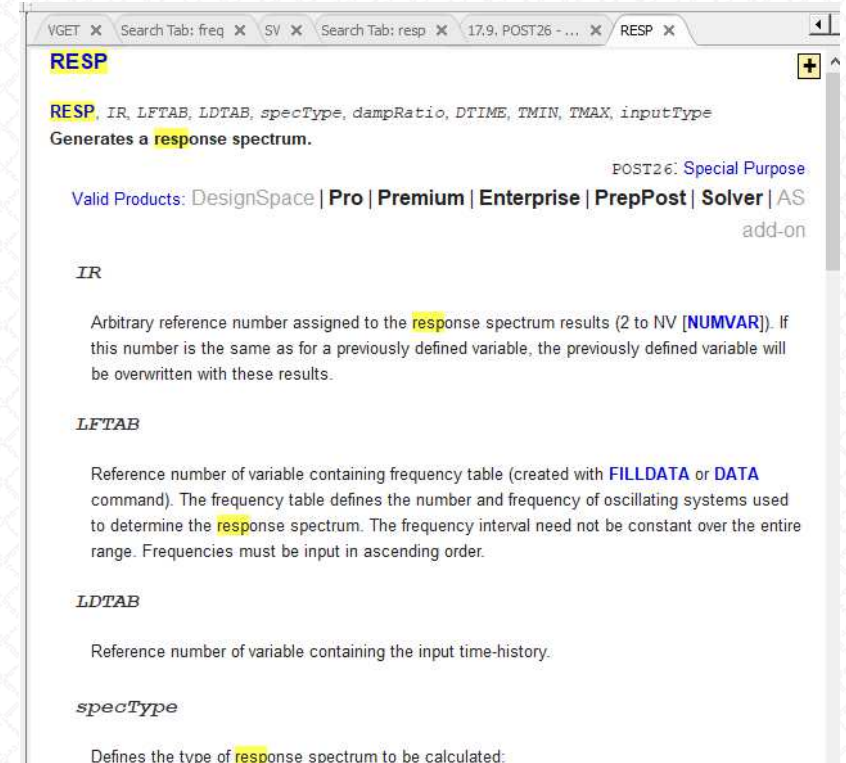
- Perhaps less widely known today is the fact that ANSYS offers its own functionality for generating shock spectra
- This is an APDL command called 'RESP'. And in the spirit of this blog, we're providing this solution to users of ANSYS as well (for more, search for 'RESP' in the documentation. A screenshot 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. After a modal analysis...npi=acos(-1)n*amplitude=20 ! pulse amplitude (20 g)n*width=0.01 ! pulse width (10 milli-sec)n n*dim,acel,table,4,1nace(1,0)=0.0, 0.5*width, width, 5.0*widthnace(1,1)=0.0, amplitude, 0.0, 0.0n n/axlab,x,TIME n/axlab,y,ACEL (g)n*vpplot,acel(1,0),acel(1,1) ! verify acceleration inputThe 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 forvput,acel(1,0),1,0 ! variable 1 is time valuesnvp,acel(1,1),2,0 ! variable 2 is acceleration valuesndata,5,1,10,1,Frequency ! 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 readableSpec=6 ! variable to contain spectrum outputnFreq=5 ! variable with frequencies to be solved forInp_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)n/xvar,Freq ! x axis is frequencynplvar,Spec ! plot acceleration spectrumn



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

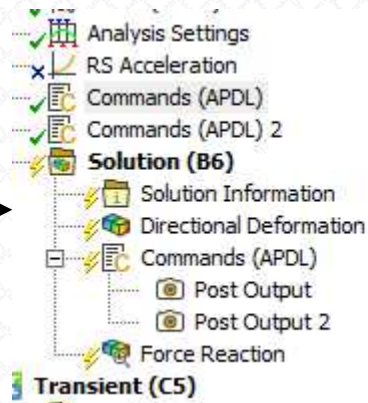
Reference number of variable containing the input time-history.

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



```
6 ! See Solving Units in
7
8 !The default for spectrum analysis is
9 !Specifying 'total' in the fourth arg
10 !this is consistent with the default
11 srss,0.0,disp,,total
```

```
5 ! NOTE: Any data that requires units (such as mass) is assumed to
6 ! See Solving Units in the help system for more info
7
8 fini
9 ddamp = 0.05 !damping value (irrelevant for singl
10 pi=acos(-1) !pi
11 numfrqs = 19 !number of frequencies
12 amplitude = 50 !half-sine amplitude in g's
13 maxfreq = 500 !maximum frequency to calculate SRS
14 width = 0.011 !width of half-sine pulse
15 *dim,acel,table,numfrqs+1,1
16 !*dim,freqs,,numfrqs
17 *dim,spectrum,,numfrqs+1,2
18 counter=0.0
19 *do,i,1,numfrqs
20 accel(i,0)=counter
21 accel(i,1) = amplitude*sin(counter*pi/width)
22 counter = counter+width/(numfrqs-1)
23 *enddo
24 accel(numfrqs+1,0)=2*width
25
26
```

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'

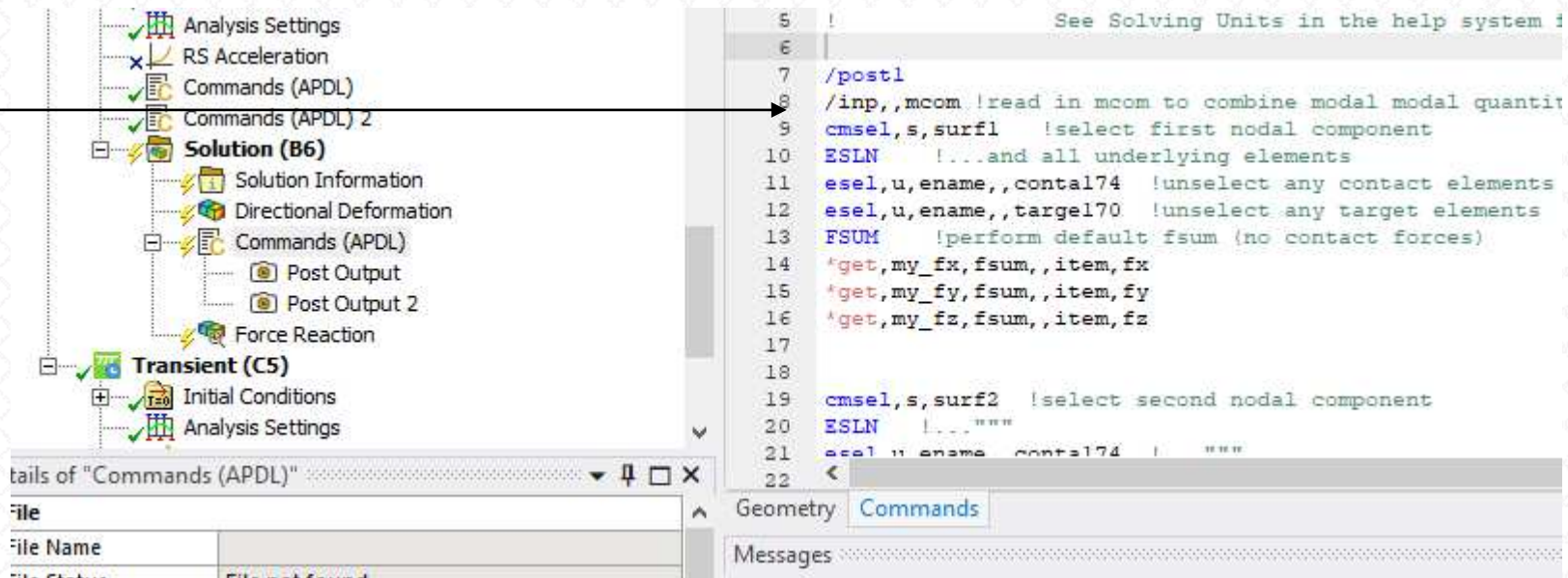
The screenshot displays the ANSYS software interface. On the left, the Search Outline shows a tree structure with 'Named Selections' containing 'surf1' and 'surf2'. The main window shows a 3D model of a diamond-shaped plate with two rectangular components on top. The Command Window on the right contains the following APDL code:

```
5 ! See Solving Units in the help system !
6
7 /post1
8 /inp,,mcom !read in mcom to combine modal modal quantities
9 cmsel,s,surf1 !select first nodal component
10 ESLN !...and all underlying elements
11 esel,u,ename,,contal74 !unselect any contact elements
12 esel,u,ename,,target170 !unselect any target elements
13 FSUM !perform default fsum (no contact forces)
14 *get,my_fx,fsum,,item,fx
15 *get,my_fy,fsum,,item,fy
16 *get,my_fz,fsum,,item,fz
17
18
19 cmsel,s,surf2 !select second nodal component
20 ESLN !...""
21 esel,u,ename,contal74 !...""
22 <
```

The Messages window at the bottom shows the output of the commands, including 'Geometry' and 'Commands'.

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

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

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:

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

Spectrum Analysis

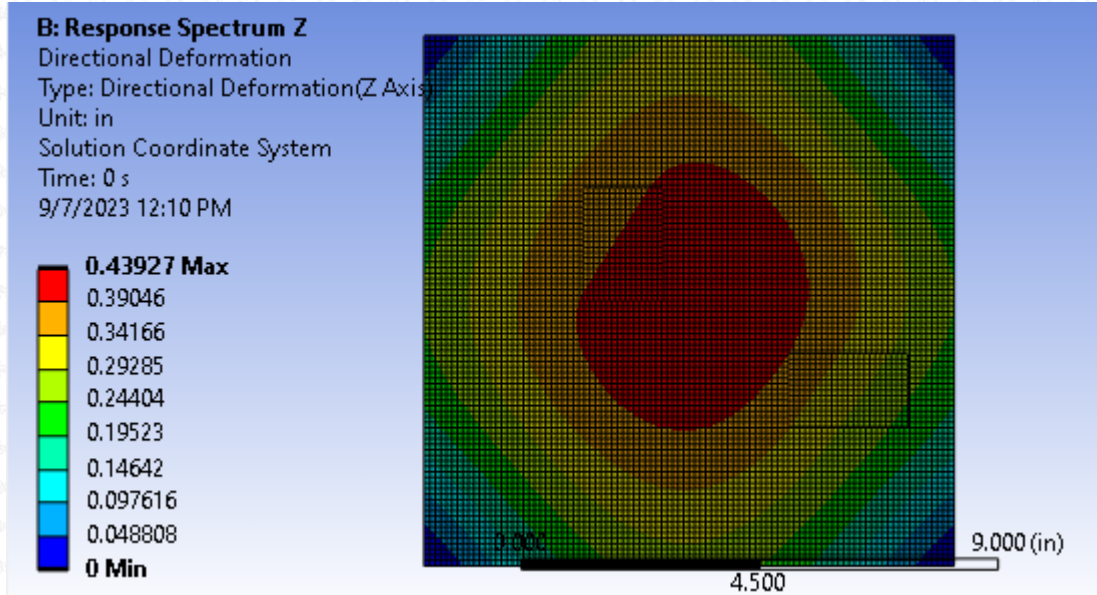
- Now, ANSYS automatically does this in a spectrum analysis (as you can verify by looking at the end of the ds.dat input file as shown here). But notice that it's only storing reaction forces for one 'compnames' variable. This corresponds to the fixed support. This ensures that we'll get the correct 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.



Response Spectrum 3 (B5)
 Modal (Modal)
 Analysis Settings
 RS Acceleration
 Commands (APDL)
 Commands (APDL) 2
 Solution (B6)
 Solution Information
 Directional Deformation
 Commands (APDL)
 Post Output
 Post Output 2
 Force Reaction
 Transient (C5)
 Initial Conditions
 Analysis Settings

```

6 /post1
7 /inp,,mcom !read in mcom to combine modal modal quantities before :
8 cmsel,s,surf1 !select first nodal component
9 ESLN !...and all underlying elements
10 esel,u,ename,,contal74 !unselect any contact elements
11 esel,u,ename,,target170 !unselect any target elements
12 FSUM !perform default fsum (no contact forces)
13 *get,my_fx,fsum,,item,fx
14 *get,my_fy,fsum,,item,fy
15 *get,my_fz,fsum,,item,fz
16
17
18 cmsel,s,surf2 !select second nodal component
19 ESLN !...""
20 esel,u,ename,,contal74 !...""
21 esel,u,ename,,target170 !...""
22 FSUM
23 *get,my_fx2,fsum,,item,fx
24 *get,my_fy2,fsum,,item,fy
25 *get,my_fz2,fsum,,item,fz
26 <
    
```

Details of "Commands (APDL)"

Target	Mechanical APDL
Input Arguments	
<input type="checkbox"/> ARG1	
<input type="checkbox"/> ARG2	
<input type="checkbox"/> ARG3	
<input type="checkbox"/> ARG4	
<input type="checkbox"/> ARG5	
<input type="checkbox"/> ARG6	
<input type="checkbox"/> ARG7	
<input type="checkbox"/> ARG8	
<input type="checkbox"/> ARG9	
Results	
<input type="checkbox"/> my_fx	8.8027e-002
<input type="checkbox"/> my_fy	0.12201
<input type="checkbox"/> my_fz	6.909
<input type="checkbox"/> my_fx2	0.12899
<input type="checkbox"/> my_fy2	0.21003
<input type="checkbox"/> my_fz2	5.2091

Geometry Commands
 Messages
 Show Errors (0) Show Warnings (1) Show Info (0) Merge Messages Pop
 Text
 Warning: One or more parts were found to be unmodified so smart updated.

$|F_{z_{max}}| = 6.909 \text{ lbf @ Contact Region 1}$

$|F_{z_{max}}| = 5.209 \text{ lbf @ Contact Region 2}$



Spectrum Analysis

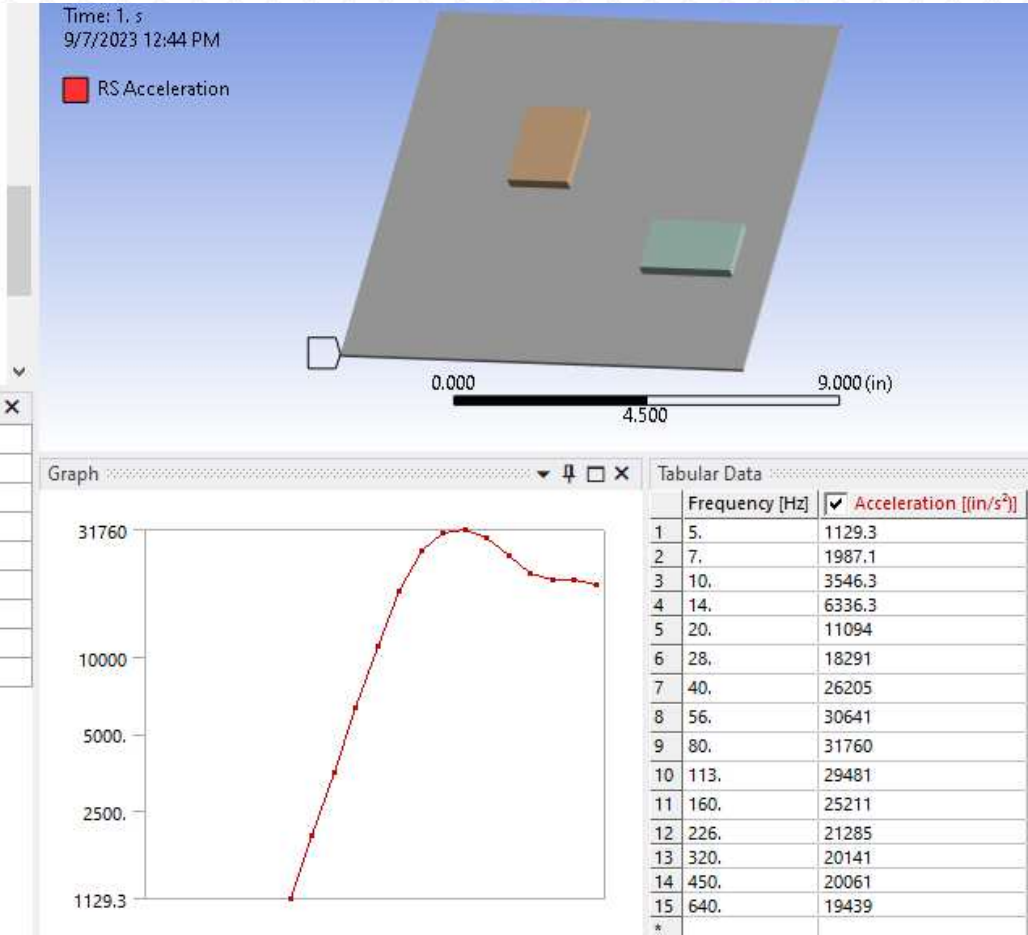
Finally, to compare spectrum results with macro-generated SRS to those with the spectrum calculated with Python (see slide 15), suppress the second macro (see below) and unsuppress the spectrum load in the tree outline

The screenshot shows the software tree outline with the following structure:

- Total Deformation (D0)
- Response Spectrum 3 (B5)
 - Modal (Modal)
 - Analysis Settings
 - RS Acceleration
 - Commands (APDL)
 - Commands (APDL) 2
- Solution (B6)
 - Solution Information
 - Directional Deformation
 - Commands (APDL)
 - Post Output
 - Post Output 2
 - Force Reaction
- Transient (C5)

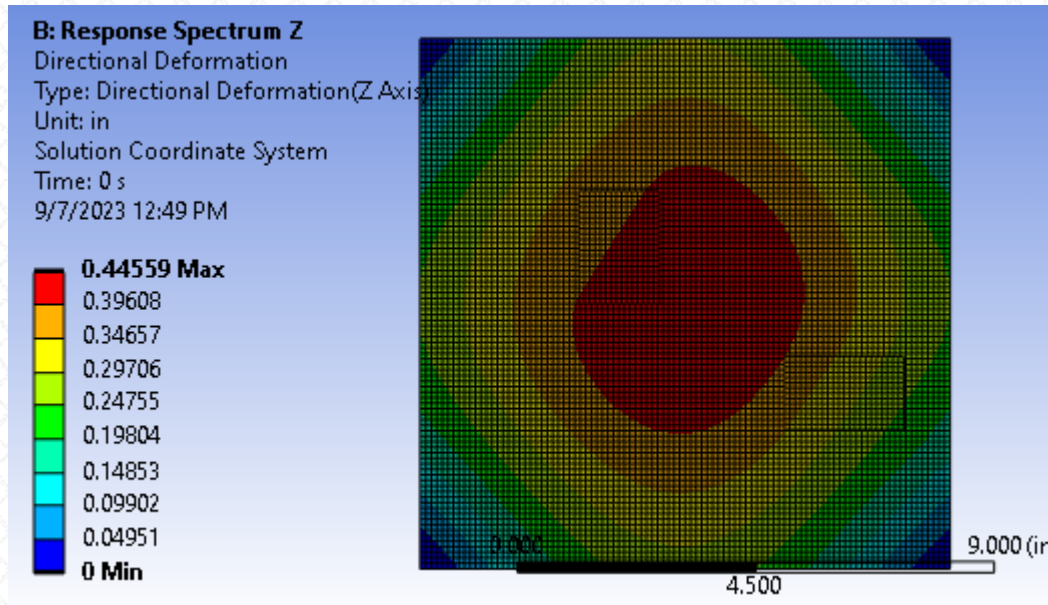
Below the tree is a table for "RS Acceleration":

Boundary Condition	All Supports
Definition	
Data	Tabular Data
Factor	1.
Direction	Z Axis
Including Mass Effect	No
Including Response Effect	No
Suppressed	No



Spectrum Analysis

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



```

5 ! See Solving Units in the help system
6 /post1
7 /inp,,mcom !read in mcom to combine modal modal quanti
8 cmsel,s,surf1 !select first nodal component
9 ESLN !...and all underlying elements
10 esel,u,ename,,contal174 !unselect any contact elements
11 esel,u,ename,,target170 !unselect any target elements
12 FSUM !perform default fsum (no contact forces)
13 *get,my_fx,fsum,,item,fx
14 *get,my_fy,fsum,,item,fy
15 *get,my_fz,fsum,,item,fz
16
17
18 cmsel,s,surf2 !select second nodal component
19 ESLN !...""
20 esel,u,ename,,contal174 !...""
21 esel,u,ename,,target170 !...""
22 FSUM
23 *get,my_fx2,fsum,,item,fx
24 *get,my_fy2,fsum,,item,fy
25 *get,my_fz2,fsum,,item,fz
26
    
```

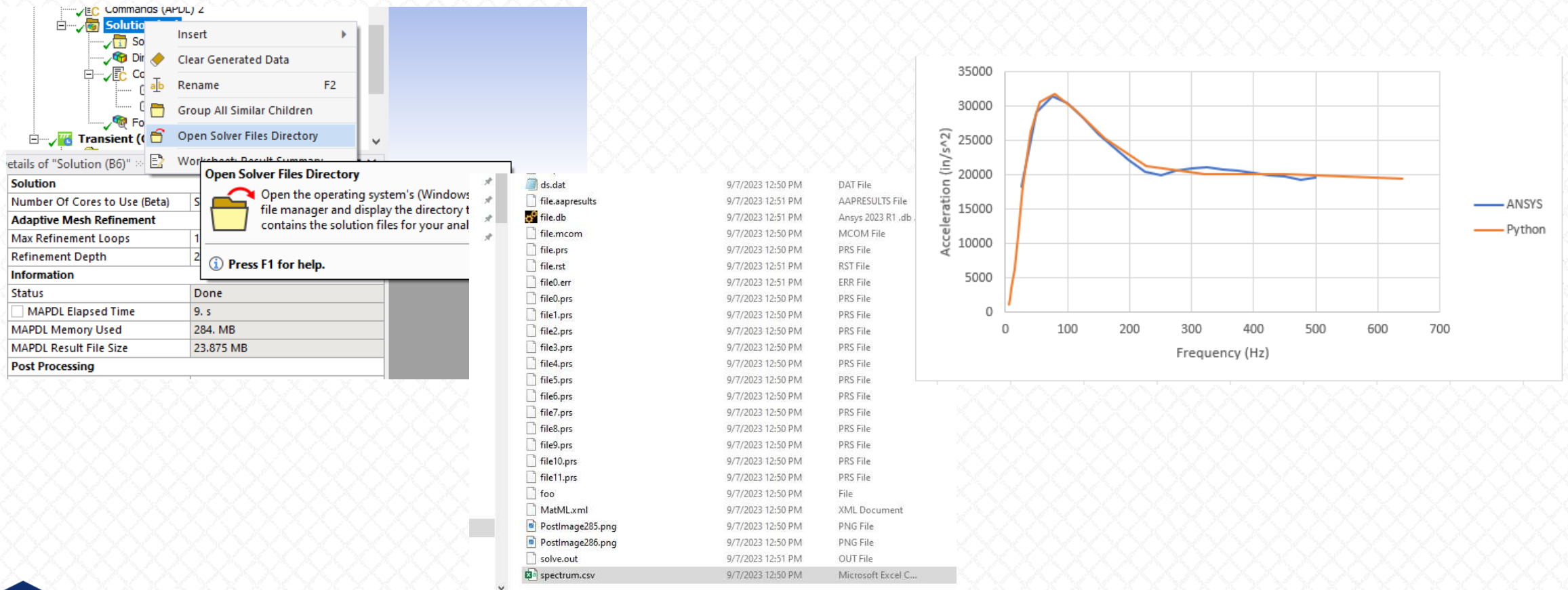
Target	Mechanical APDL
Input Arguments	
<input type="checkbox"/> ARG1	
<input type="checkbox"/> ARG2	
<input type="checkbox"/> ARG3	
<input type="checkbox"/> ARG4	
<input type="checkbox"/> ARG5	
<input type="checkbox"/> ARG6	
<input type="checkbox"/> ARG7	
<input type="checkbox"/> ARG8	
<input type="checkbox"/> ARG9	
Results	
<input type="checkbox"/> my_fx	8.8172e-002
<input type="checkbox"/> my_fy	0.12246
<input type="checkbox"/> my_fz	7.0047
<input type="checkbox"/> my_fx2	0.13069
<input type="checkbox"/> my_fy2	0.21121
<input type="checkbox"/> my_fz2	5.2842

$|F_{z_{max}}| = 7.00 \text{ lbf @ Contact Region 1}$

$|F_{z_{max}}| = 5.284 \text{ lbf @ Contact Region 2}$

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 demonstrate 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 script 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

