Engineering Overview of ACS SASSI NQA V4.3 Application to Seismic SSI Analysis of Safety-Related NPP Buildings

Ghiocel Predictive Technologies Inc.

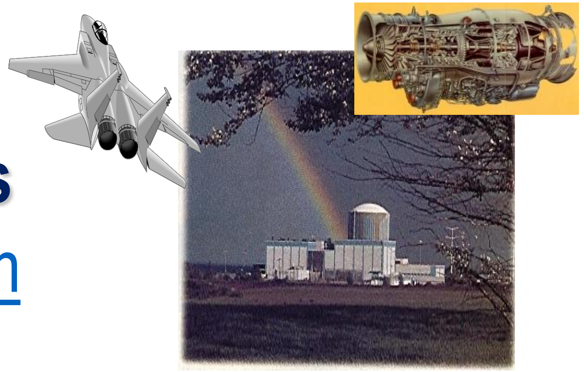
Dr. Dan M. Ghiocel

Member of ASCE 4 & 43 Standards

Email: dan.ghiocel@ghiocel-tech.com

Ghiocel Predictive Technologies Inc.

http://www.ghiocel-tech.com



Part 4: Description of ACS SASSI Advanced Options A-AA and NON

GP Technologies, Inc., Rochester, New York

October 2021

Part 4 Presentation Content:

1. Option A-AA: Integration with ANSYS Modeling

2. Option NON: Nonlinear SSI Analysis for Nonlinear RC Shearwall Structures

1. Option A-AA:
Integration with ANSYS Modeling

ACS SASSI-ANSYS Integration Capabilities (Options A and AA)

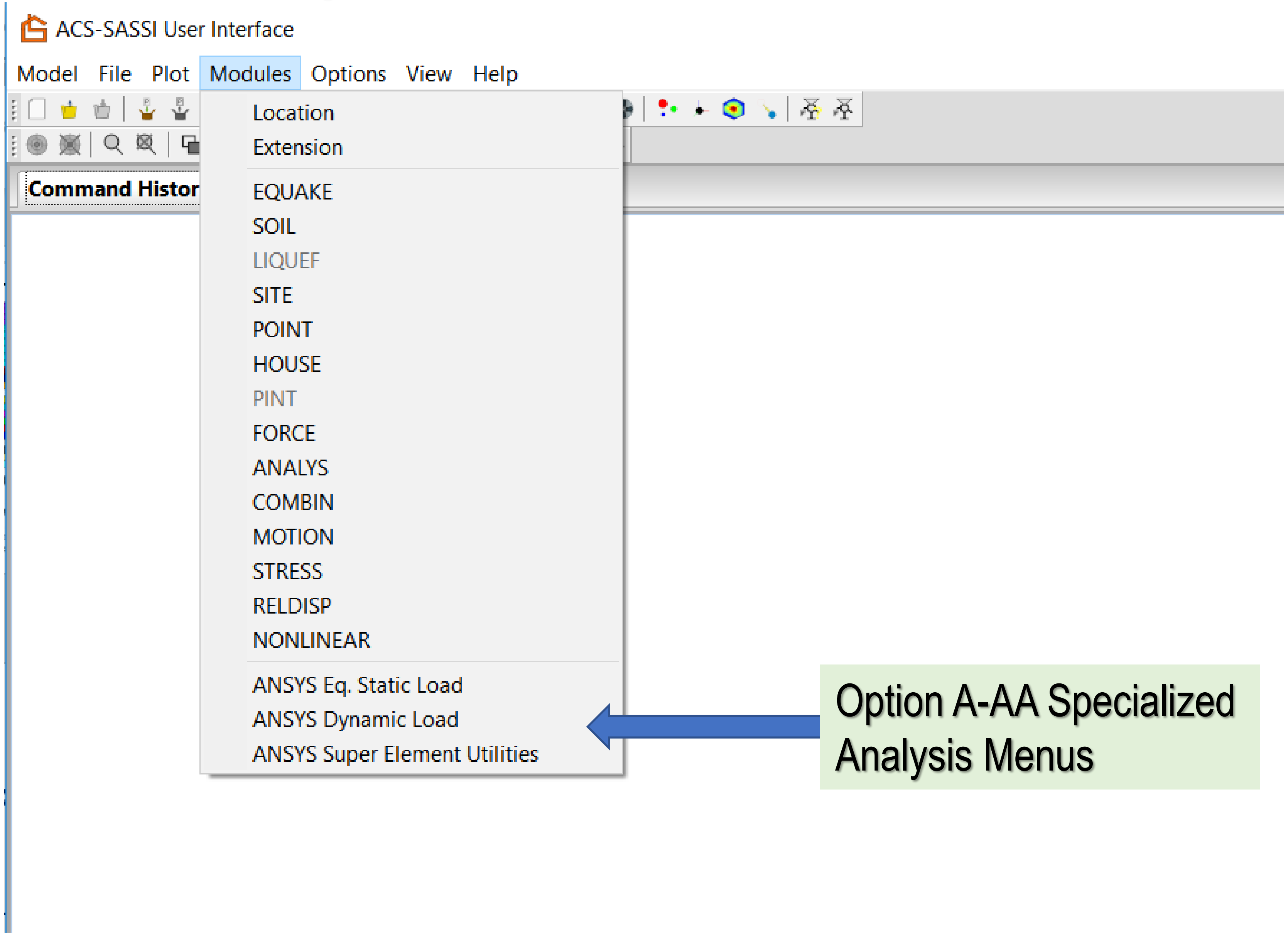
Two engineering analysis options in ACS SASSI:

i) ***One Step analysis*** *using ACS SASSI* for computing overall SSI responses of ACS SASSI or ANSYS FE model *(Option AA, AA-R)*

ii) ***Two Step analysis*** *using ACS SASSI in 1st step and ANSYS in 2nd step.* The 2nd step uses SSI response as input BCs *(Option A).*

Option AA-R uses ANSYS for harmonic SSI analysis by exporting the condensed excavated soil impedance matrix from ACS SASSI as a frequency dependent MATRIX50 super-element.

Option A-AA Menu Selections



OPTION A: ACS SASSI-ANSYS Interface for SSI Analysis Using ANSYS Models

ACS SASSI-ANSYS interfacing provides useful analysis capabilities:

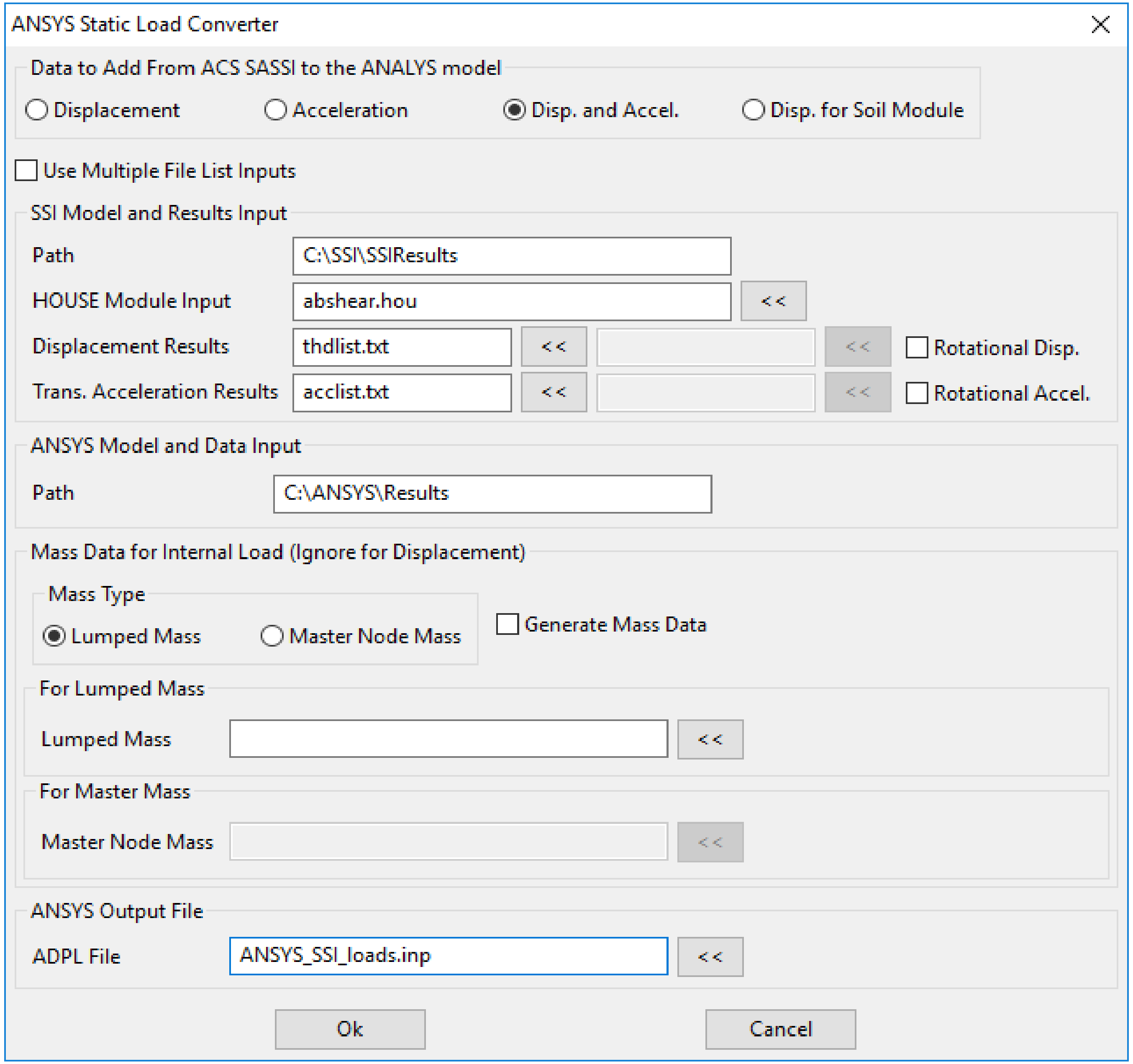
For *structural stress analysis (Demo 5)*:

- *ANSYS Equivalent-Static Seismic SSI Analysis* Using Refined FE Models (including refined mesh, element types including local nonlinearities, nonlinear materials, contact elements, etc.)
- *ANSYS Dynamic Seismic SSI Analysis* Using More Refined FE Models (including refined mesh, element types including local nonlinearities, nonlinear materials, contact elements, etc.)

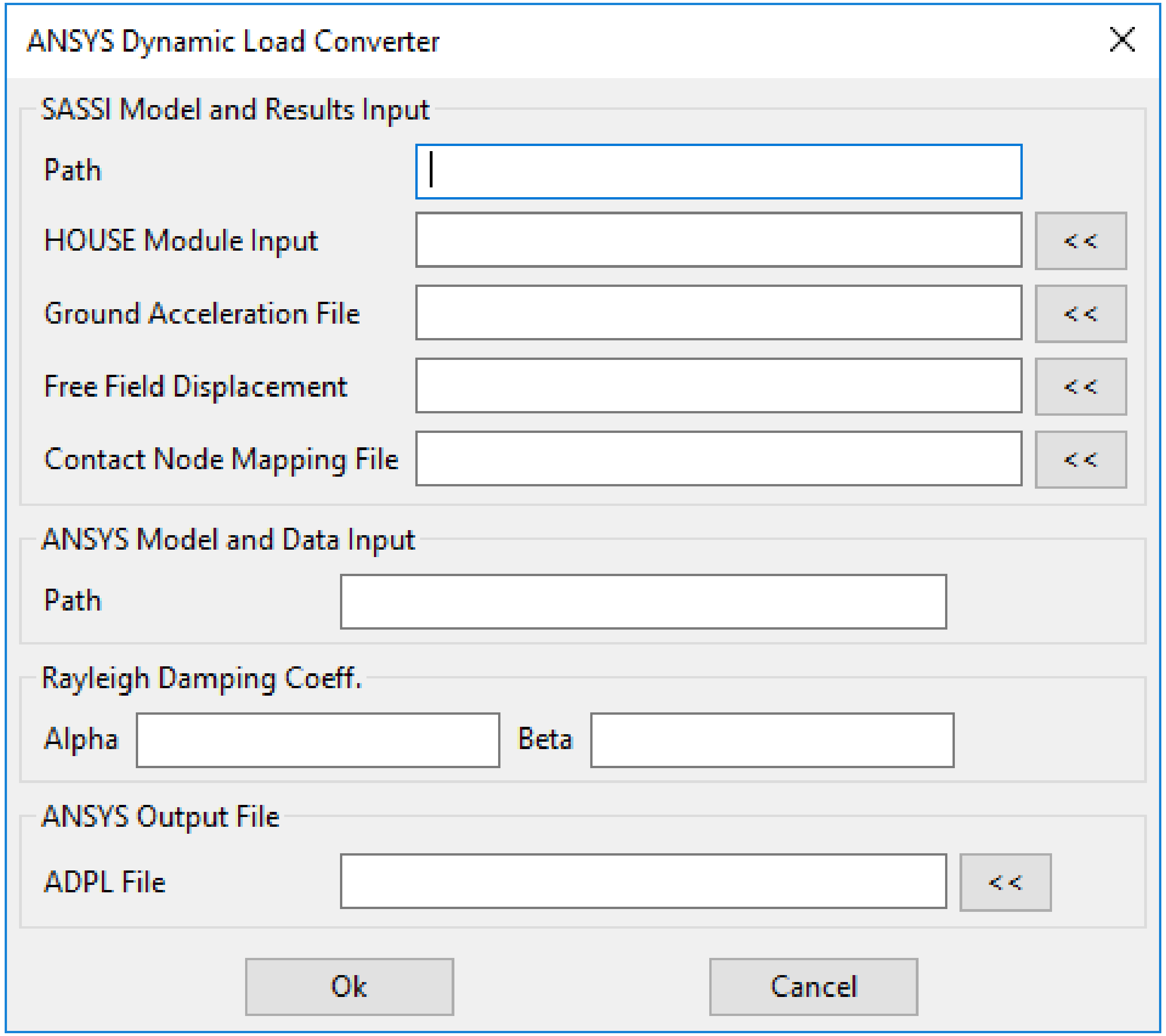
For *soil pressure computation (approximate) (Demo 6):*

- *ANSYS Equivalent-Static Seismic Soil Pressure Computation* including Soil-Foundation Separation Effects

UI Input for Option A Equivalent Static Option

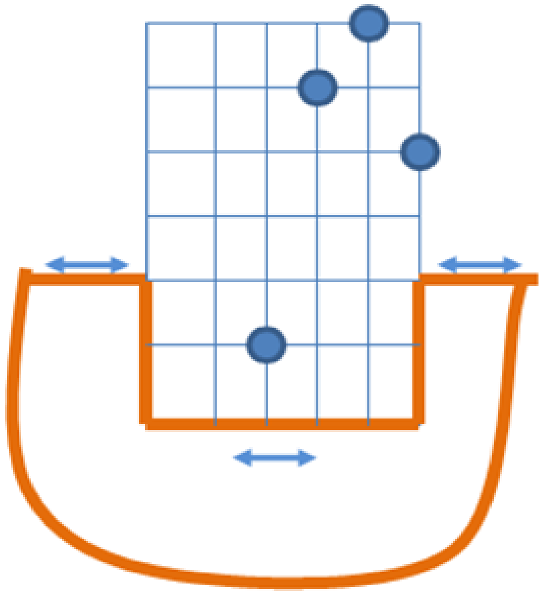


UI Input Windows for Option A Dynamic Option

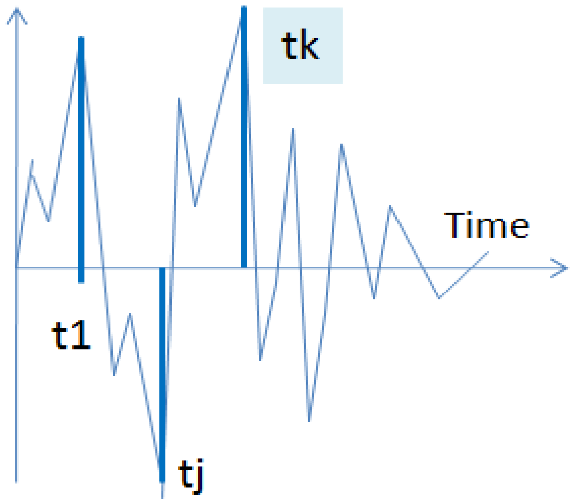


ACS SASSI Seismic SSI Analysis

Computing Structural Stress/Forces



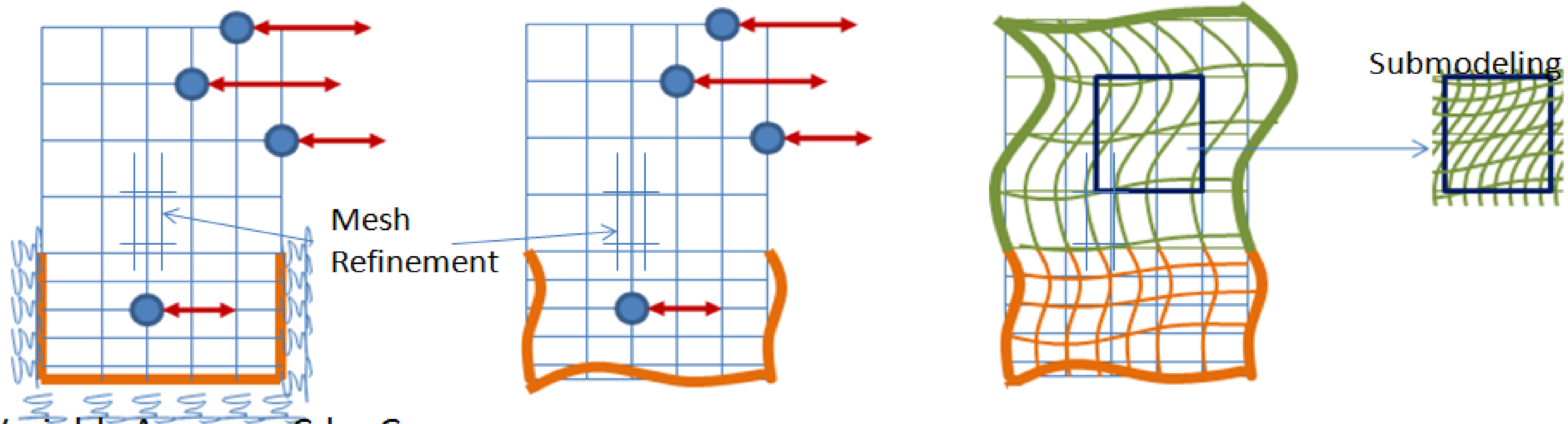
Selected Critical Time Steps for Maximum Stresses
To be Used for Equivalent Static Structural Analysis



Same mesh for ANSYS and ACS SASSI models is recommended for stress analysis.

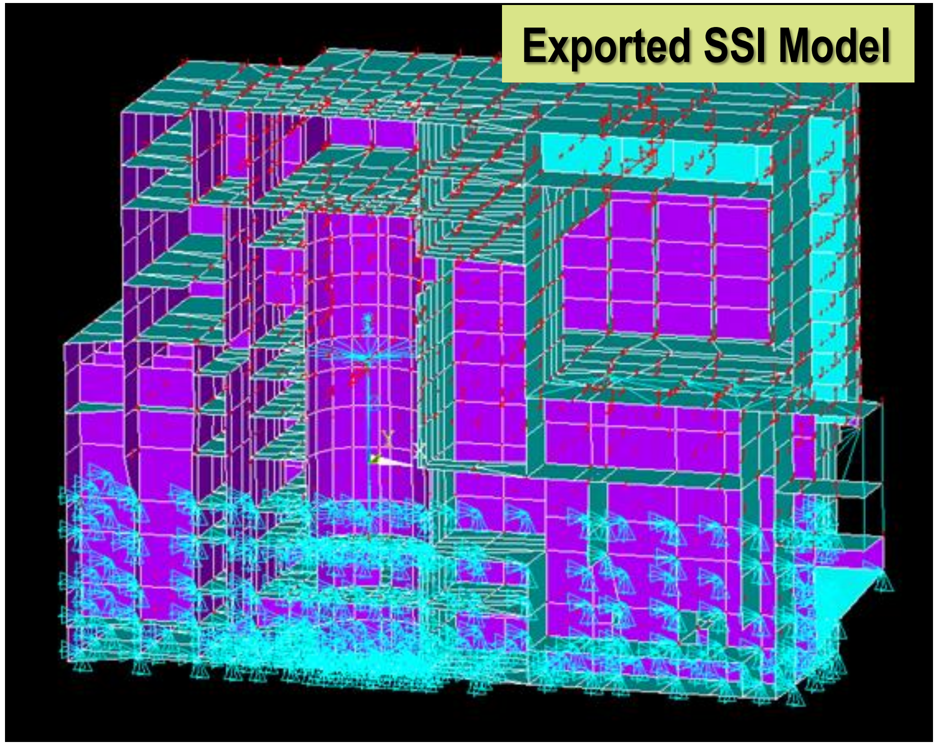
SSI Solution Time Frames As Equivalent Static Structural Loading at Critical Time Steps

EQS Forces + BC Springs | EQS Forces + BC Displacements | EQS Relative Displacements

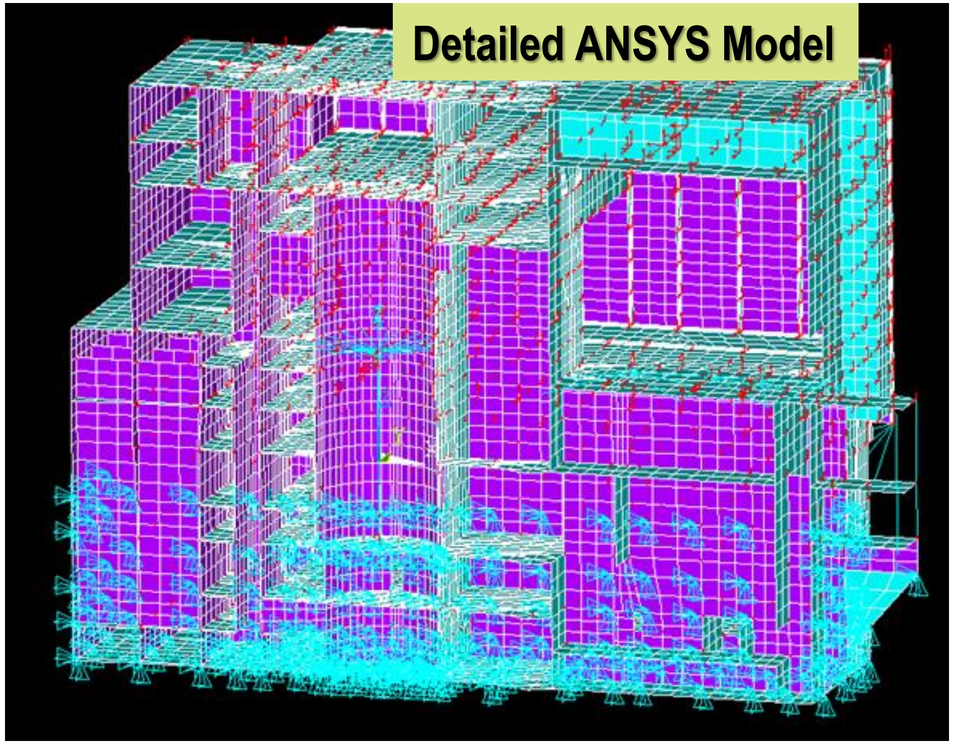


Variable Accuracy C-by-C | "Exact" | "Exact"

Option A for A Refined Seismic Stress Analysis (Demo 5)



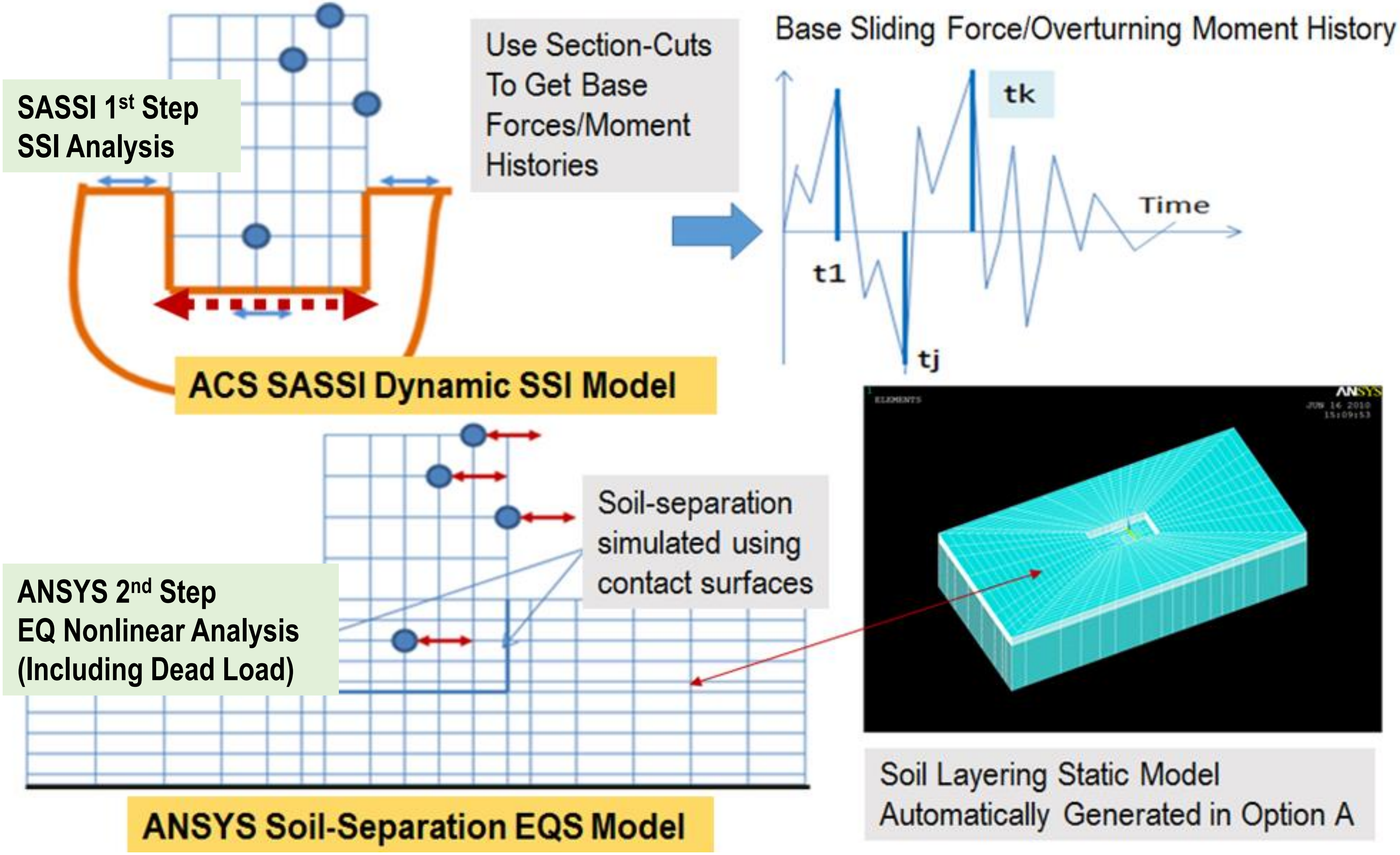
ANSYS Structural Model
Automatically Converted From
ACS SASSI Using PREP Module



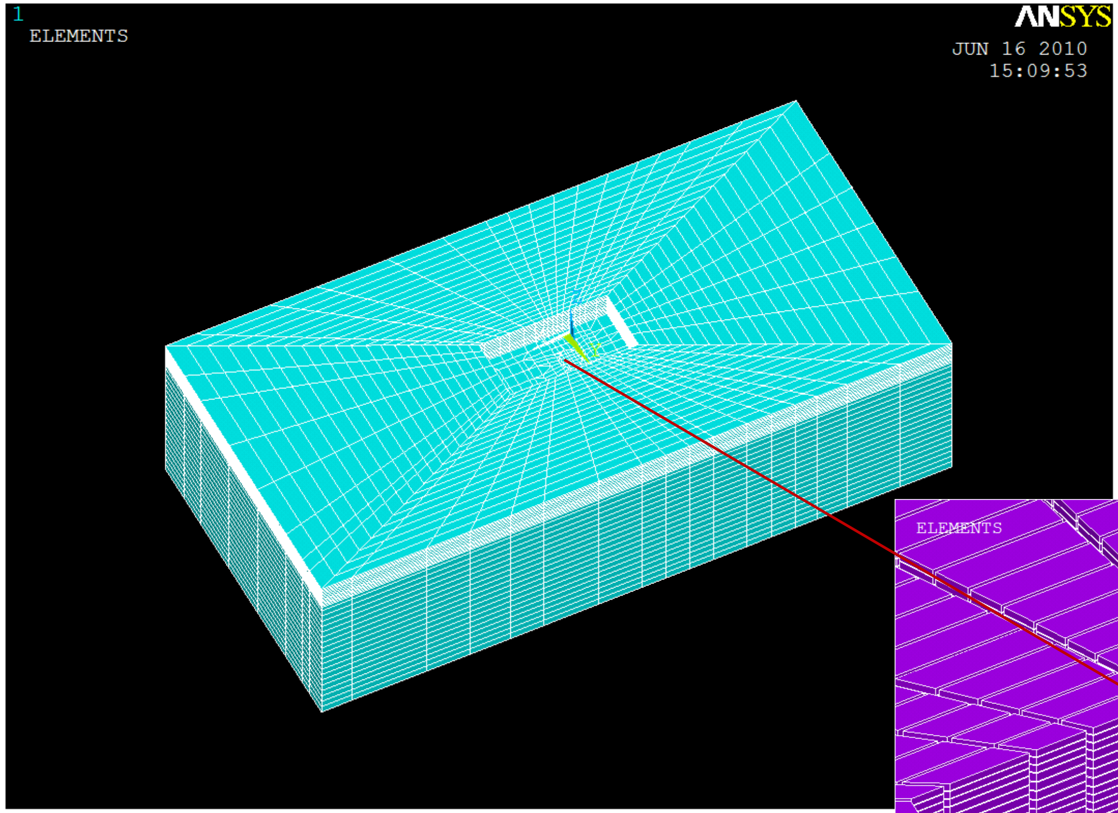
ANSYS Refined Structural Model
Using EREFINE command or
ANSYS GUI (rank 1-6)

Demo 5

Option A-Based Nonlinear Analysis for Computing Soil Separation Effects (2nd Step is in ANSYS)

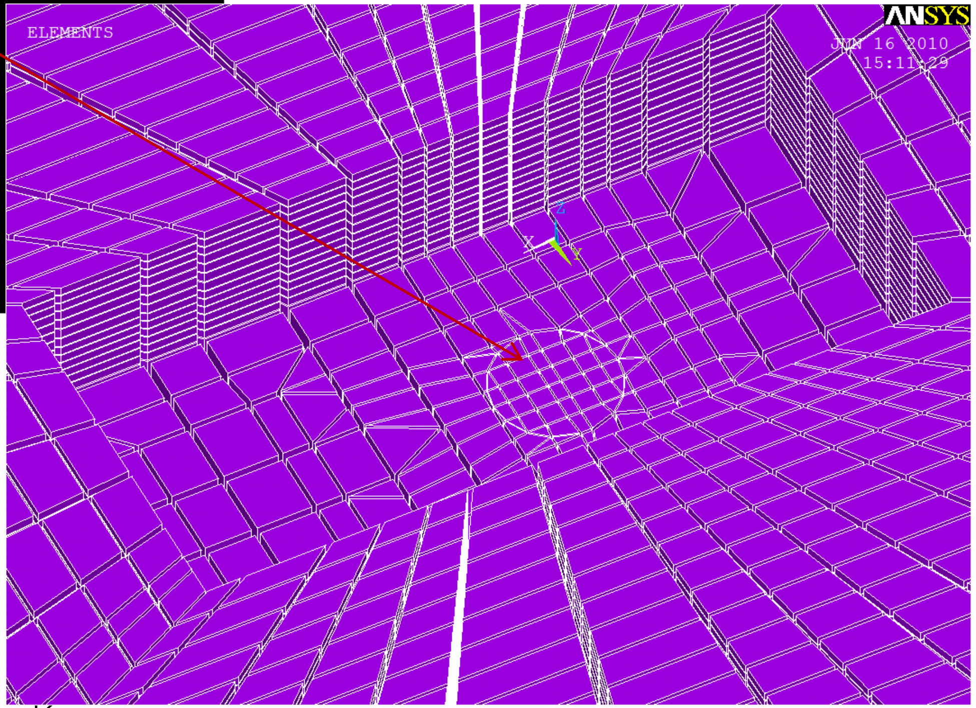


Option A for Seismic Soil Pressure Analysis (Demo 6)



ANSYS Soil FE Model
Is Automatically Generated
by SOILMESH Module

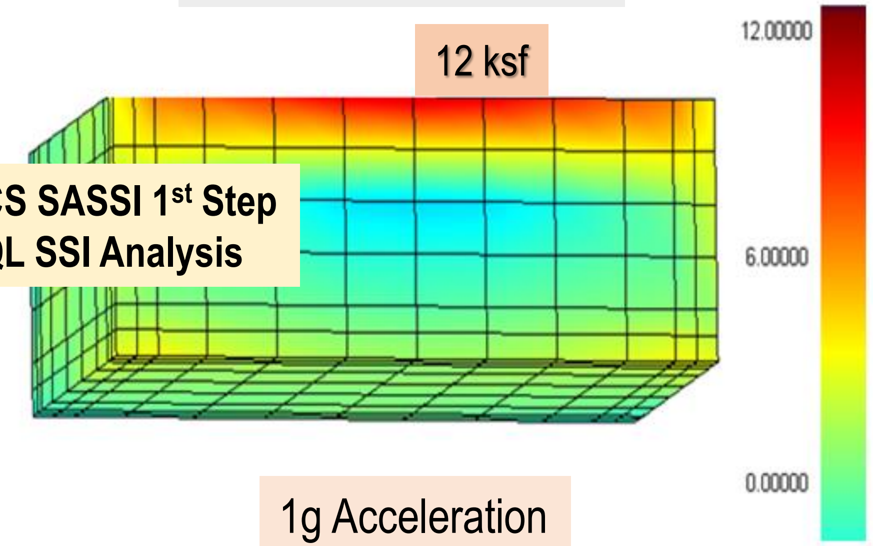
Demo 6



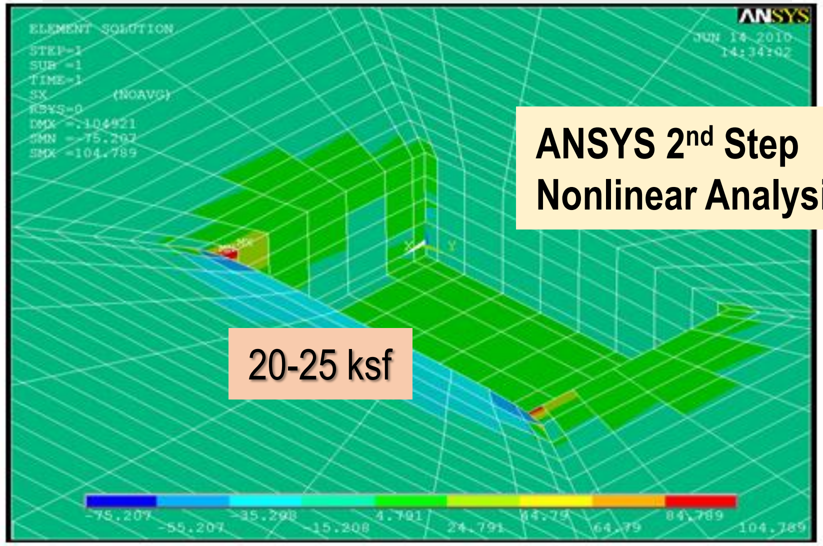
Embedment mesh is extended.
User controls extension size and
mesh density. Can use EREFINE.
Contact surfaces automatically added
by SOILMESH and SOILCONTACT commands.

Soil Separation Example Using Two-Step SSI Approach

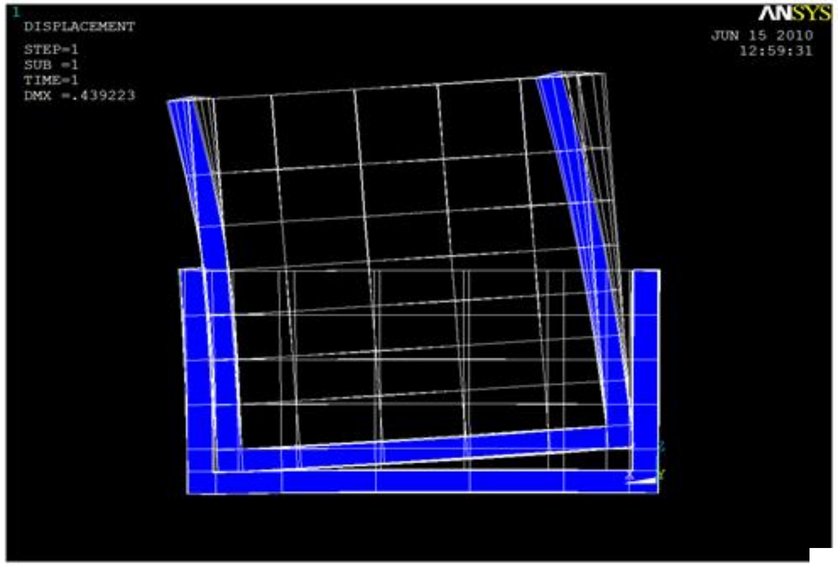
SASSI Soil Pressures

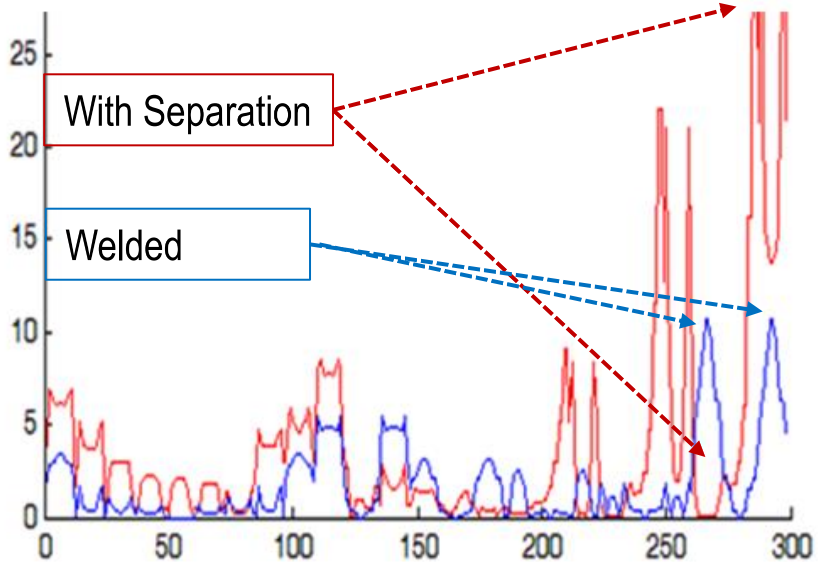


ACS SASSI Soil Pressures (Option A)



1g Acceleration





(Ghiocel and Saremi, SMiRT24 Paper 468, 2017)

SURFACE SSI MODEL

Flexible Base (w/ Displ. BC)

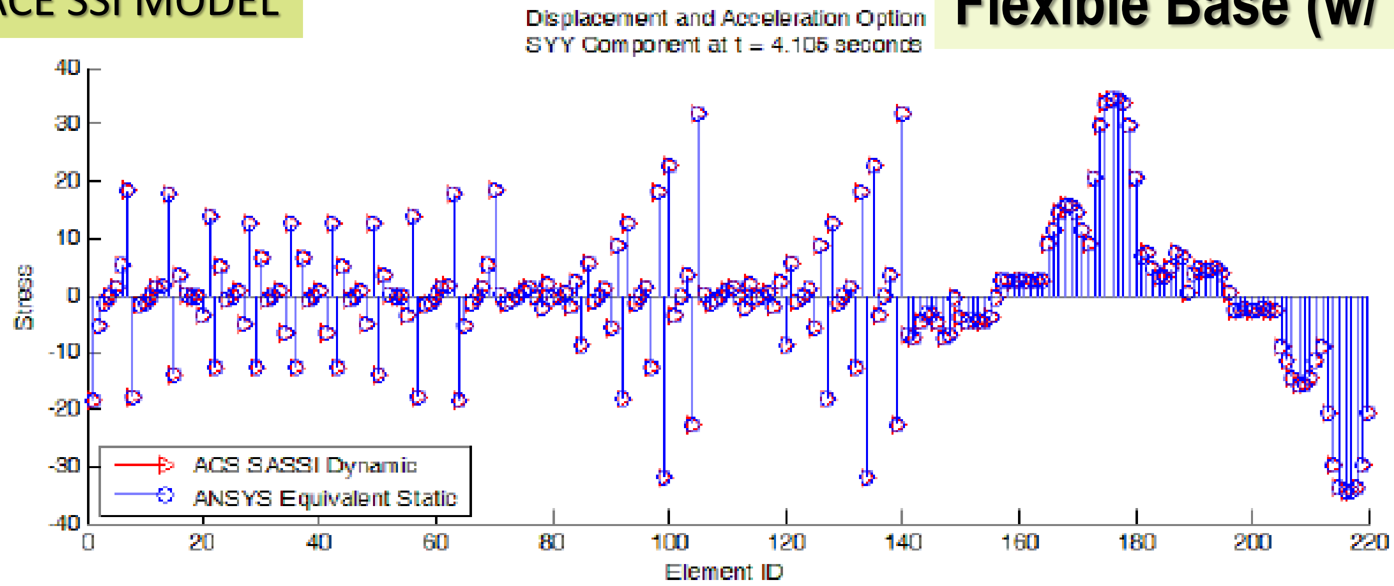


Figure 193: Case b) SYY Element Center Stresses for "Displacement and Acceleration" Option

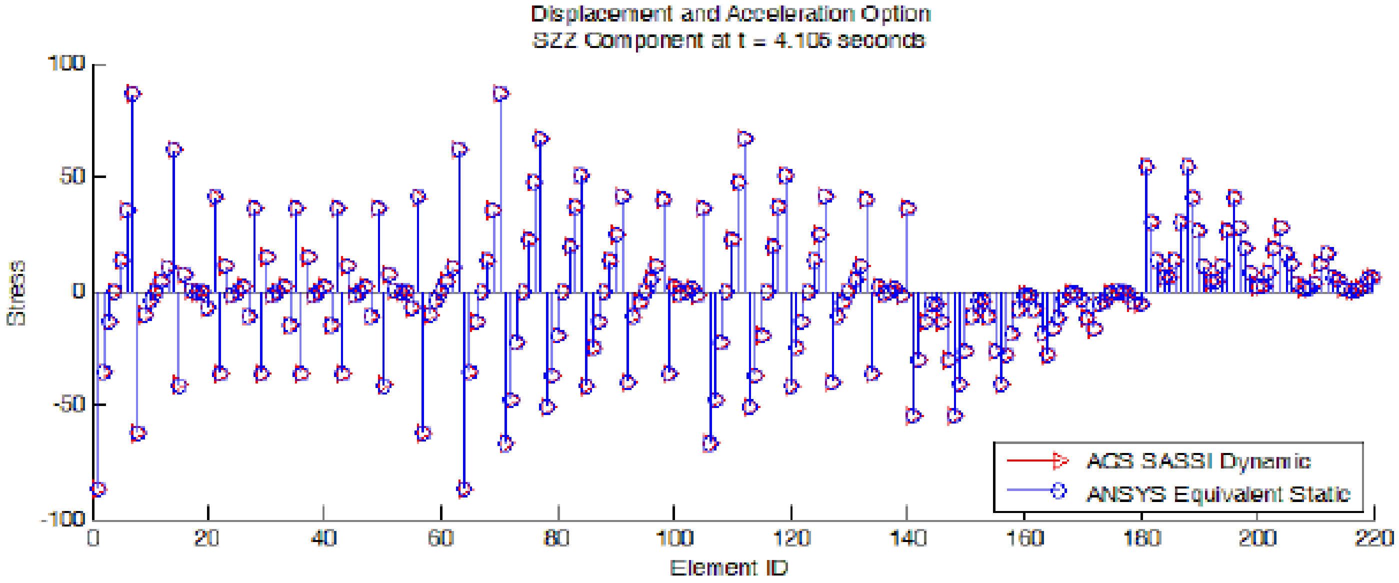


Figure 194: Case b): SZZ Element Center Stresses for "Displacement and Acceleration" Option

SURFACE SSI MODEL

Rigid Base (w/o Displ. BC)

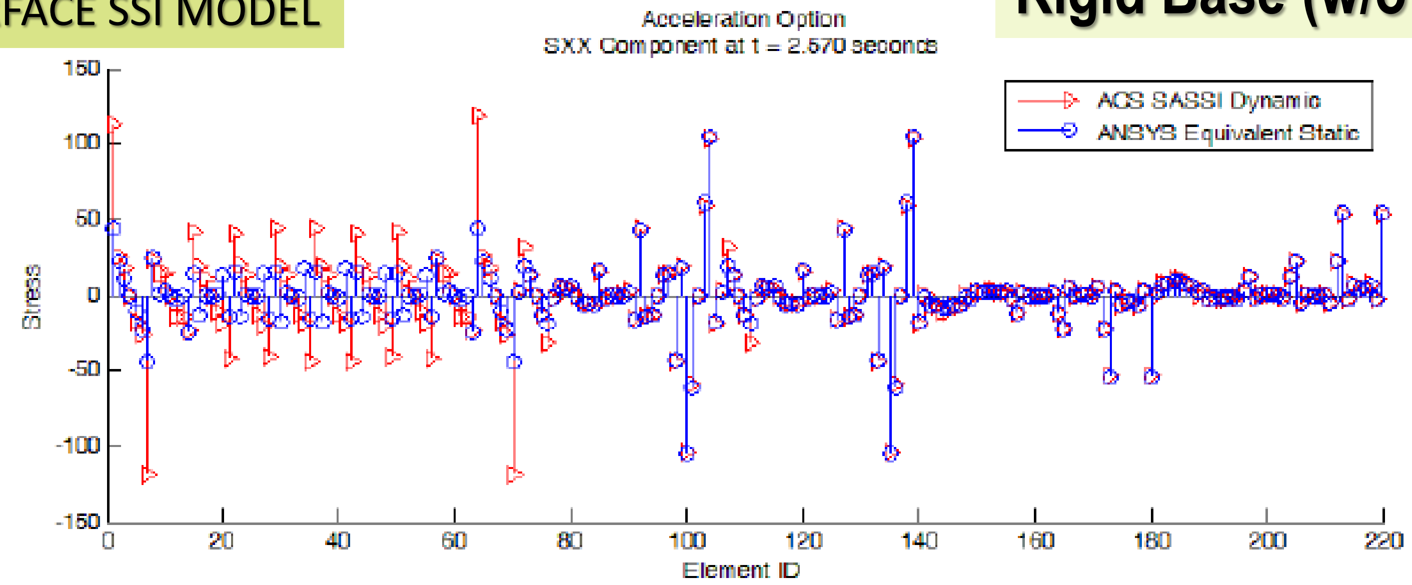


Figure 197: Case c): SXX Element Center Stresses for "Acceleration" Option

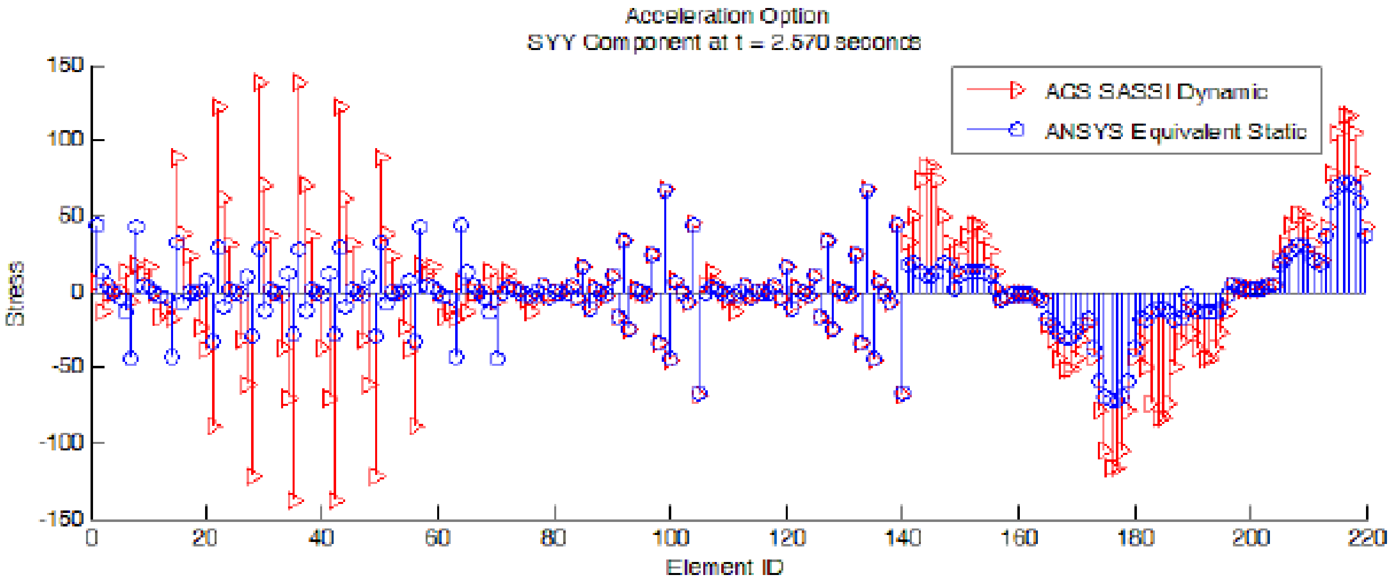
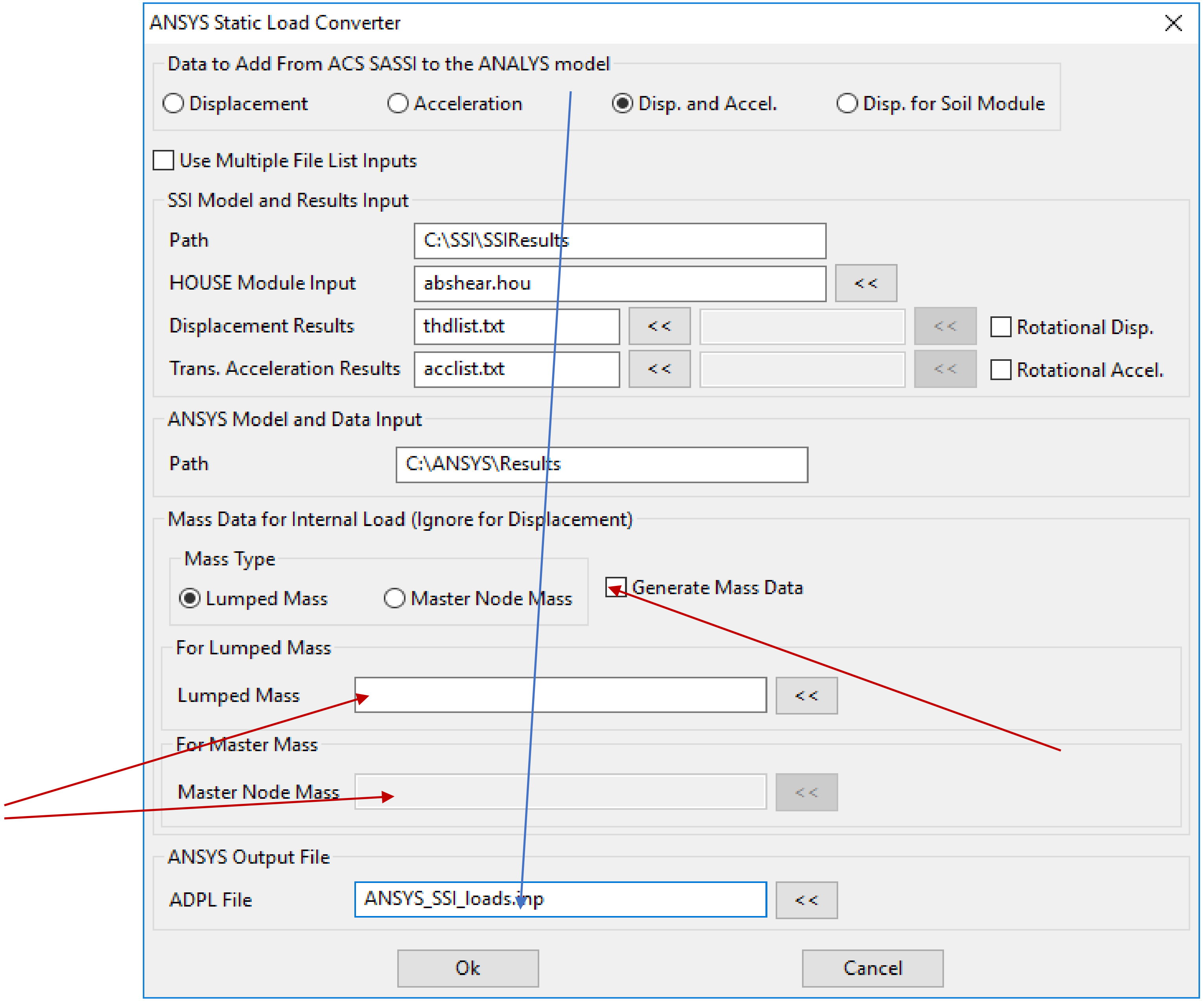


Figure 198: Case c): SYY Element Center Stresses for "Acceleration" Option

UI Input for Option A Equivalent Static Option



SOILMESH, <Model>,<Scale X>,<Scale Y>, <Hori>, <Vert>,<mX>, <mY>,<Thick>,<Contact> ,<RC num>

The SOILMESH command creates a soil FE mesh for the active model and stores the model data in the user specified Model.

<Model>- User specified integer model number.

<Scale X> - Percentage of growth in the X direction of each level, i.e. 0.07

<Scale Y> - Percentage of growth in the Y direction of each level, i.e. 0.07

<Hori> - Number of horizontal levels to build away from the embedment.

<Vert> - Number of vertical levels to build away from the embedment.

<mX> - Centroid correction in the X direction

<mY> - Centroid correction in the Y direction

<Thick> - Thickness of each new level.

<Contact> - If equal 0 for no contact surfaces, 1 for contact surfaces

<RC num> - Defines the constant set number to be used in ANSYS for the contact surface Real Constants

WARNING: It does not work for nonvertical walls.

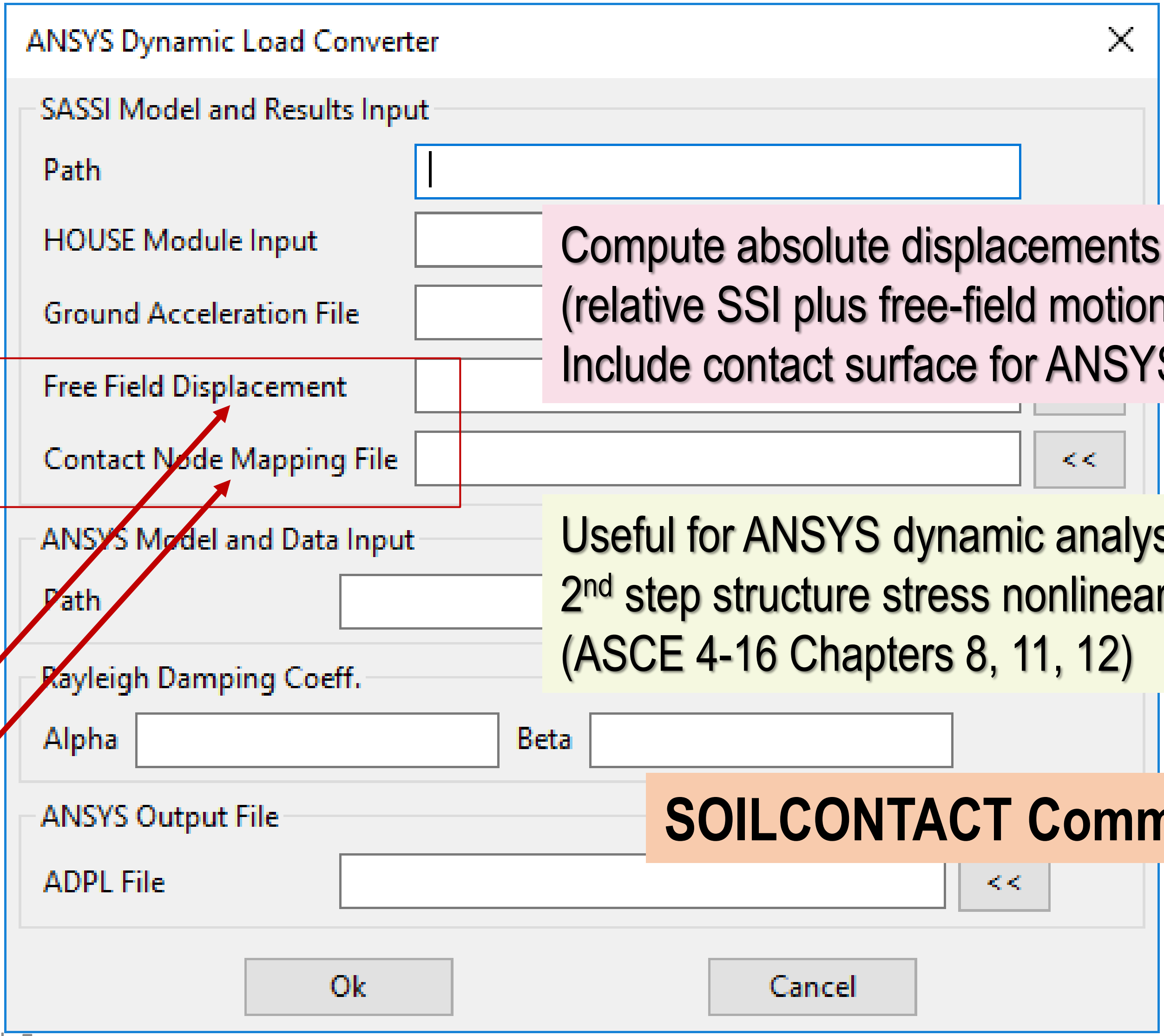
Example of ADPL file for Soil FE Model

box_soil.inp - Notepad

File Edit Format View Help

```
/PREP7
! Element Type
ET,101,CONTA173
ET,102,TARGE170
ET,103,SOLID45
! Nodes
N,145,3,3,3
N,146,3,6,3
N,147,3,13,3
N,148,3,23,3
N,149,3,33,3
N,150,3,43,3
N,151,3,53,3
N,152,3,63,3
N,153,3,73,3
N,154,3,80,3
N,155,3,83,3
N,158,6,3,3
N,159,6,6,3
N,160,6,13,3
N,161,6,23,3
N,162,6,33,3
N,163,6,43,3
N,164,6,53,3
N,165,6,63,3
N,166,6,73,3
N,167,6,80,3
N,168,6,83,3
N,171,13,3,3
N,172,13,6,3
N,173,13,13,3
N,174,13,23,3
N,175,13,33,3
N,176,13,43,3
N,177,13,53,3
N,178,13,63,3
N,179,13,73,3
N,180,13,80,3
N,181,13,83,3
N,184,23,3,3
N,185,23,6,3
N,186,23,13,3
N,187,23,23,3
N,188,23,33,3
N,189,23,43,3
N,190,23,53,3
N,191,23,63,3
N,192,23,73,3
N,193,23,80,3
```

UI Input Windows for Option A Dynamic Option



Compute absolute displacements (relative SSI plus free-field motion). Include contact surface for ANSYS.

Useful for ANSYS dynamic analysis option for 2nd step structure stress nonlinear analysis (ASCE 4-16 Chapters 8, 11, 12)

SOILCONTACT Command

OPTION AA: ACS SASSI-ANSYS Interface for SSI Analysis Using ANSYS Models (Demo 7)

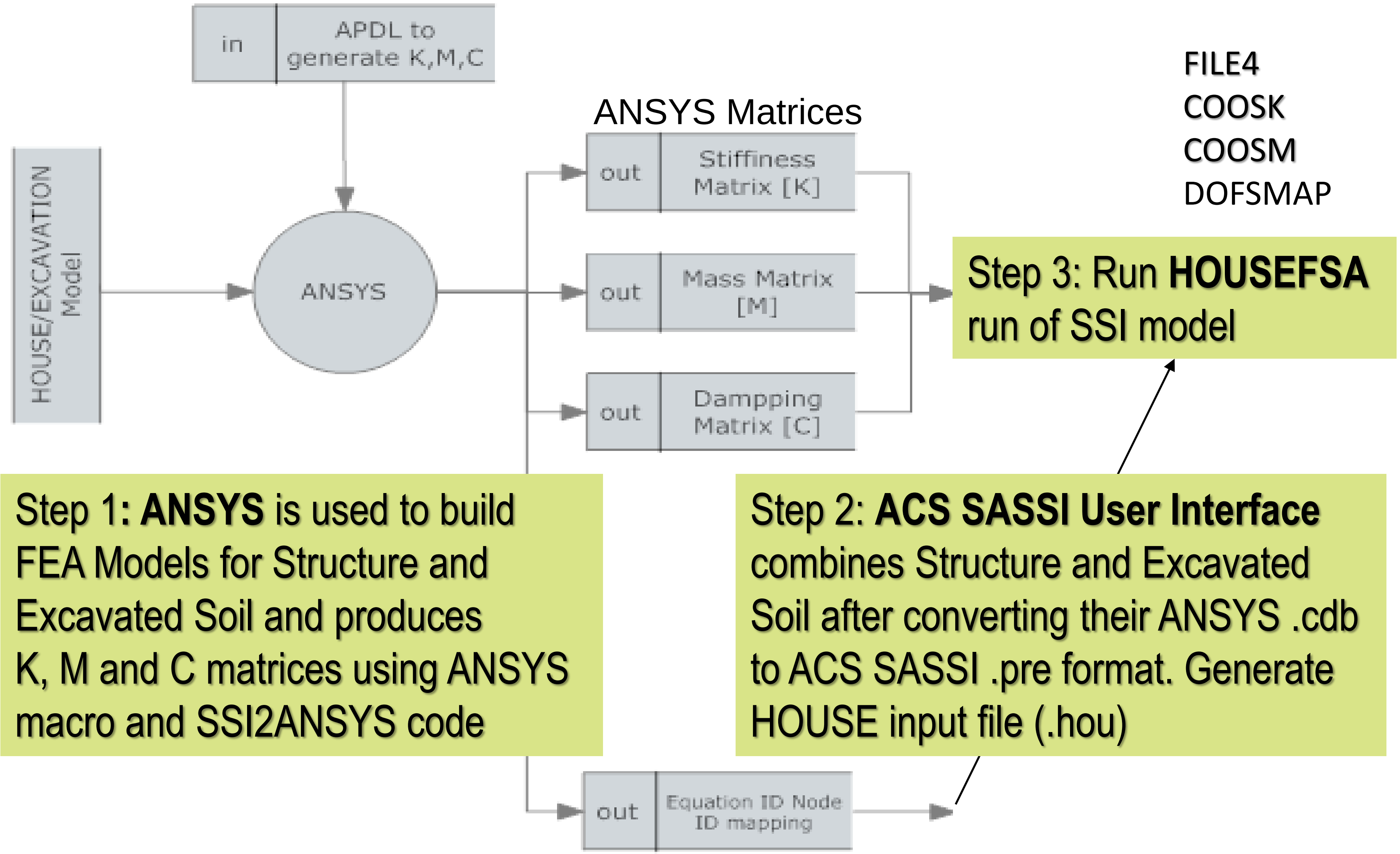
OPTION AA uses directly ANSYS structural model for SSI analysis

Sequence of Steps:

1) Develop ANSYS *structural* FEA model with no modeling restrictions (any FE type, CP, CE, rigid links)
2) If embedded, develop also the ANSYS *excavated soil* FEA model
3) Using an ANSYS ADPL macro generate matrices K, M, C
4) Using ACS SASSI UI read ANSYS model .cdb for structure and excavation to convert the ANSYS model geometry configuration to ACS SASSI for post-processing
5) Merge Structure and Excavation models using new UI. Add interaction nodes and AFWRITE the SSI model to produce HOUSE input.
6) Run SSI analysis using HOUSEFSA and ANALYSFSA

Demo 7

Steps for Running SSI analysis Using ANSYS Model



Using ANSYS with gen_kmc.mac APDL Macro

FOR STRUCTURE ANSYS Model:

At the ANSYS command line input
gen_kmc,'.',0,'.'

APDL Macro produces the following files:
coosk_r, cooski_r, coosm_r, coosmi_r, coosc_r, coosci_r, and Node2Equ_Stru.map

FOR EXCAVATION ANSYS Model:

At the ANSYS command line input
gen_kmc,'.',1,'.'

APDL Macro produces the following files:
cooek_r, cooeki_r, cooem_r, cooemi_r, cooec_r, cooeci_r, and Node2Equ_Excv.map

Using ANSYS with gen_kmc.mac APDL Macro

cooec_r	7/25/2014 1:27 PM	File
cooeci_r	7/25/2014 1:27 PM	File
cooek_r	7/25/2014 1:27 PM	File
cooeki_r	7/25/2014 1:27 PM	File
cooem_r		
cooemi_r		
coosc_r	7/25/2014 11:06 AM	File
coosci_r	7/25/2014 11:06 AM	File
coosk_r	7/25/2014 11:06 AM	File
cooski_r	7/25/2014 11:06 AM	File
coosm_r	7/25/2014 11:06 AM	File
coosmi_r	7/25/2014 11:06 AM	File
Node2Equ_Excv.map		
Node2Equ_Stru.map		
sldbox45_excv.cdb		
sldbox45_struct.cdb		

ANSYS Model K, C and M Matrix Files (APDL macro)

ANSYS Model Mapping Files (APDL macro)

ANSYS Model .cdb Files (ANSYS)

User Interface Procedure to Merge ANSYS Structure and Excavation Models for Option AA

It is assumed that the ground surface is at Z=0. and the FV method will be used

**Convert ANSYS Structure.cdb in Model 1*

Actm,1

Convert,ansys,struct.cdb,32.2

Etypegen,1

Actm,2

Convert, ansys,Soil.cdb,32.2

** Define excavation elements of type 2*

Etypegen,2

** Create SSI model by combining Models 1 and 2 in Model 3*

Actm,3

MergeSoil,1,2,1,,,,mappingfile.txt

Groundelev, 0

Intgen, 1

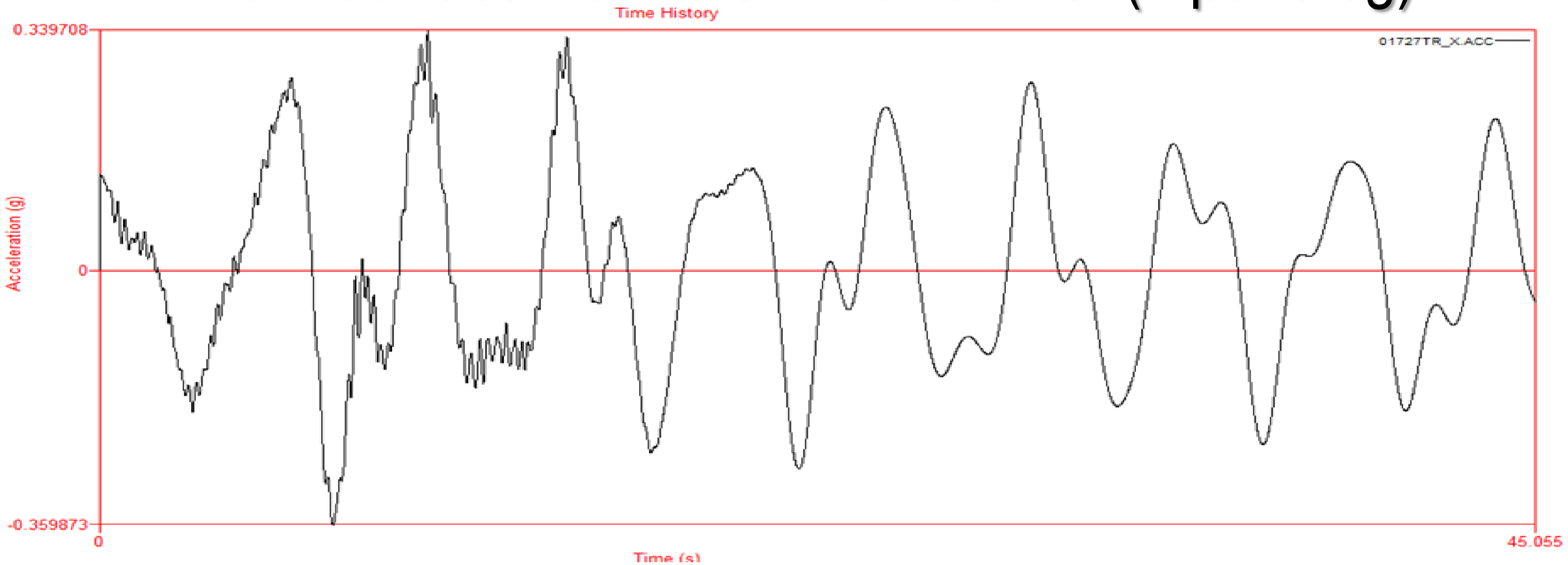
ANSYS FE Types Compatible with Option AA

- SOLID element types: SOLID45 and SOLID185;
- SHELL element types: SHELL63 and SHELL181;
- BEAM element types: BEAM44 and BEAM188;
- PIPE element types: PIPE288;
- COMBIN element types: COMBIN14;
- Couple nodes (CP command) and Constraint equations (CE command)
- Multipoint constraint element types: MPC184 Rigid Link and Rigid Beam
- Fluid element types: FLUID80 (legacy element).

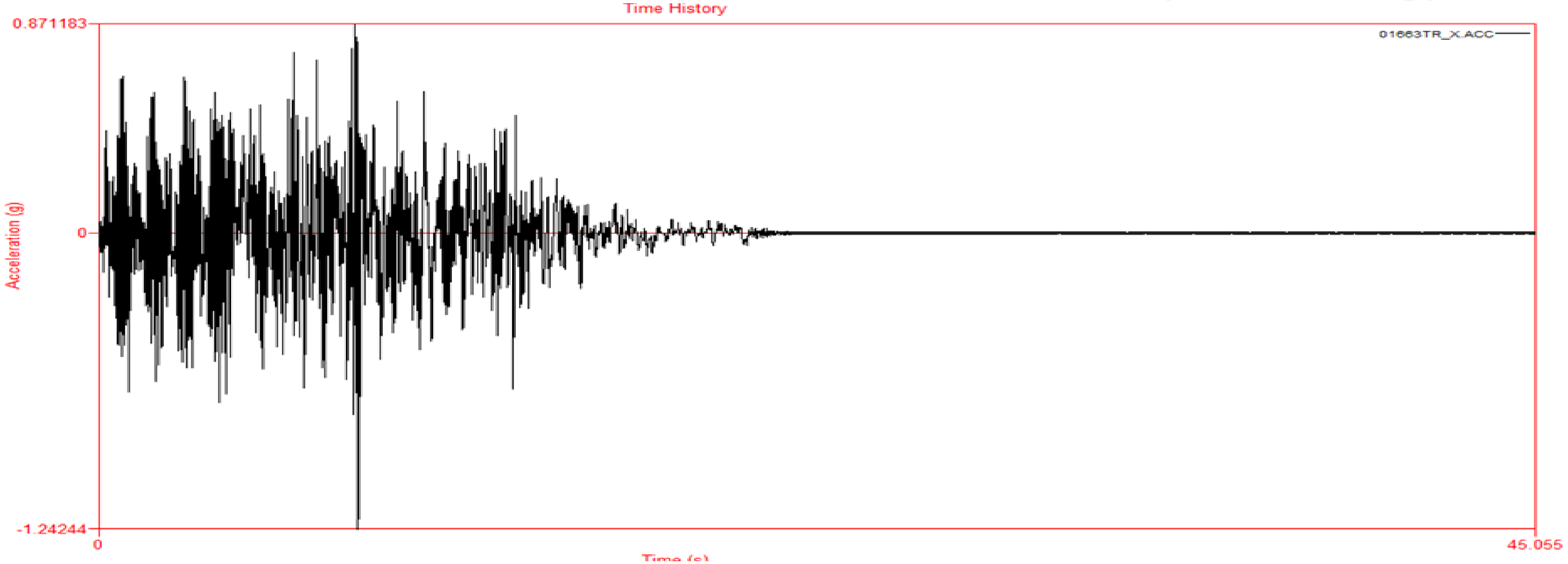
- MATRIX50 Super Element
  - Included in Option AA using ANSYS model
  - Converted to General Matrix Element for the ACS SASSI Model

REMARK: Not all keyopt or othere parameter values work!

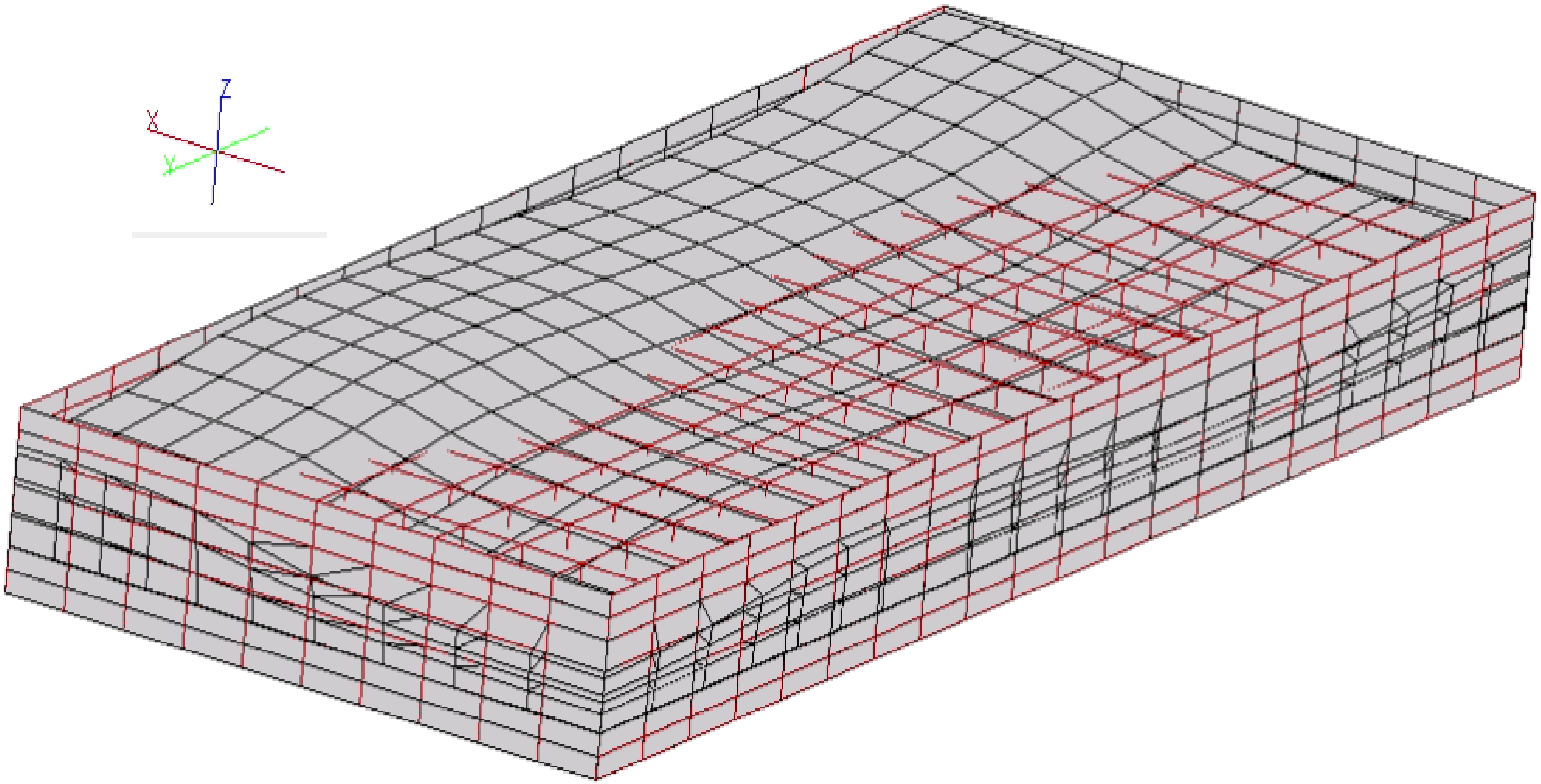
Fluid Surface Acceleration at Center (Input 0.3g)



Wall Transverse Acceleration at Center (Input 0.3g)



Pool Water Wave Displacement Response



OPTION AA: ACS SASSI-ANSYS Interface for SSI Analysis Using ANSYS Models (Demo 7)

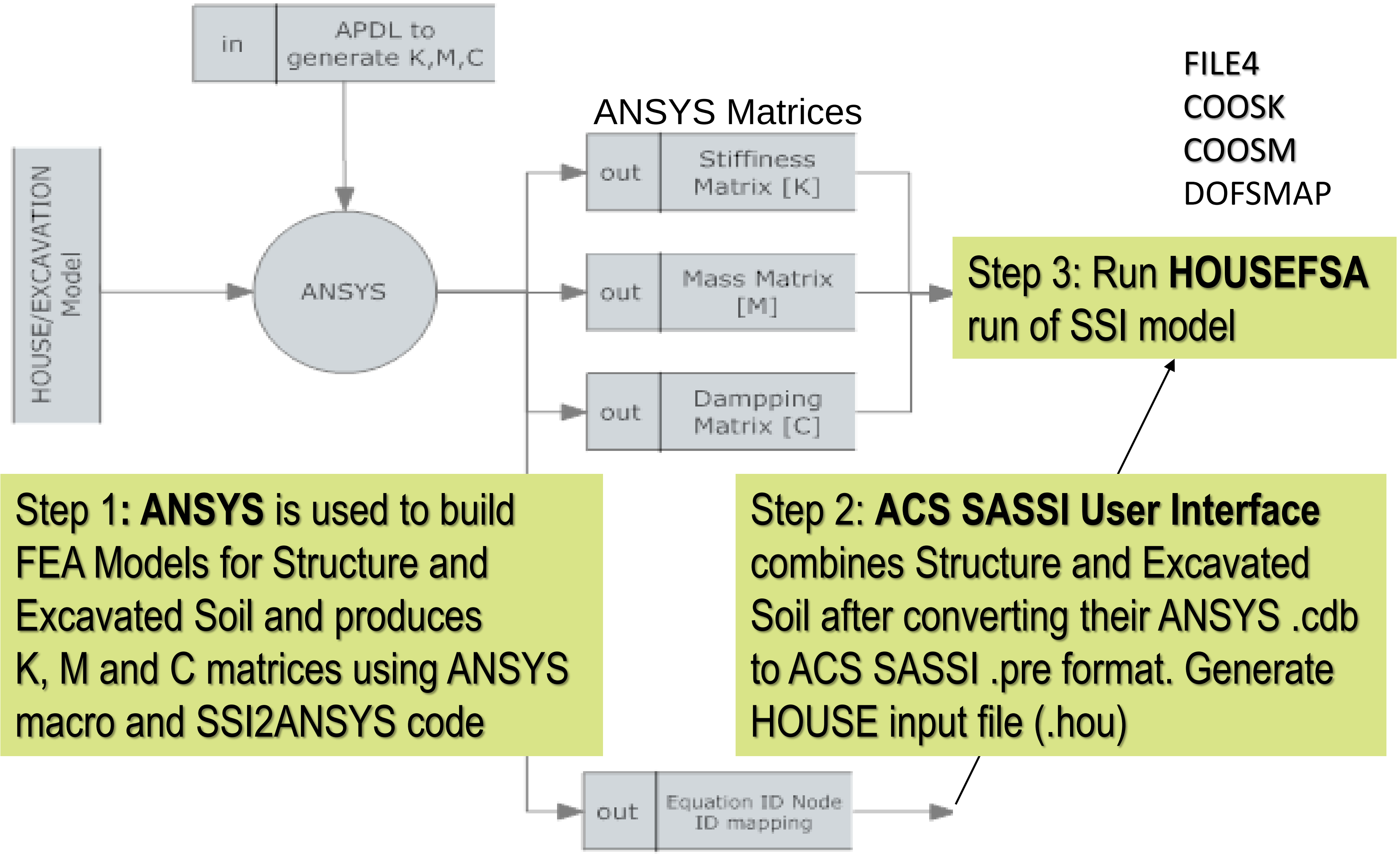
OPTION AA uses directly ANSYS structural model for SSI analysis

Sequence of Steps:

1) Develop ANSYS *structural* FEA model with no modeling restrictions (any FE type, CP, CE, rigid links)
2) If embedded, develop also the ANSYS *excavated soil* FEA model
3) Using an ANSYS ADPL macro generate matrices K, M, C
4) Using ACS SASSI UI read ANSYS model .cdb for structure and excavation to convert the ANSYS model geometry configuration to ACS SASSI for post-processing
5) Merge Structure and Excavation models using new UI. Add interaction nodes and AFWRITE the SSI model to produce HOUSE input.
6) Run SSI analysis using HOUSEFSA and ANALYSFSA

Demo 7

Steps for Running SSI analysis Using ANSYS Model



Using ANSYS with gen_kmc.mac APDL Macro

FOR STRUCTURE ANSYS Model:

At the ANSYS command line input
gen_kmc,'.',0,'.'

APDL Macro produces the following files:
coosk_r, cooski_r, coosm_r, coosmi_r, coosc_r, coosci_r, and Node2Equ_Stru.map

FOR EXCAVATION ANSYS Model:

At the ANSYS command line input
gen_kmc,'.',1,'.'

APDL Macro produces the following files:
cooek_r, cooeki_r, cooem_r, cooemi_r, cooec_r, cooeci_r, and Node2Equ_Excv.map

User Interface Procedure to Merge ANSYS Structure and Excavation Models for Option AA

It is assumed that the ground surface is at Z=0. and the FV method will be used

```
*Convert  ANSYS Structure.cdb in Model 1
Actm,1
Convert,ansys,struct.cdb,32.2
Etypegen,1
Actm,2
Convert, ansys,Soil.cdb,32.2
* Define excavation elements of type 2
Etypegen,2
* Create SSI model by combining Models 1 and 2 in Model 3
Actm,3
MergeSoil,1,2,1,,,,mappingfile.txt
Groundelev, 0
Intgen, 1
```

ANSYS FE Types Compatible with Option AA

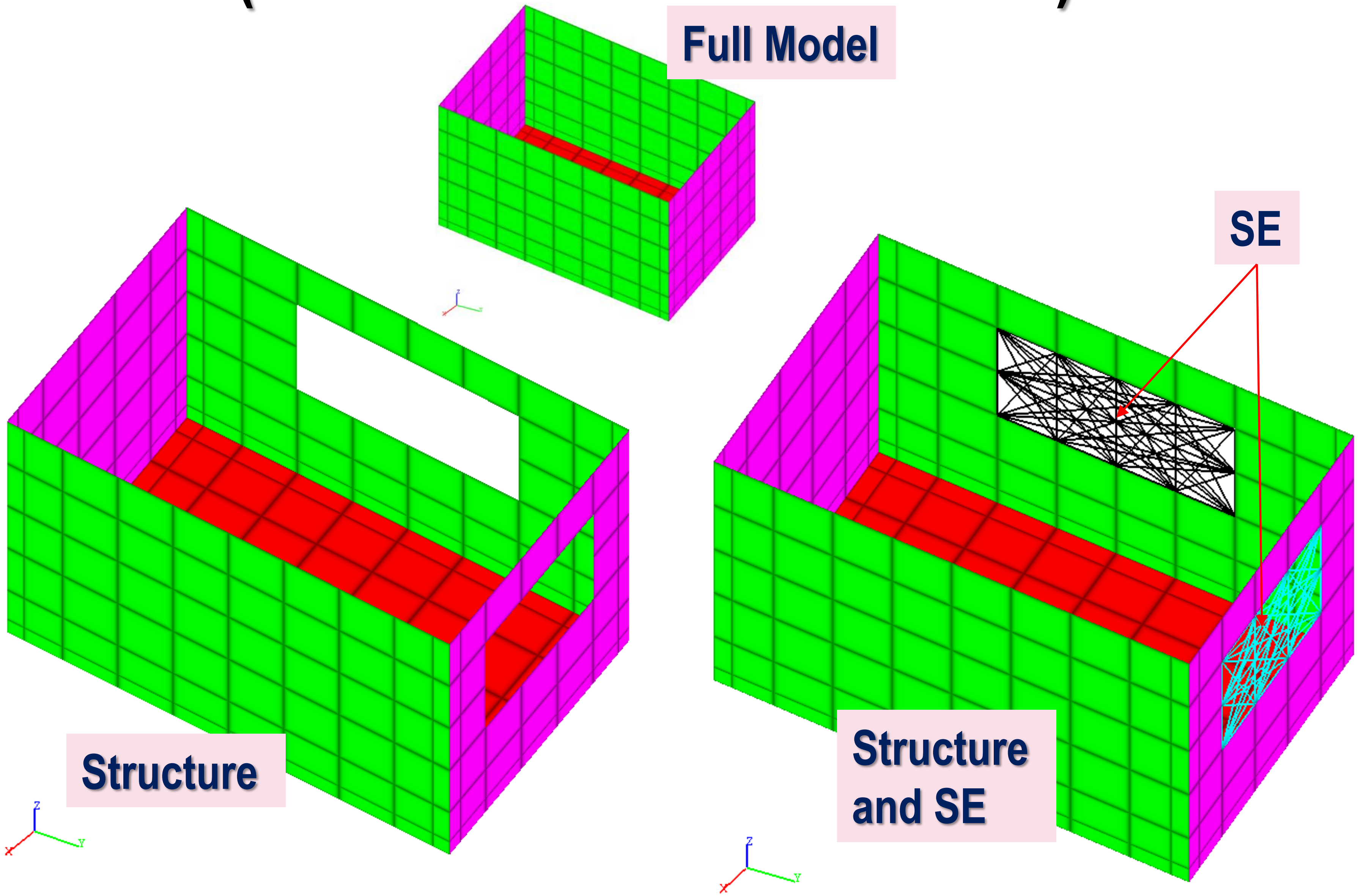
- SOLID element types: SOLID45 and SOLID185;
- SHELL element types: SHELL63 and SHELL181;
- BEAM element types: BEAM44 and BEAM188;
- PIPE element types: PIPE288;
- COMBIN element types: COMBIN14;
- Couple nodes (CP command) and Constraint equations (CE command)
- Multipoint constraint element types: MPC184 Rigid Link and Rigid Beam
- Fluid element types: FLUID80 (legacy element*).*

FLUID30 can be included using the new Option AA-REDUCE or AA-R using a condensed excavated soil stiffness matrix (NuScale SSI methodology for Fluid-SSI analysis)

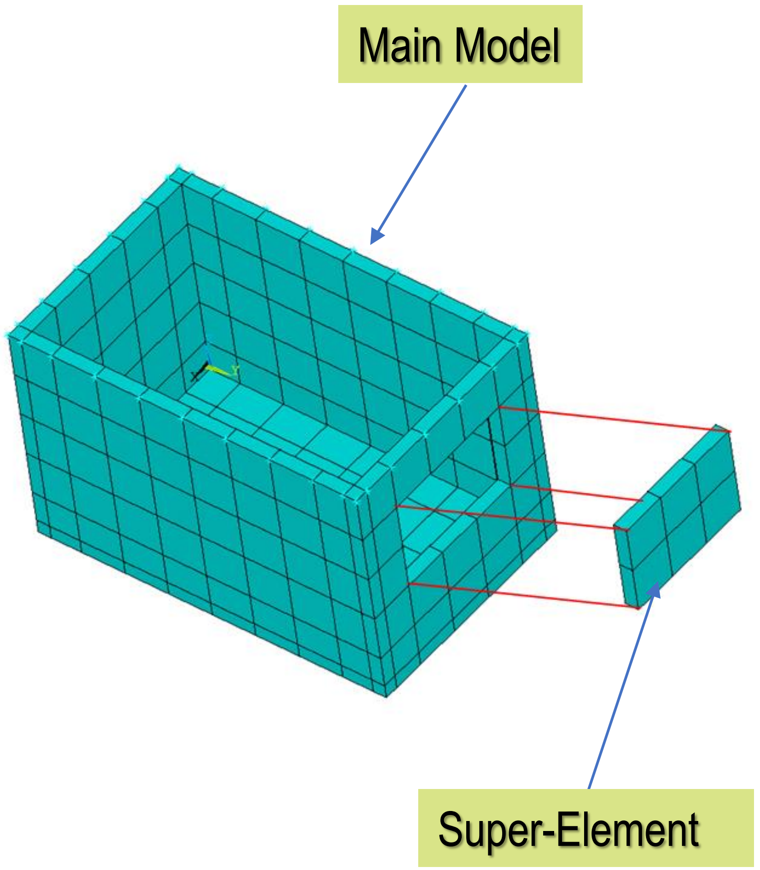
- MATRIX50 Super Element
  - Included in Option AA using ANSYS model
  - Converted to General Matrix Element for the ACS SASSI Model

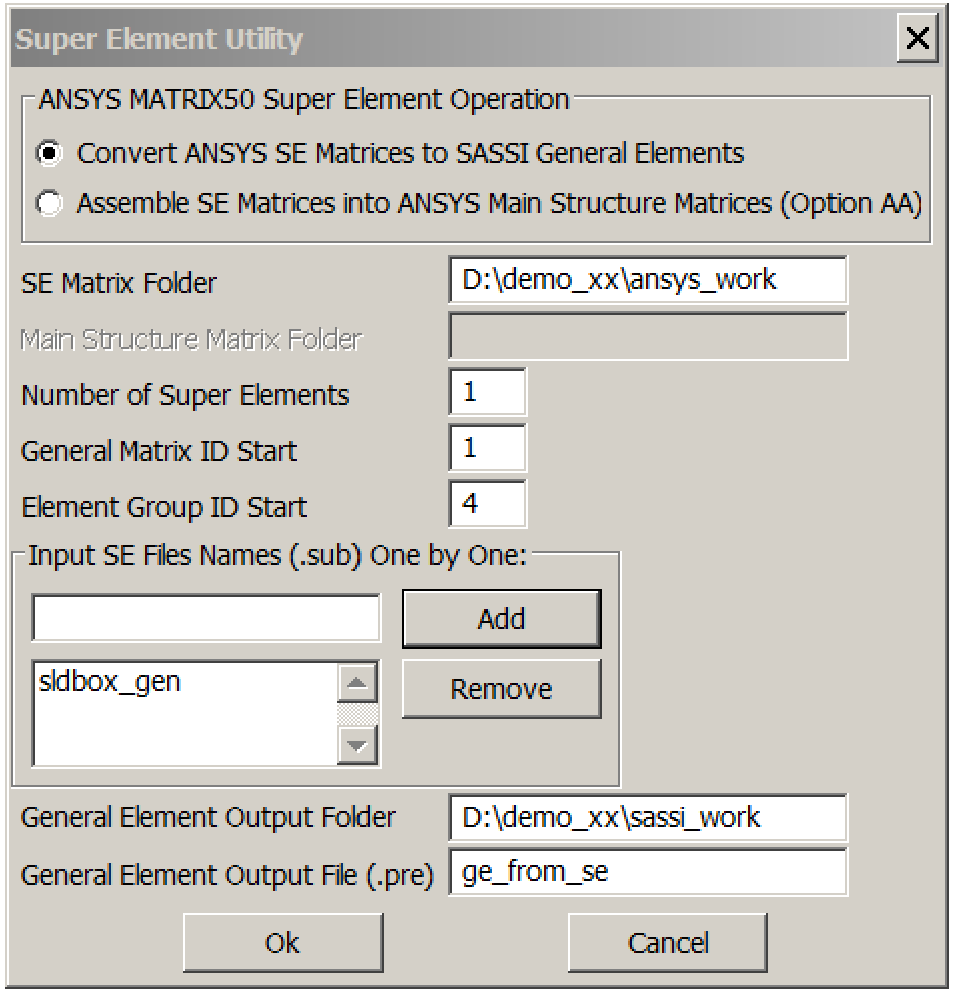
REMARK: Not all ANSYS keyopts or all input parameter values work!

Using ANSYS MATRIX50 to Build Super Elements (see Demo 11 and V&V Pb 52)

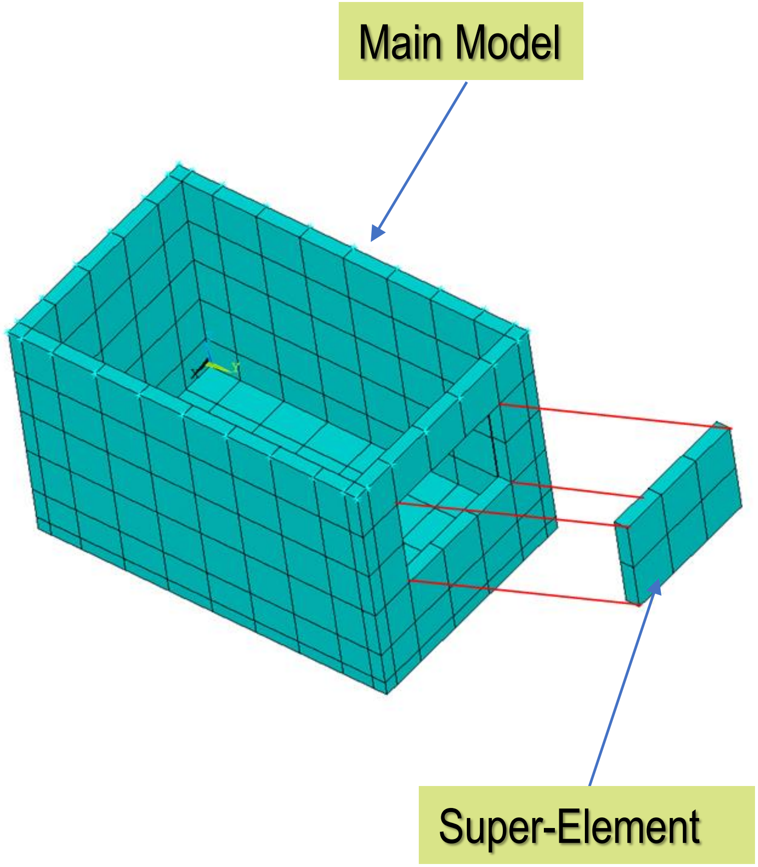


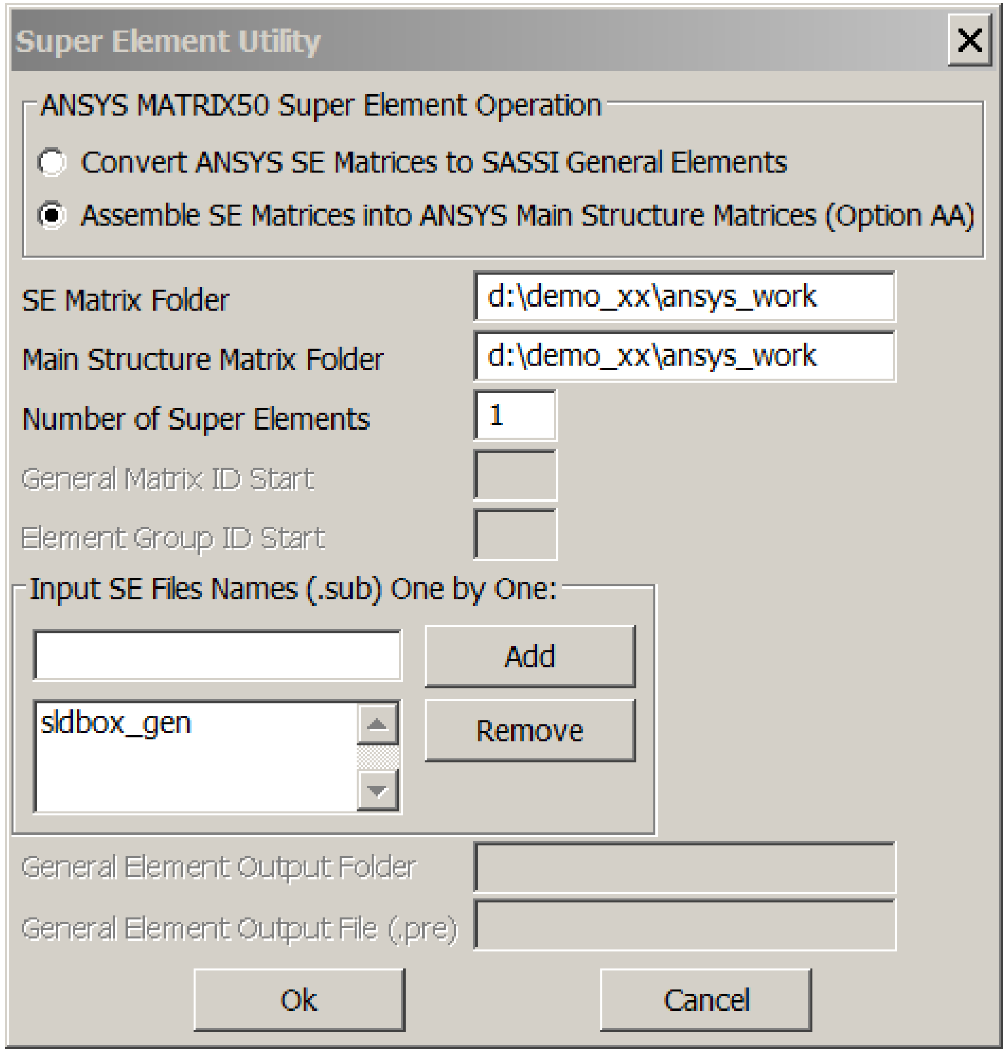
ANSYS Super Element (SE) Converted to ACS SASSI Using General Matrix Elements (GM)





ANSYS Super Element (SE) Using Option AA By Adding Main Model and SE Model Matrices





Option AA-REDUCE or AA-R

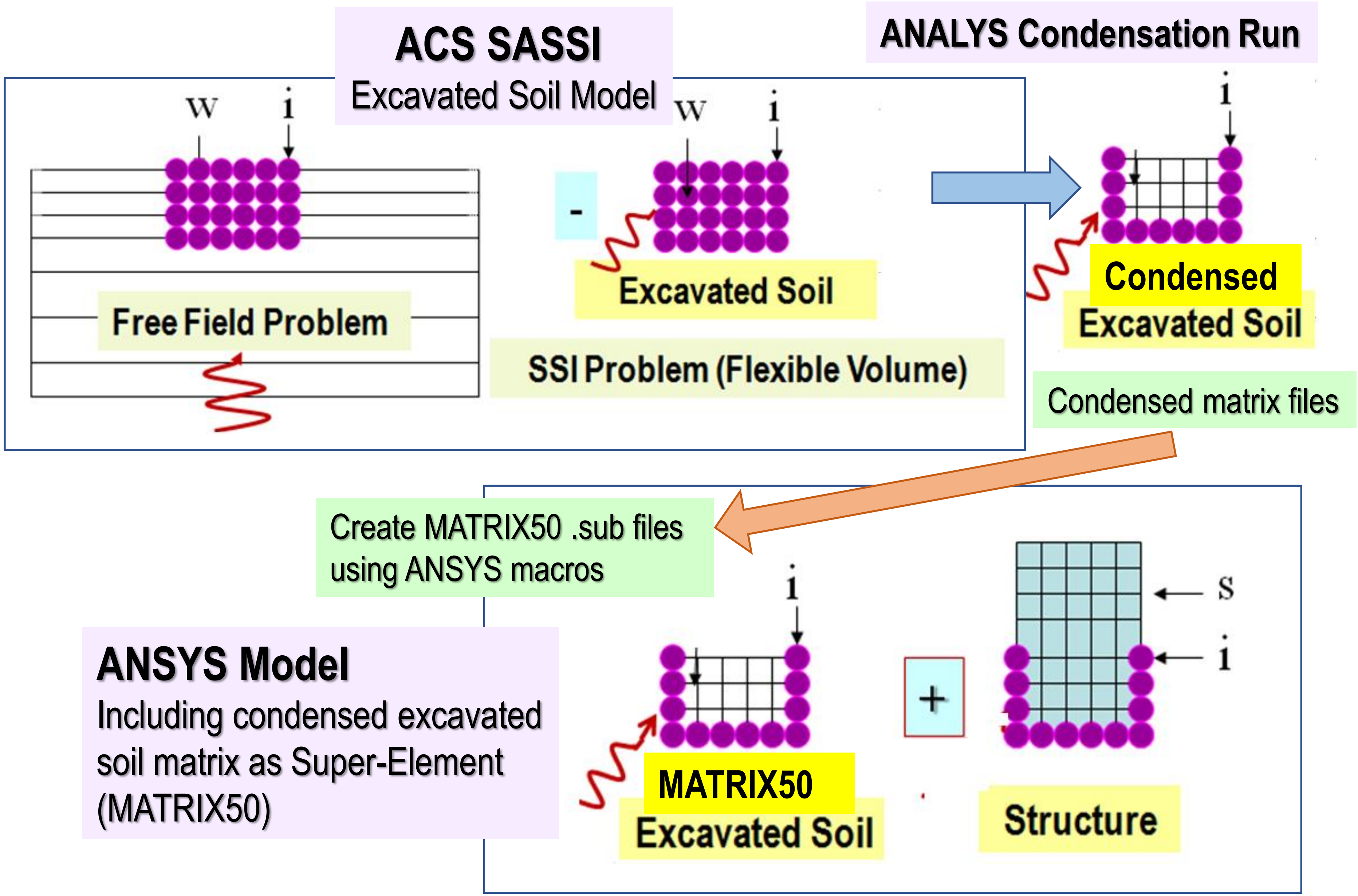
Option AA-R Capability Description

The Option AA-R largely extends the Option AA capability of using directly ANSYS structure FE models for seismic SSI analysis. The Option AA-R ANSYS models are applicable only to the ANSYS Harmonic SSI analysis in complex frequency.

To make Option AA-R more practical a reduced-size soil impedance matrix and a reduced-size seismic load vector are used, based on the condensation of the excavated soil impedance full matrix done in the ANALYS module. There is no loss of accuracy in SSI solution by using the reduced soil matrix and reduced load vector.

Further the frequency-dependent condensed excavated soil matrix and load vector produced by ANALYS can be exported to ANSYS as a super-element (MATRIX50 element) that is attached to the ANSYS structure FE model.

Option AA-R Flowchart for ANSYS SSI Analysis



Excavated Soil Impedance Condensation

For embedded models, the ANALYS module includes a new option to condense the excavated soil impedance matrix. This static condensation is performed for the excavated soil dynamic stiffness matrix $\mathbf{Z}(\omega)$ at each SSI frequency. Then, the SSI response is obtained using the reduced SSI system at each SSI frequency.

Currently the condensed matrix is transferred to ANSYS as a frequency-dependent super element (MATRIX50) via Option AA-R.

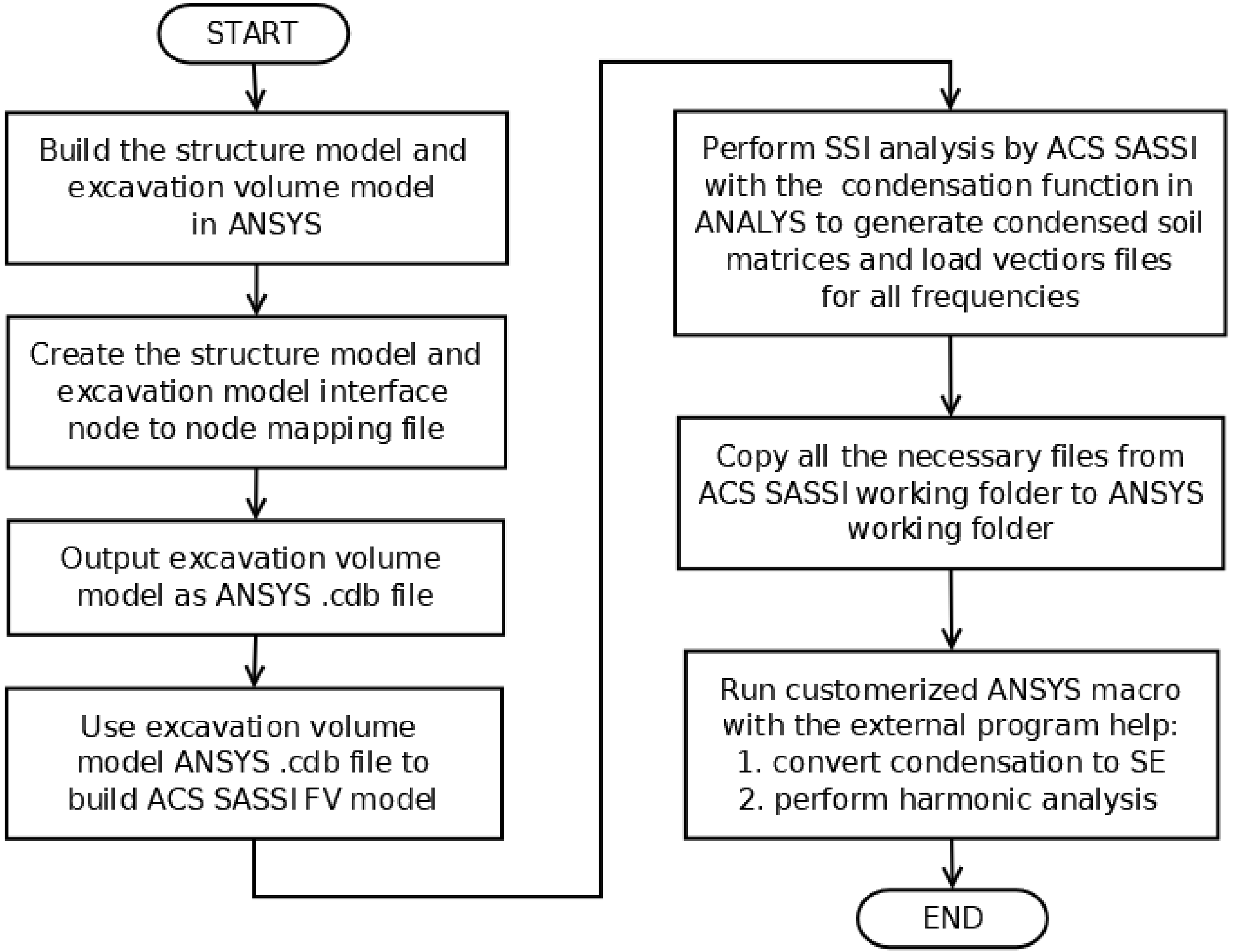
The condensed equations of SSI system motion include only the foundation-soil interface nodes, or more generally, the excavated soil model-structure model interface :

$$\begin{cases} \left([C_{ii}^{s}] + \tilde{Z}_{ii}\right)\{U_i\} + [\,C_{is}^{s}]\{U_s\} = \{\tilde{F}_i\} \\ \quad [C_{si}^{s}\,]\{U_i\} + [\,C_{ss}^{s}]\{U_s\} = \{0\} \end{cases}$$

ANSYS SSI Harmonic Analysis Via Option AA-R.
ACS SASSI Condensed Soil Matrix Passed to ANSYS

The ACS SASSI condensed excavated soil impedance matrix passed as a super-element (SE) to ANSYS that is automatically integrated with ANSYS structure model.

ANSYS or ACS SASSI to prepare excavation FE model for ACS SASSI condensation run



ACS SASSI excavation impedance condensation run

ANSYS SSI harmonic analysis run using prepared macros

ACS SASSI ANALYS Run Option for Excavation Soil Impedance Condensation

- **Step 1**: Build the excavated soil model in ACS SASSI or ANSYS. This model should be mesh compatible with the ANSYS structure model to be used via Option AA-R;
- **Step 2**: Prepare the SSI analysis inputs for ONLY excavated soil model
- **Step 3**: Generate the nodes mapping file, INT_NODES_IF, between the interface nodes of excavation and structure models; see CONDMAP
- **Step 4**: Run ACS SASSI SSI Modules as follows.

  SITE Module ➔ POINT Module ➔ HOUSE Module ➔ ANALYS with "Condense Impedance" Option (Mode=7)

  After ANALYS completes the condensation matrices and load vectors are generated in the work folder, IMP_EXCVxxx and IMP_EXCV_xxx, plus a equation mapping file DOFSMAP_IMP_EXCV ,
- **Step 5**: Copy the condensation files to the ANSYS work folder for performing ANSYS SSI harmonic analysis later. Next ANSYS will need to be run using the prepared macros installed in \ANSYS\ installation folder

ANSYS SSI Harmonic Analysis Using Condensed soil Impedance and Seismic Load Vectors

- **Step 1**: Make sure your structure model in ANSYS is met the requirements for SSI analysis, pay special attention to define the correct material damping for the model (differences in ANSYS V17 and V19);
- **Step 2**: Prepare a text file to define the harmonic frequencies for SSI, which would be same frequencies in ACS SASSI analysis;
- **Step 3**: Run ANSYS, enter the macro "***do_cdns_ssi***" with right parameters. This macro will call other macros to perform the SSI harmonic analysis using the given frequency data, condensation matrices, and load vectors. The macro does automatically the tasks:.

  1> Add the super-element (MATRIX50) to ANSYS structure model;

  2> Generate super-element file .SUB using the first frequency condensation matrix using "***prep_se.exe***" and node mapping information.

  3> Do full SSI harmonic analysis for first frequency;

  4> Loop over the other frequency doing harmonic analysis;
- **Step 4:** Post-processing SSI analysis results (generating FILE8)

Option AA-R Files for ANSYS SSI Analysis

The Option AA-R files required for the ANSYS Runs are:

prep_se.exe	Application
SSI2ANSYS.exe	Application
do_cdns_ssi.mac	MAC File
do_condense_hrm.mac	MAC File
fread_data.mac	MAC File
gen_condense_se.mac	MAC File
use_condense_se.mac	MAC File

To run the *ANSYS SSI harmonic analysis*, the user needs ONLY to input in the ANSYS command line the name of the ***do_cdns_ssi*** macro:

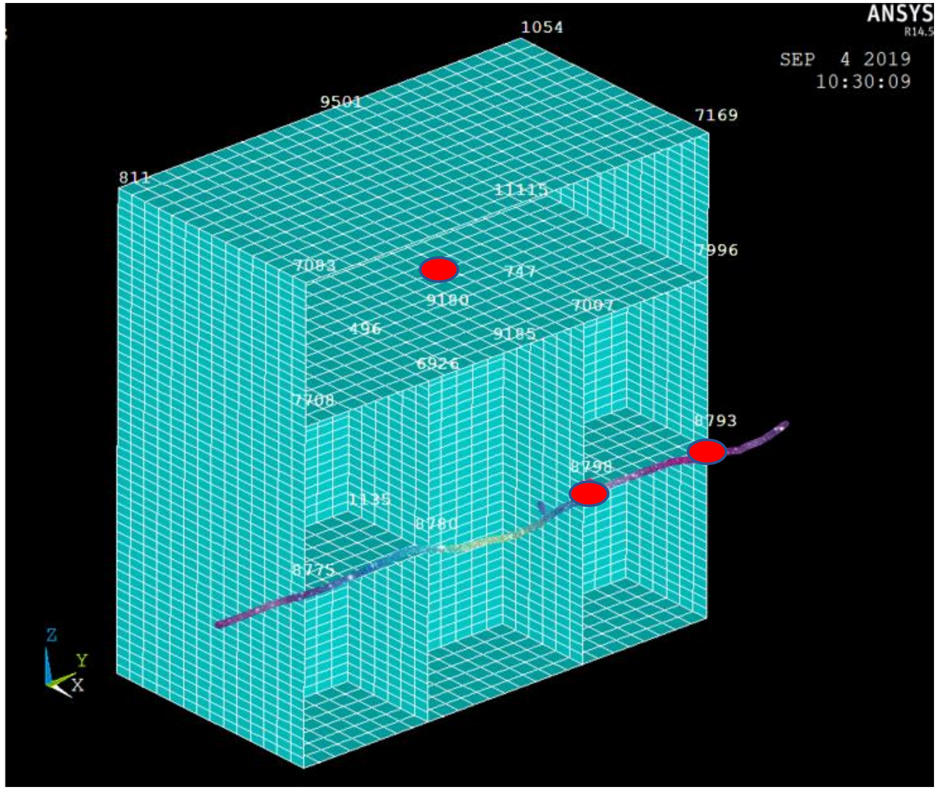
do_cdns_ssi,'structure_modelname,'ssi_freqs','txt'

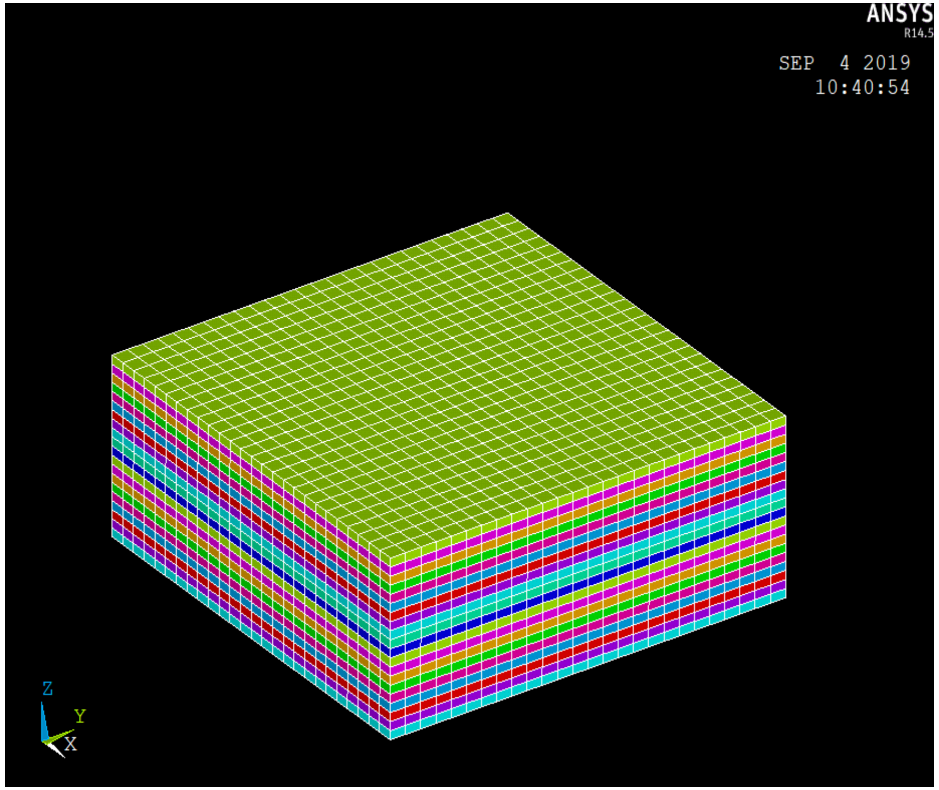
where the ***ssi_freqs.txt*** file includes the SSI analysis frequencies

ANSYS *do_cdns_ssi* Macro

```
! do_cdns_ssi
!*******************************************************************************
!*   do_cdns_ssi  Start                                                         *
!*******************************************************************************
!*##############################################################################
!    do_cdns_ssi call to few MACROs to perform SSI analysis in ANSYS using
!       harmonic analysis.
!       First, it calls MACRO of "gen_condense_se" to generate super element file
!       for for first frequency.
!       Then, it calls MACRO of "use_condense_se" to create SE data for first
!       Last, it calls MACRO of "do_condense_hrm" to do harmonic analysis frequency
!       by frequency in the order that is set in the frequency data file.
!****---------------------------------------------------------------------------
!    Pre-condition:
!
!****---------------------------------------------------------------------------
!    Call the Macro with the right arguments
!
!    do_cdns_ssi,arg1,arg2,arg3,
!
!    ARG1 [chr,sc,in] = ANSYS structure data base file name (.db), i.e.
!                       'rb_test_stru'
!    ARG2 [chr,sc,in] = The file name of the frequency for SSI analysis, i.e.
!                       'rb_test_ssi_freq'
!    ARG3 [chr,sc,in] = file extension name of the frequency for SSI analysis, i.e.
!                       'txt'
!
!    EXAMPLE: do_cdns_ssi,'rb_test_stru','rb_test_ssi_freq','txt',
!
!    Result will be save in ARG6 & ARG7 components
! ##############################################################################
!
gen_condense_se,ARG1,'se_dof','job_name'
use_condense_se,job_name,
ARG72 = ARG2
ARG73 = ARG3
do_condense_hrm,job_name,'SE_DOF',ARG72,ARG73,
```

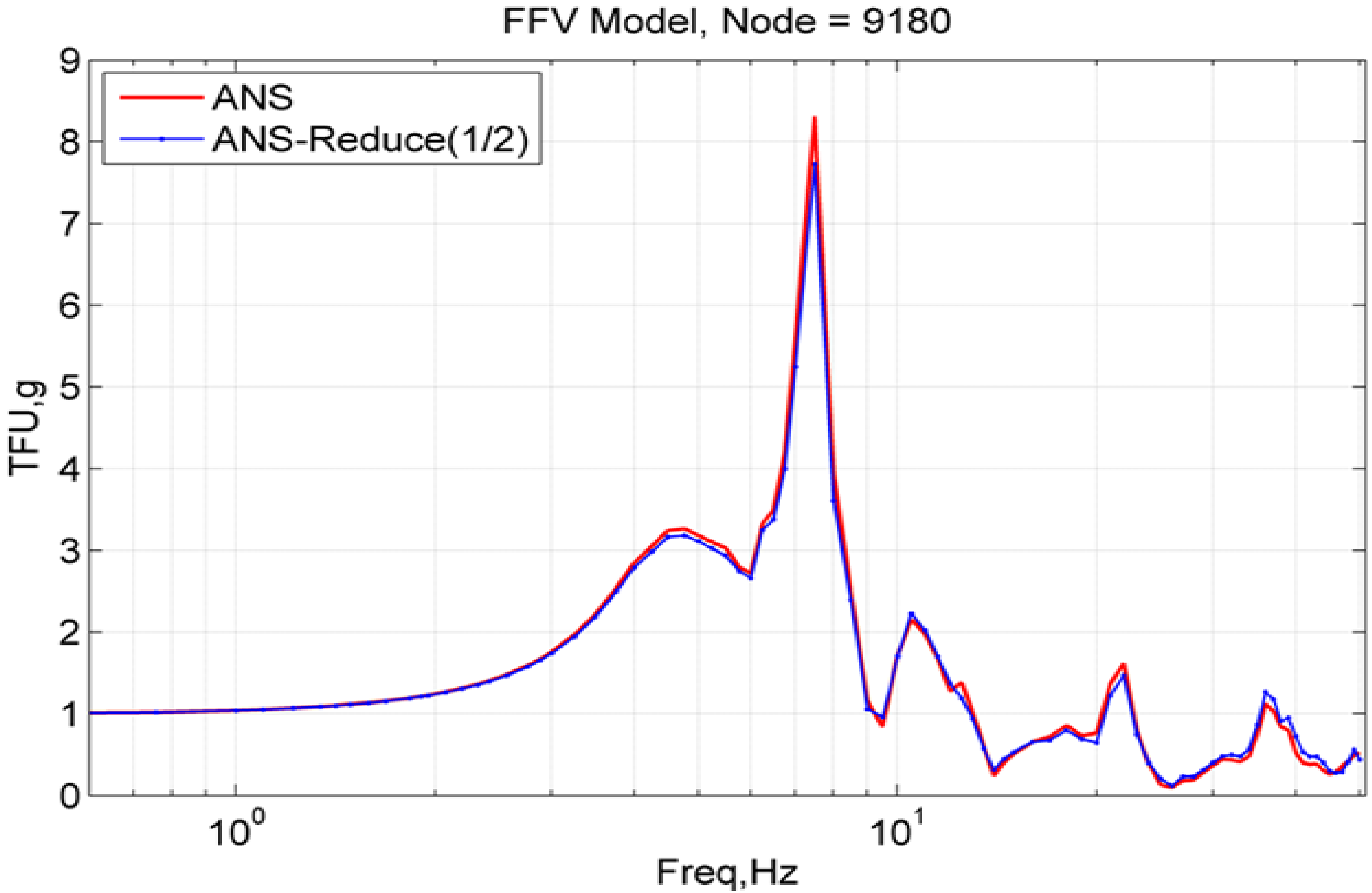
Option AA-R Sensitivity Studies for Embedded RB



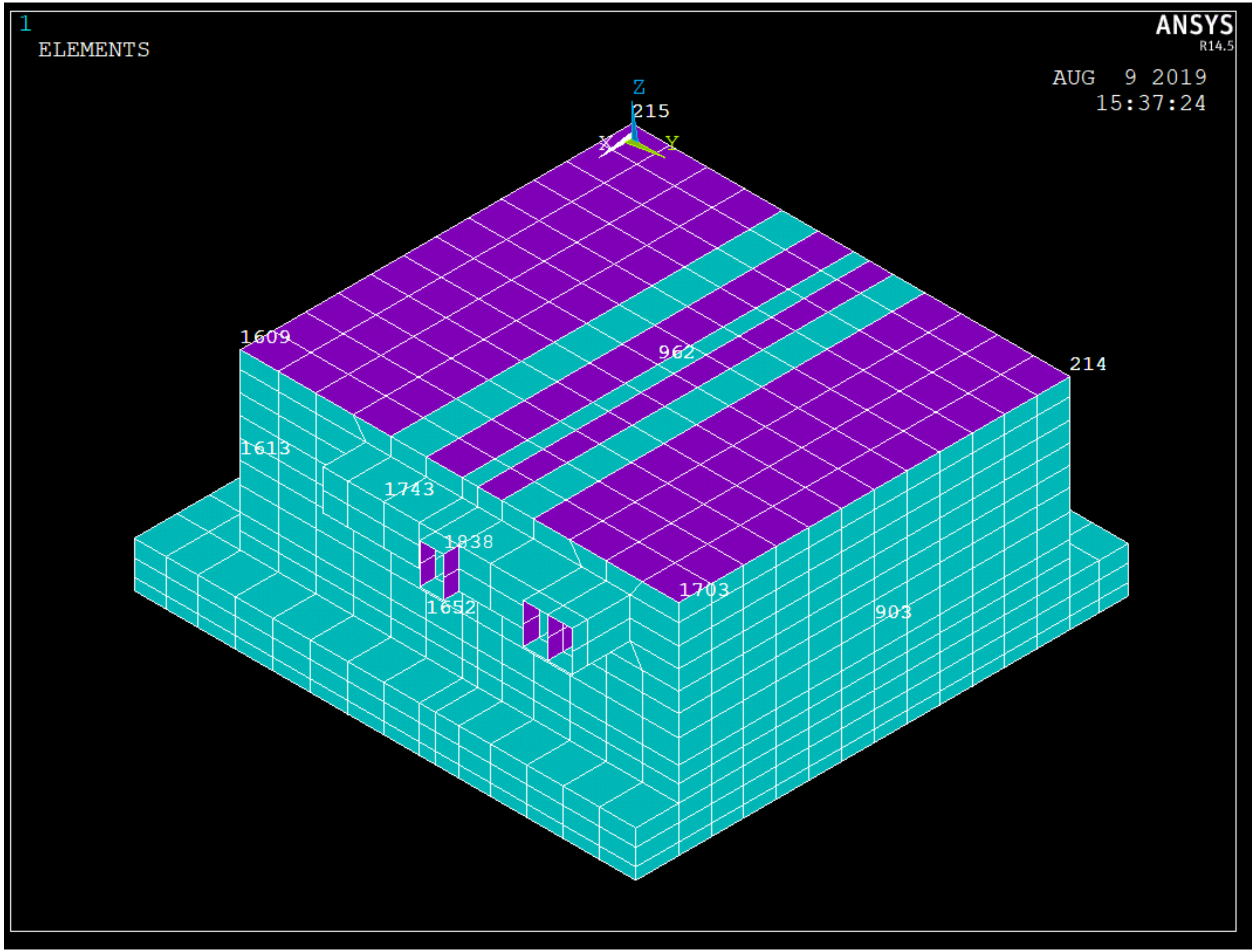


Remarks:
- Embedded RB SSI Model has 11195 nodes and 11756 SHELL elements
- Excavated soil includes 20 Embedment Layers (1.2m thickness), 15309 nodes
- Uniform soil with Vs = 720m/s
- Excavation has a regular mesh

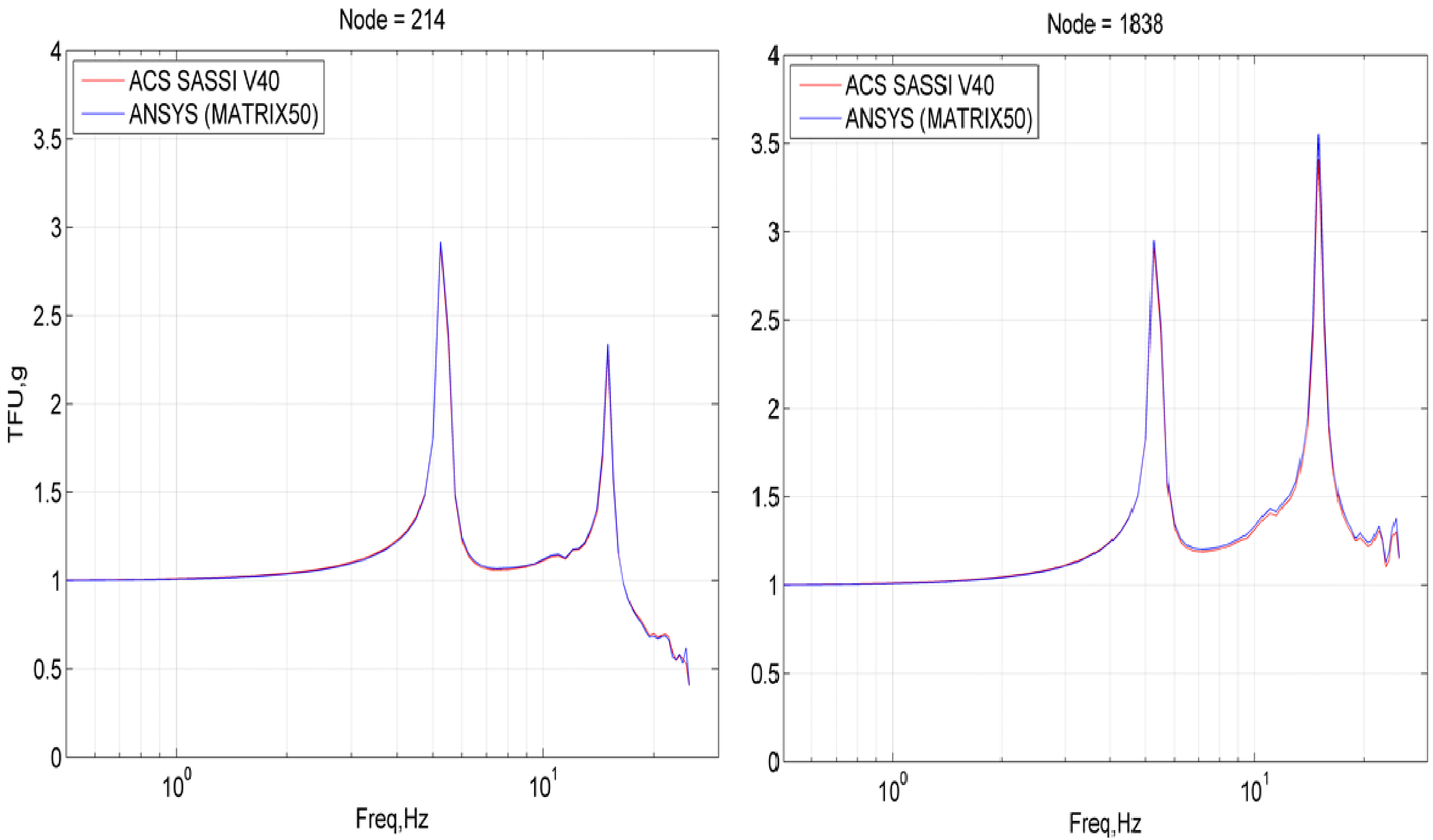
ATF for Condensed Excavation ANSYS Using All and Only Half Interface Nodes for FFV (with REDUCE Command)



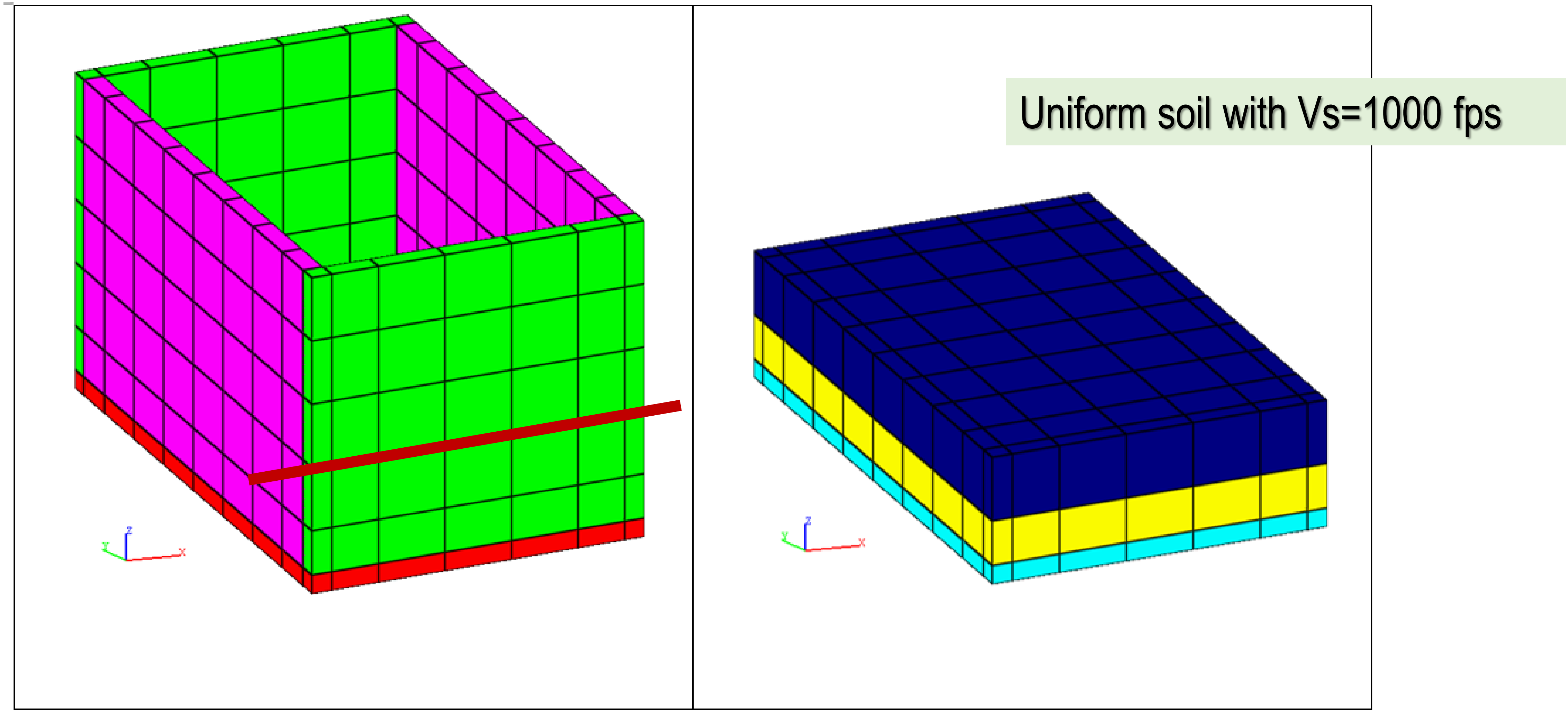
Fully Embedded Structure SSI Model



ATF for Condensed Excavation ANSYS vs. ACS SASSI Using MSM at Higher Elevations

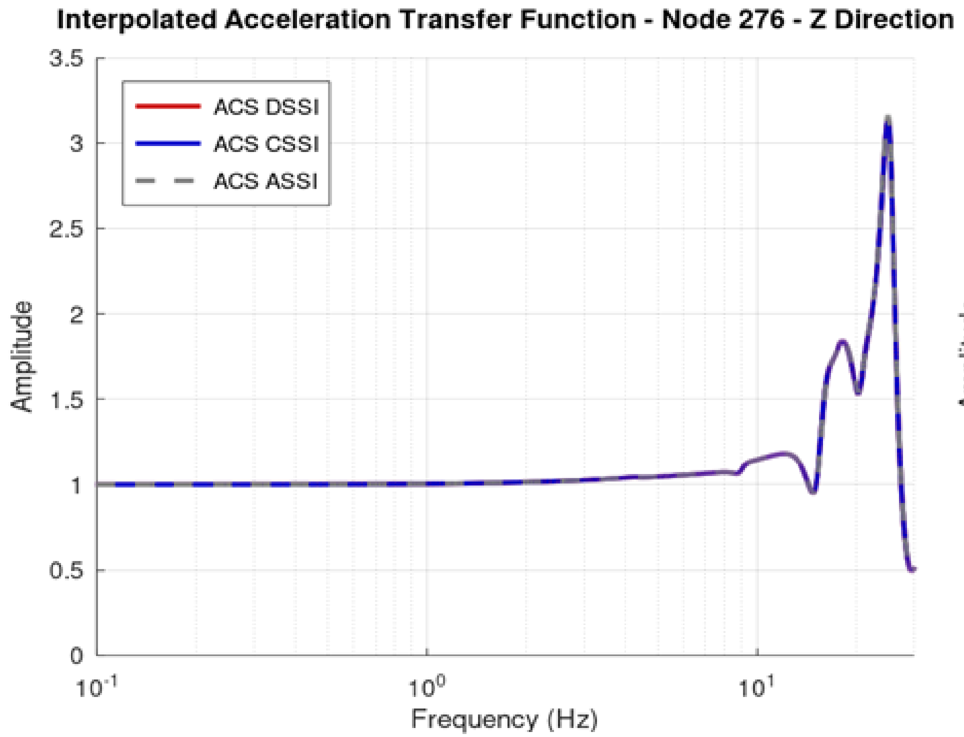


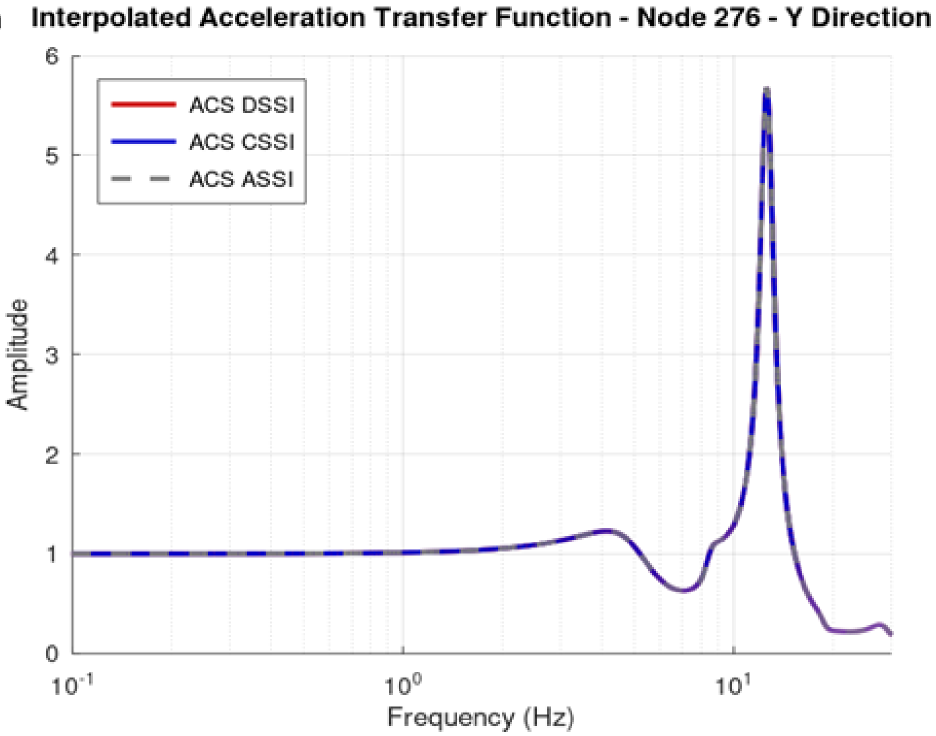
Validation of Option AA-R for ANSYS SSI Harmonic Analysis Using Condensed Soil Impedance, V&V Problem 59

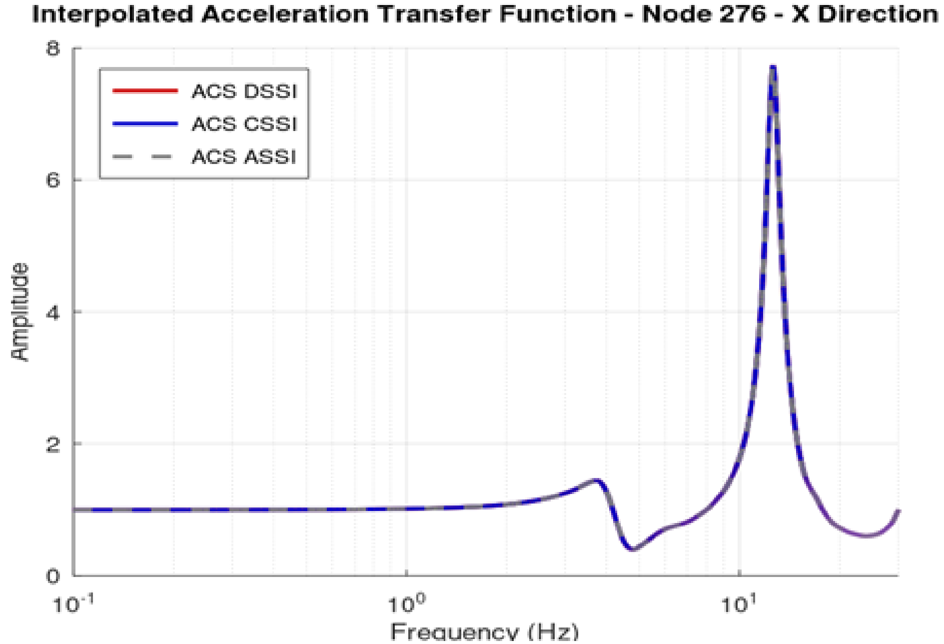


Structure Model

Excavation Model







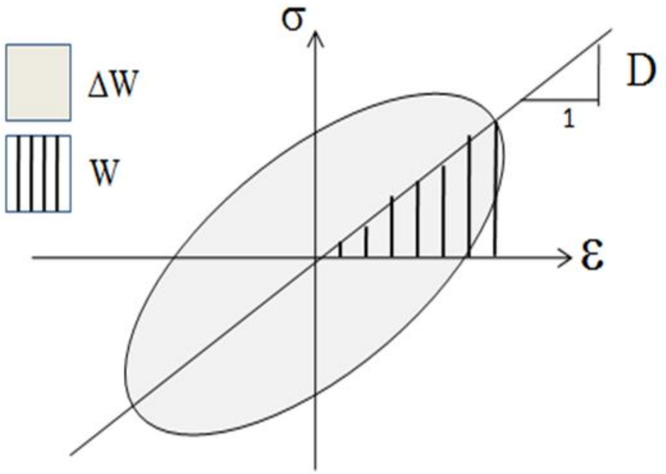
Perfect matching between ACS SASSI and ANSYS results with condensation

2. Option NON:

Nonlinear SSI Analysis for Nonlinear RC Shearwall Structures

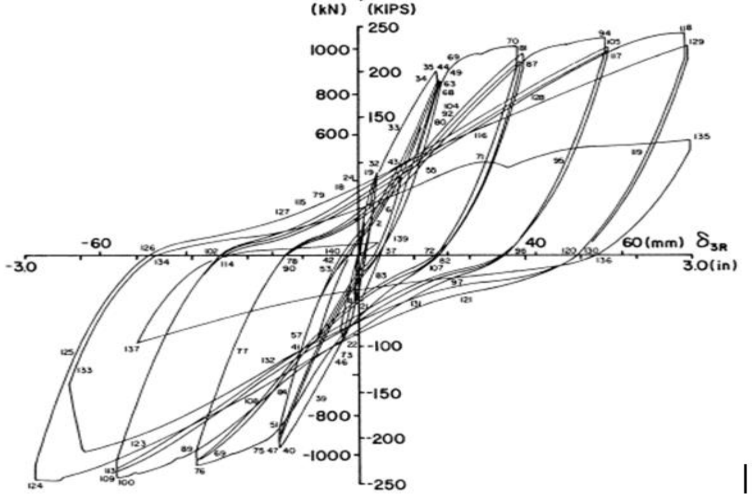
Option NON Modeling of Concrete Hysteretic Behavior

Linearized Hysteretic Model



Frequency Domain
Linearized Hysteretic Model

Experimental Hysteretic Model



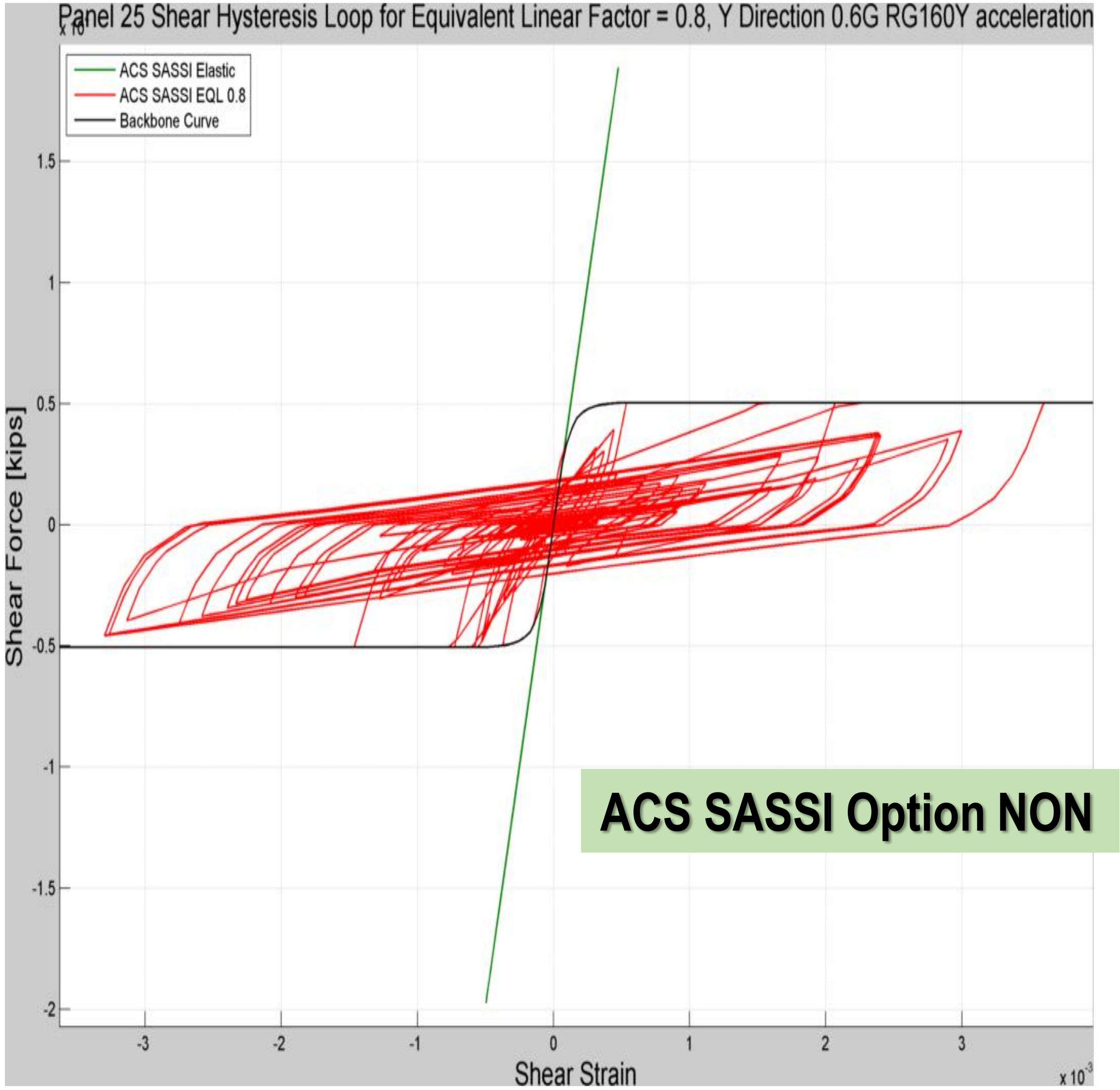
Time Domain
Hysteretic Model

Comparative nonlinear SSI analysis results of the hybrid approach against the "*true*" nonlinear time-integration approach show a *good accuracy.*

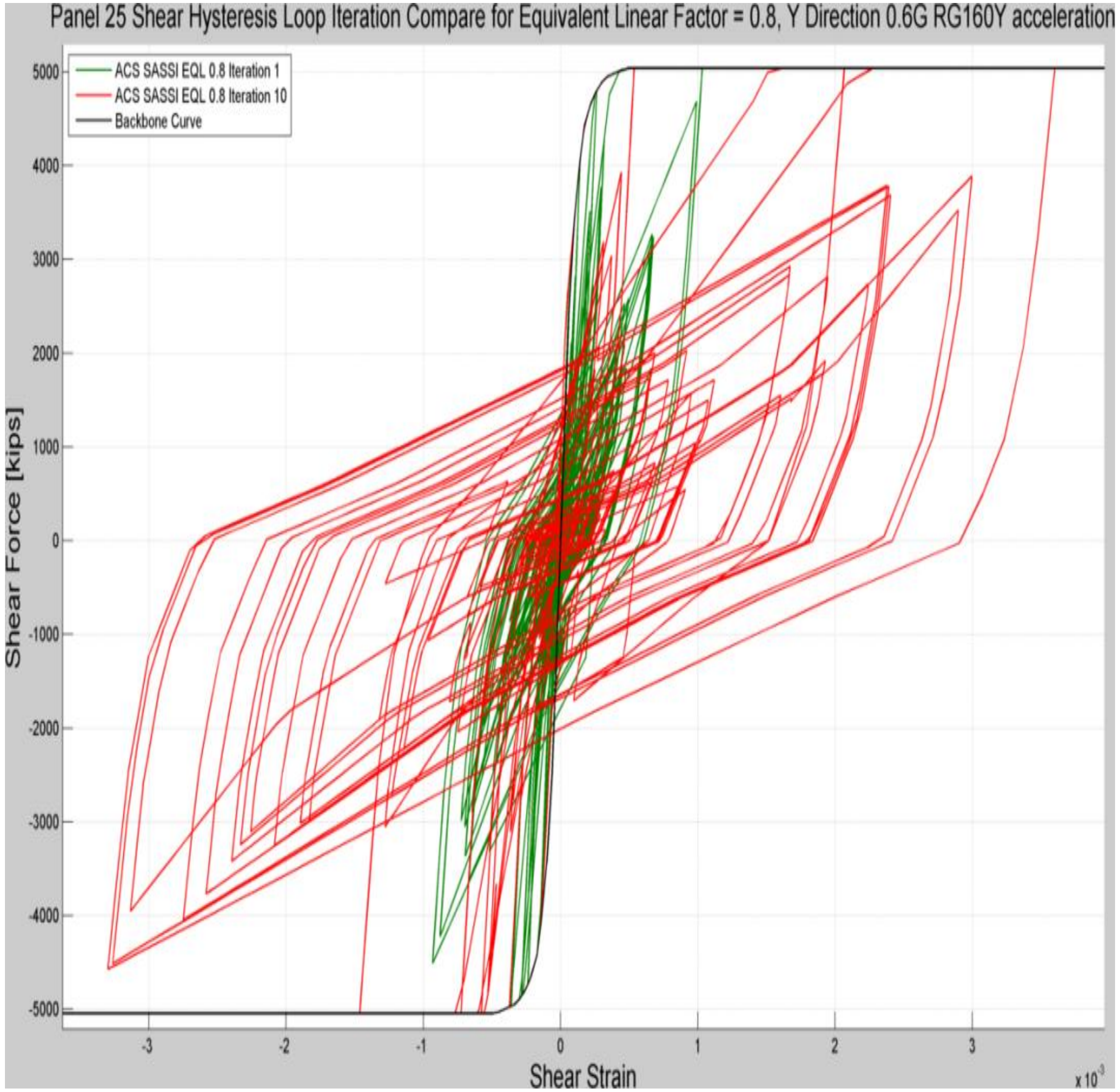
Fast and accurate nonlinear SSI analyses at a small fraction of the runtime of a time domain nonlinear analysis. *Much more robust than nonlinear time integration approaches (see also, Kausel and Assimaki, 2002)*

Reinforced Concrete Shearwall Structure Nonlinear SSI Analysis

Elastic vs. Nonlinear



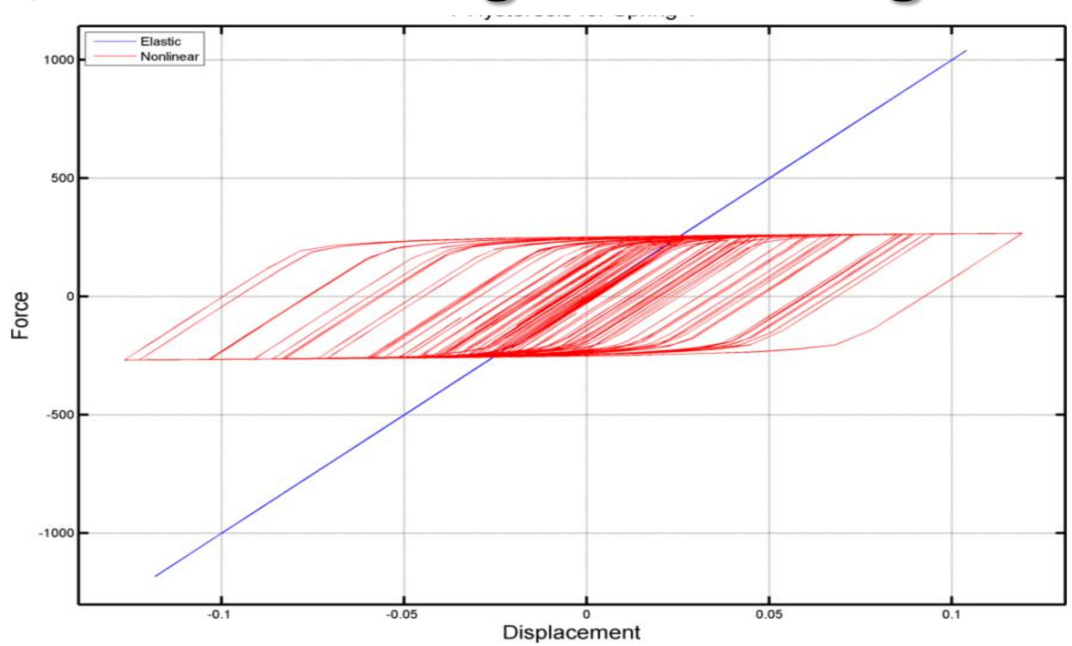
1st Iteration vs. Last Iteration

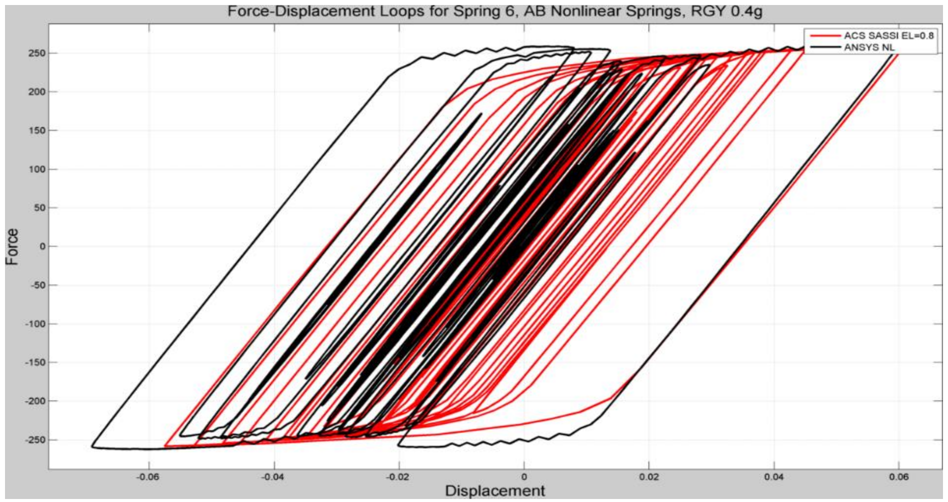


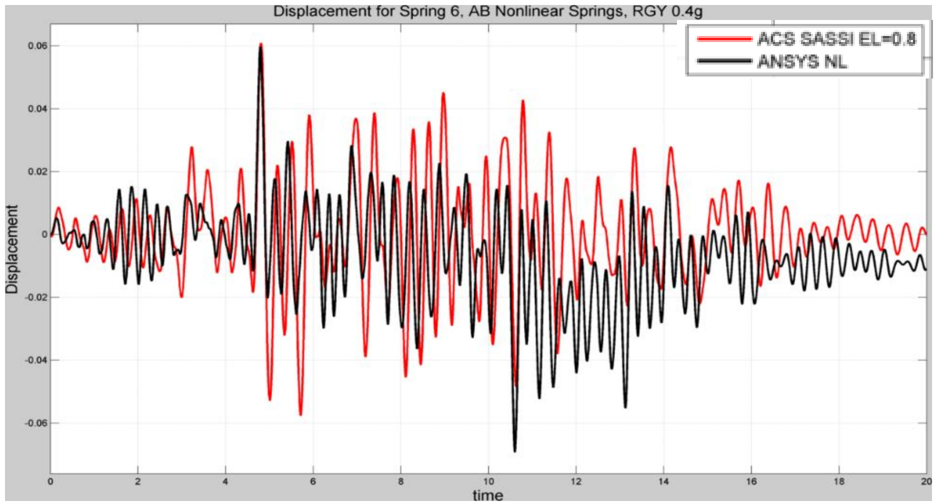
Nonlinear Springs Used for Modeling Base-Isolators, Wall-Soil Slippage, or Checking if Building Slides

Base-Isolators

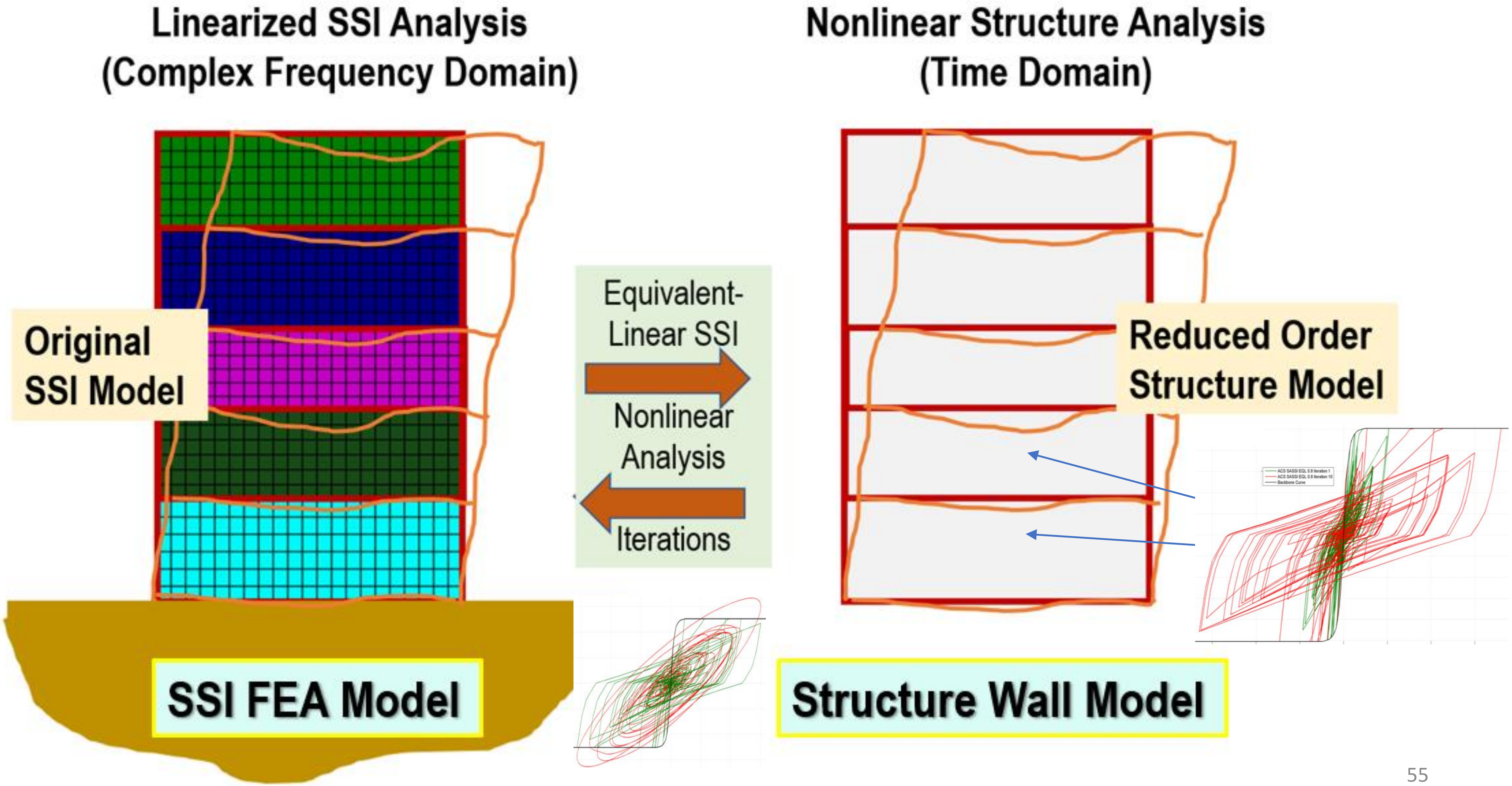
Small-Moderate Sliding







Nonlinear Structure SSI Analysis Using A Hybrid Frequency-Time Domain Approach (Iterative Coupled Global-Local Iterations)



Nonlinear SSI Analysis Using A Hybrid Frequency-Time Approach based on A Fast Iterative Procedure

The implemented SSI hybrid approach uses an iterative equivalent-linearization (EQL) based on a *global* linearized SSI solution in the complex frequency combined a *local,* "true" nonlinear wall panel behavior in time domain based on the displacement BCs computed from each SSI restart iteration (about twice faster than initial run).

The runtime of a nonlinear SSI analysis based on iterative SSI analyses is only about 2-3 times (4-6 iterations) the runtime of a linear SSI analysis.

Description of Nonlinear SSI Methodology Coupled Iterative Steps

The nonlinear SSI analysis is based on an iterative scheme that includes two separate computational steps at each iteration, as follows:

- Step 1: Perform an *equivalent-linear SSI analysis* in complex frequency via SASSI approach to compute the structural displacements for each nonlinear RC wall, and then,
- Step 2: Perform a *nonlinear time-integration analysis* for each RC wall submodel loaded with the SSI displacements from Step 1, to compute the in-plane shear and bending nonlinear wall responses using *standard-based BBCs and selected hysteretic models*. Then, determine *the equivalent-linear stiffness and damping for each wall* using DRF to be used for next SSI iteration, until converged.

REMARKS:

1) Step 1 uses the *original, refined FE SSI model*, while Step 2 uses a *reduced-order structural model* composed by nonlinear RC walls. Therefore, the nonlinear time-domain Step 2 analysis is extremely fast. For DES, *condensed soil impedance matrix* should be used for SSI iterations (ANALYS option).
2) The nonlinear SSI methodology was validated for several shearwall building models against CSI PERFORM3D code, OpenSees 3D FIBER model and 2D MVLEM software, XTRACT, LS-DYNA.

Option NON Assumptions for Nonlinear Wall Deformation

Option NON is applicable to the reinforced concrete structures for simulating the concrete cracking and post-cracking behavior in the shearwalls for the design-level and/or beyond-the-design-level seismic inputs.

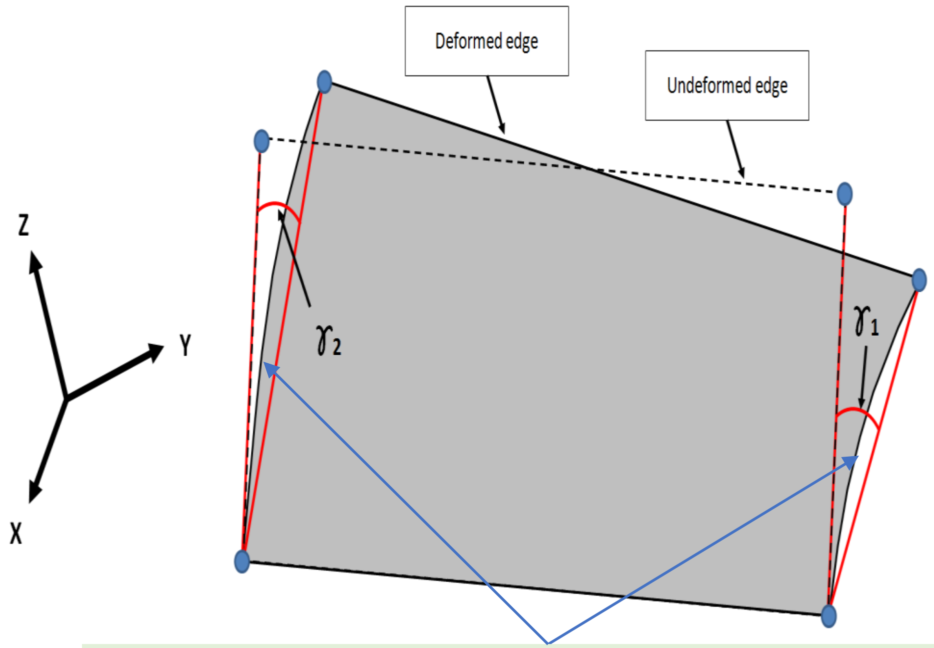
The Option NON was validated for the low-rise reinforced concrete shearwall buildings that fail primarily due to the *in-plane shear deformation*. Based on the time-domain hysteretic behavior, the elastic modulus and damping in each concrete wall are modified iteratively based on the local stress and deformation levels. No out-plane nonlinear concrete behavior is considered.

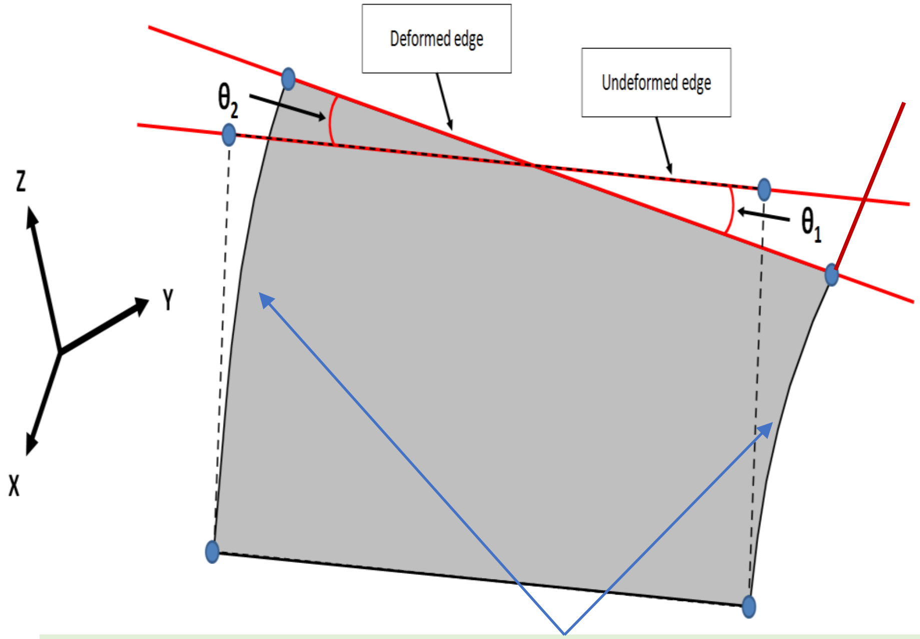
Option NON can also consider the nonlinear concrete behavior due to *the in-plane bending deformation* effects.

In the same nonlinear structure FE model, the analyst can include wall panels that fail due to either the shear deformation or the bending deformation, respectively.

Shear deformation only: *Option NON Simple*
Shear & Bending deformation: *Option NON Advanced*

Wall Shear and Bending Deformation Are Computed for Each Wall, at Each Floor Level at Each SSI Iteration





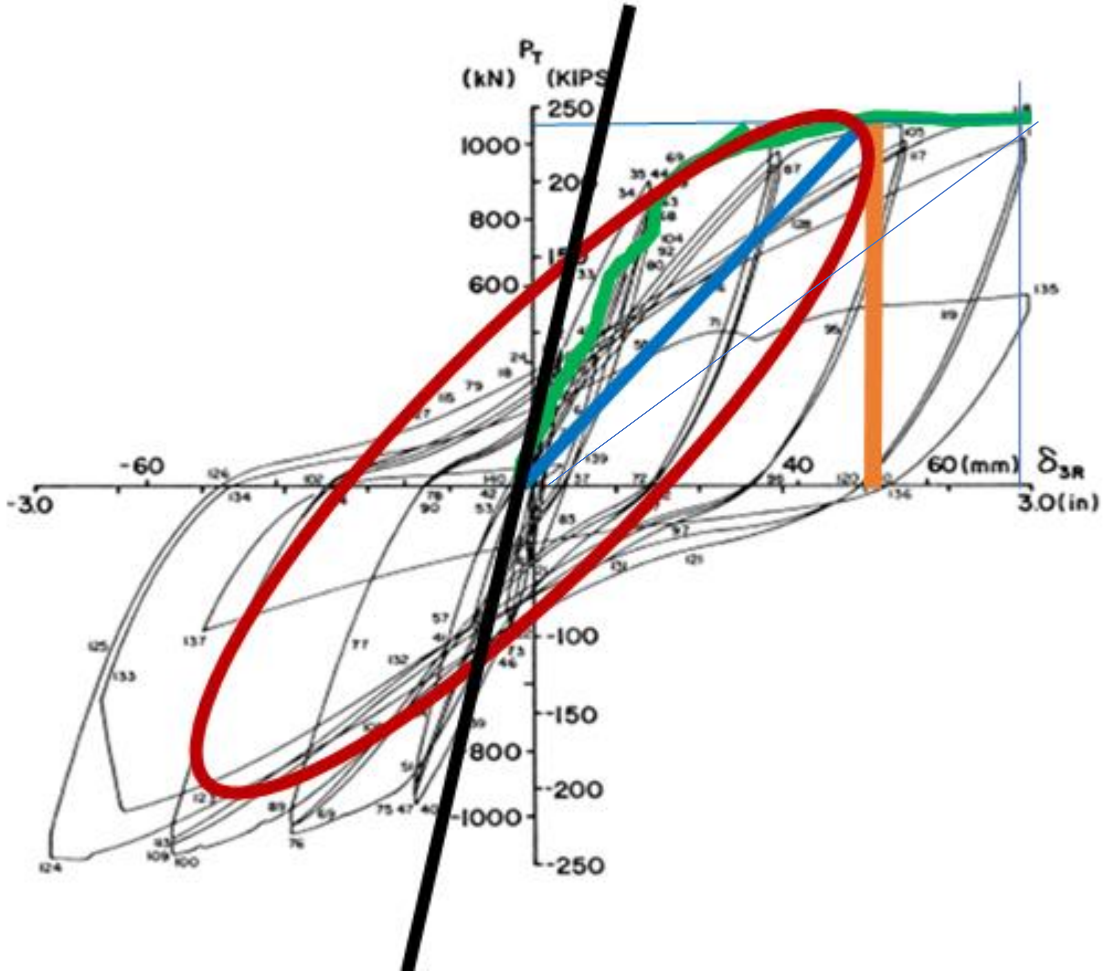
Shear Deformation (vertical edges)
(shear strain assumed constant per height, wall vertical edge displacements are linear)

Bending Deformation (horizontal edges)
(curvature assumed constant per height, wall vertical edge displacements are parabolic)

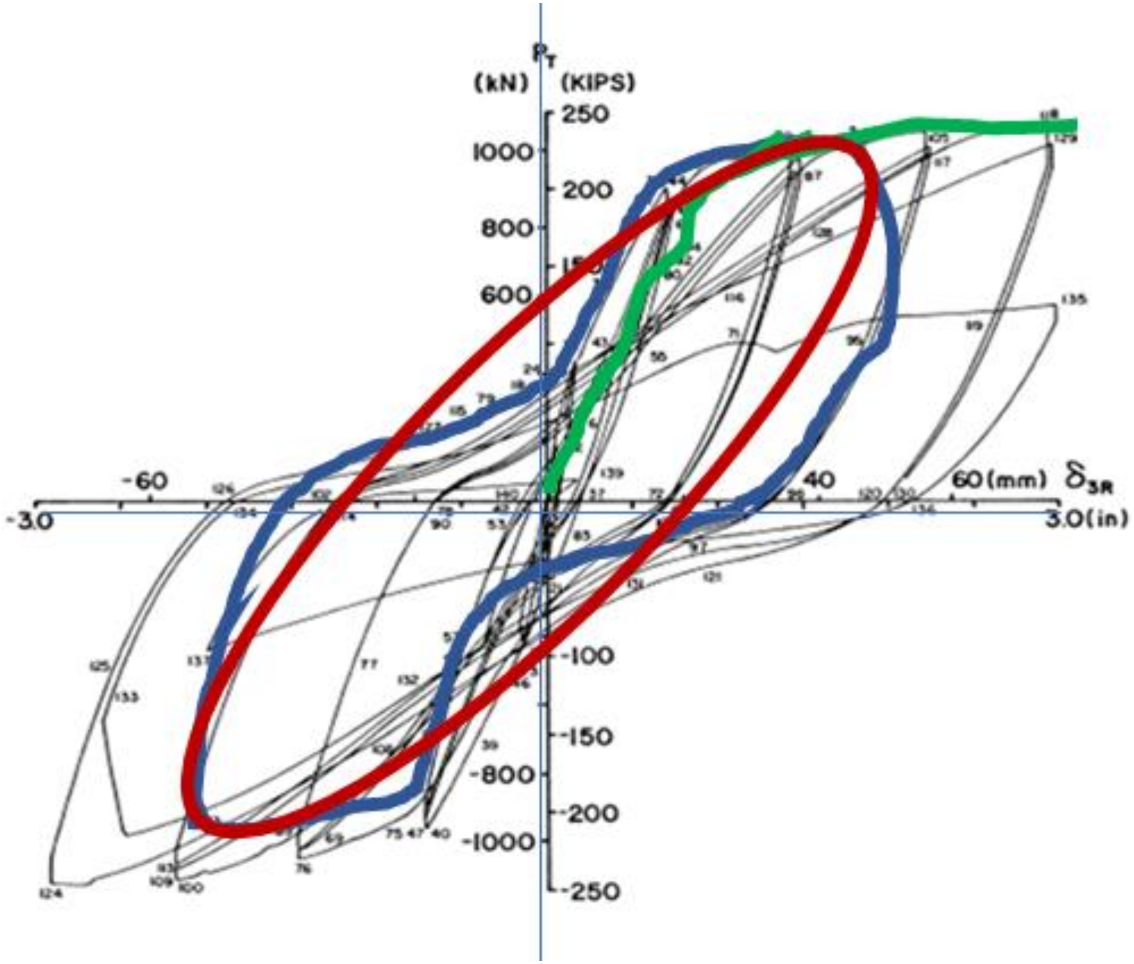
Remark: Rigid body motion is removed. Very important.

Computing Wall Equivalent Dynamic Stiffness and Damping

Computing Equivalent Stiffness (Eech)

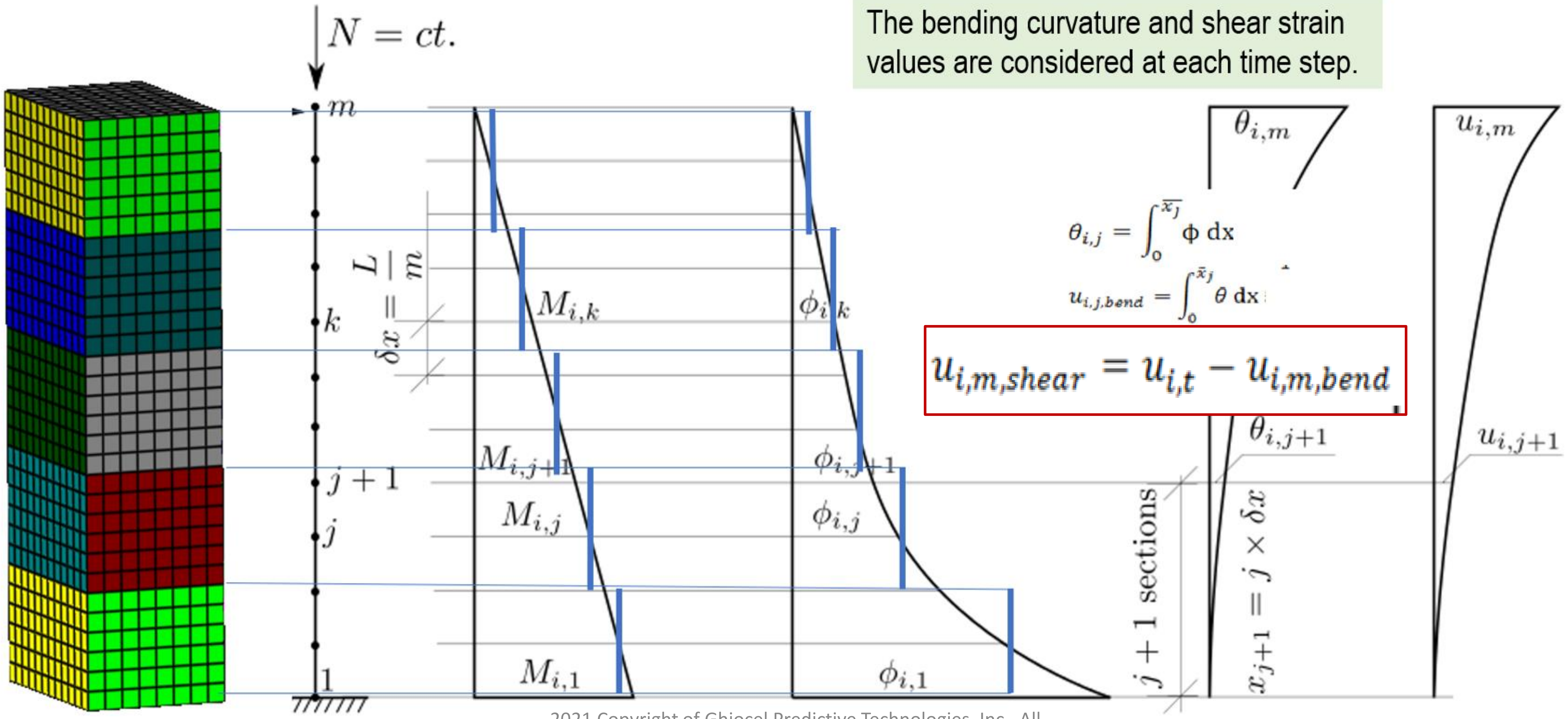


Computing Equivalent Damping (Dech)



Nonlinear Shear and Bending Responses Are Computed Based on Each Floor Structural Displacements at Each SSI Iteration

The bending curvature and shear strain values are considered at each time step.



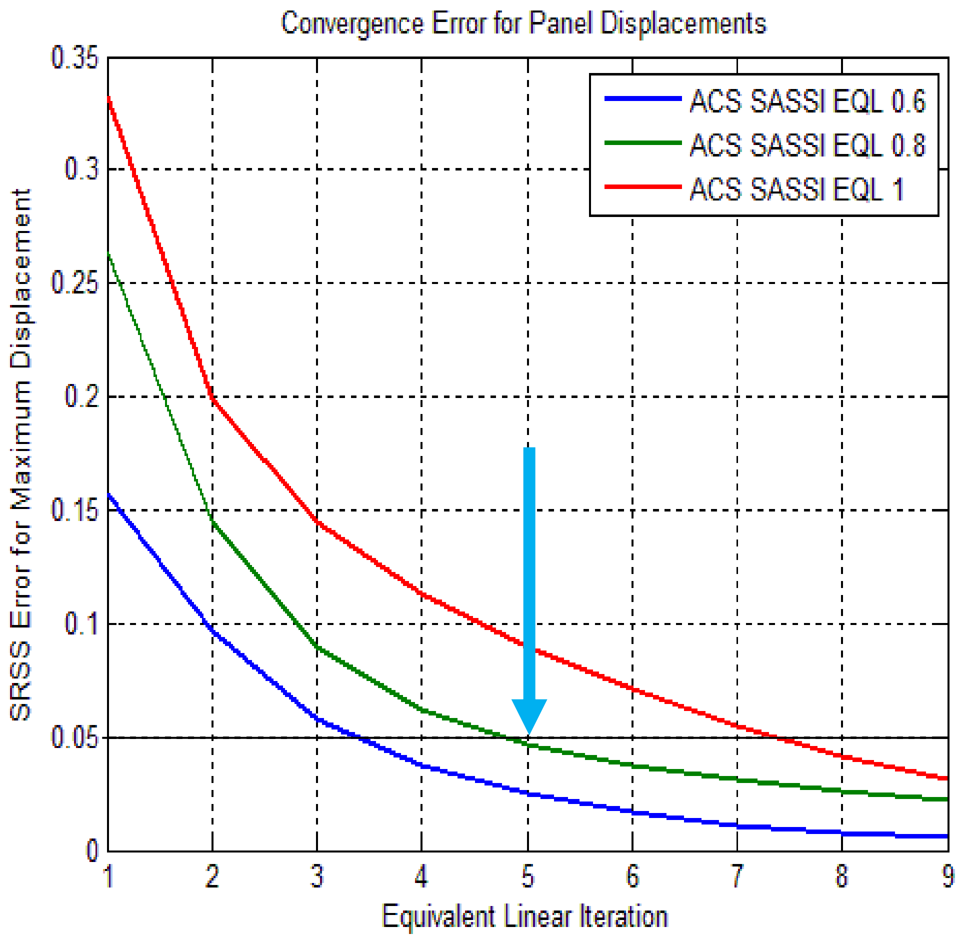
$$\theta_{i,j} = \int_0^{\bar{x}_j} \phi \, dx$$

$$u_{i,j,bend} = \int_0^{\bar{x}_j} \theta \, dx$$

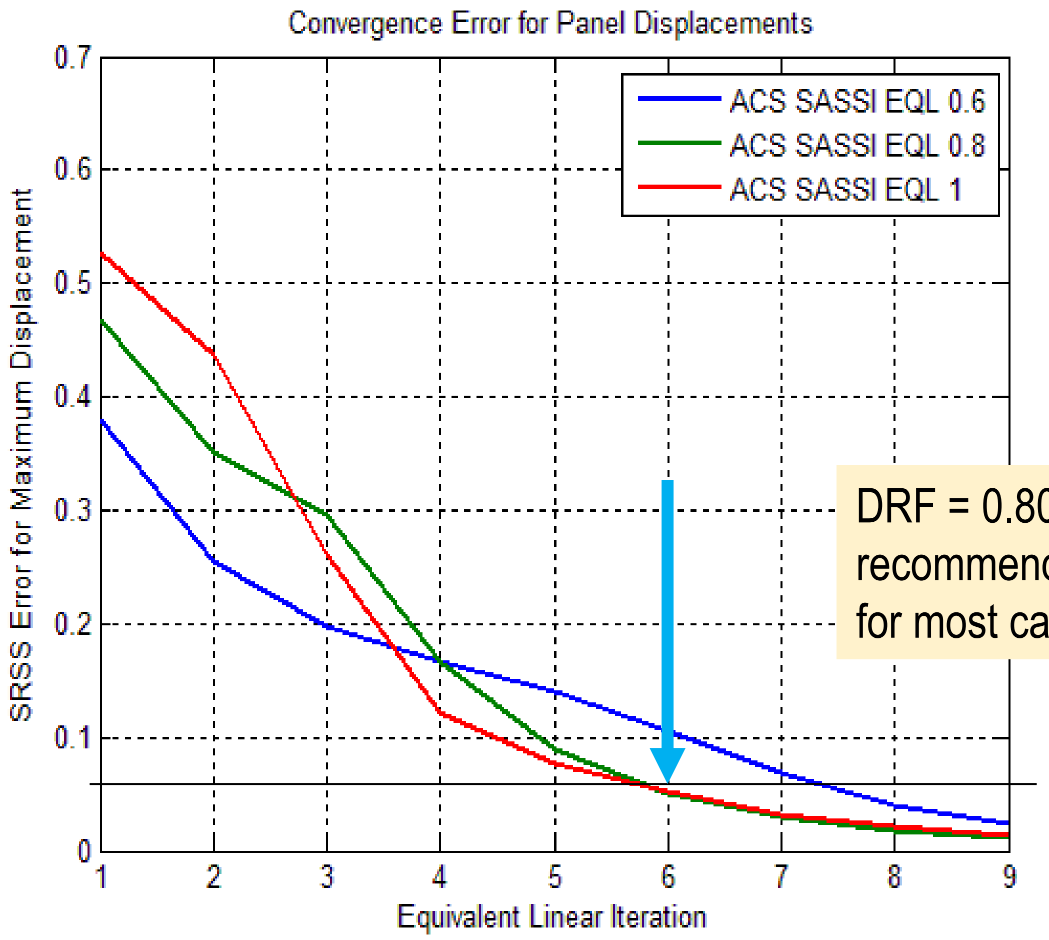
$$\boxed{u_{i,m,shear} = u_{i,t} - u_{i,m,bend}}$$

Typical Nonlinear Structure SSI Solution Convergence Using Constant DRF for Each Iteration for 5% Accuracy Tolerance

0.30g Input



0.60g Input

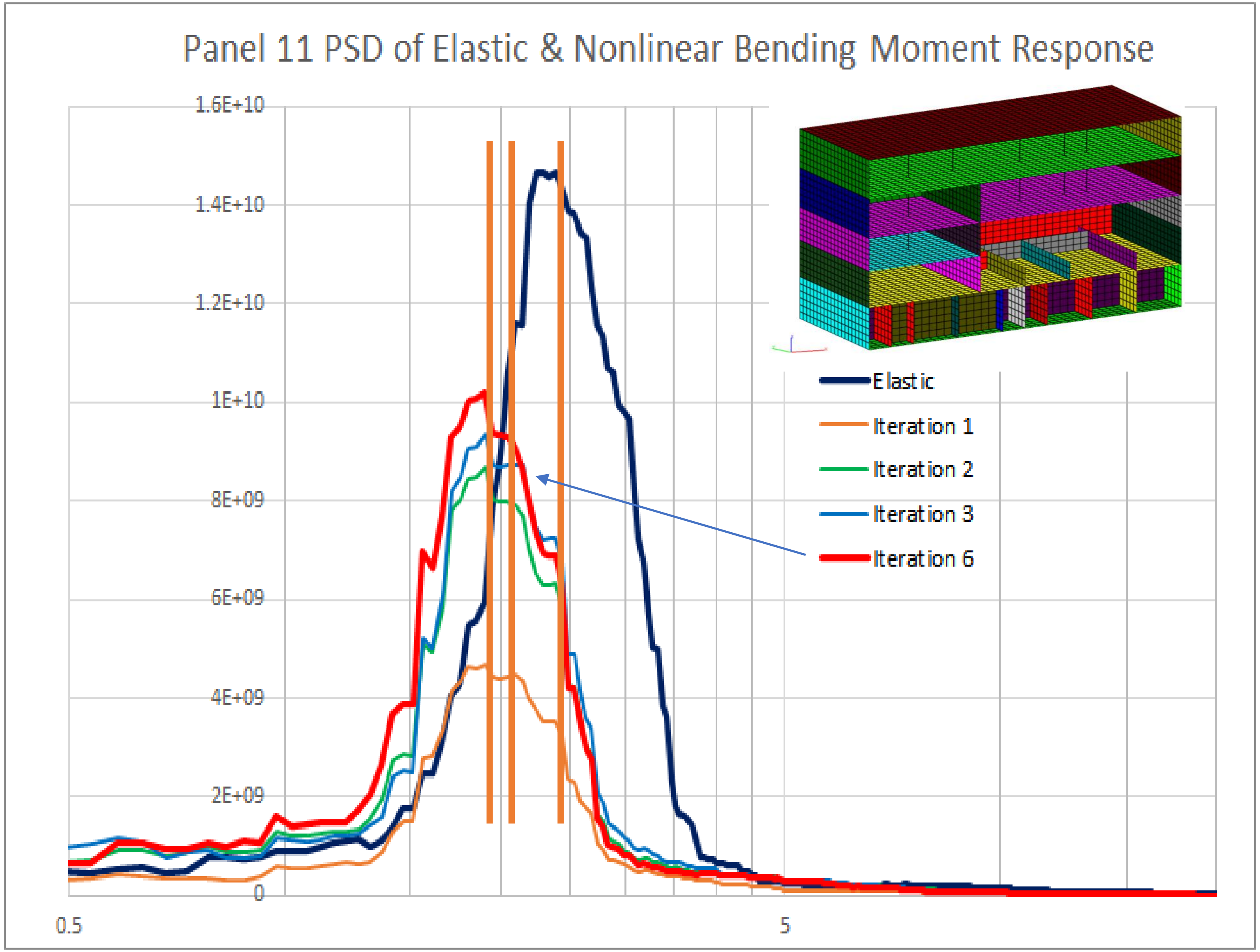


DRF = 0.80 is recommended for most cases

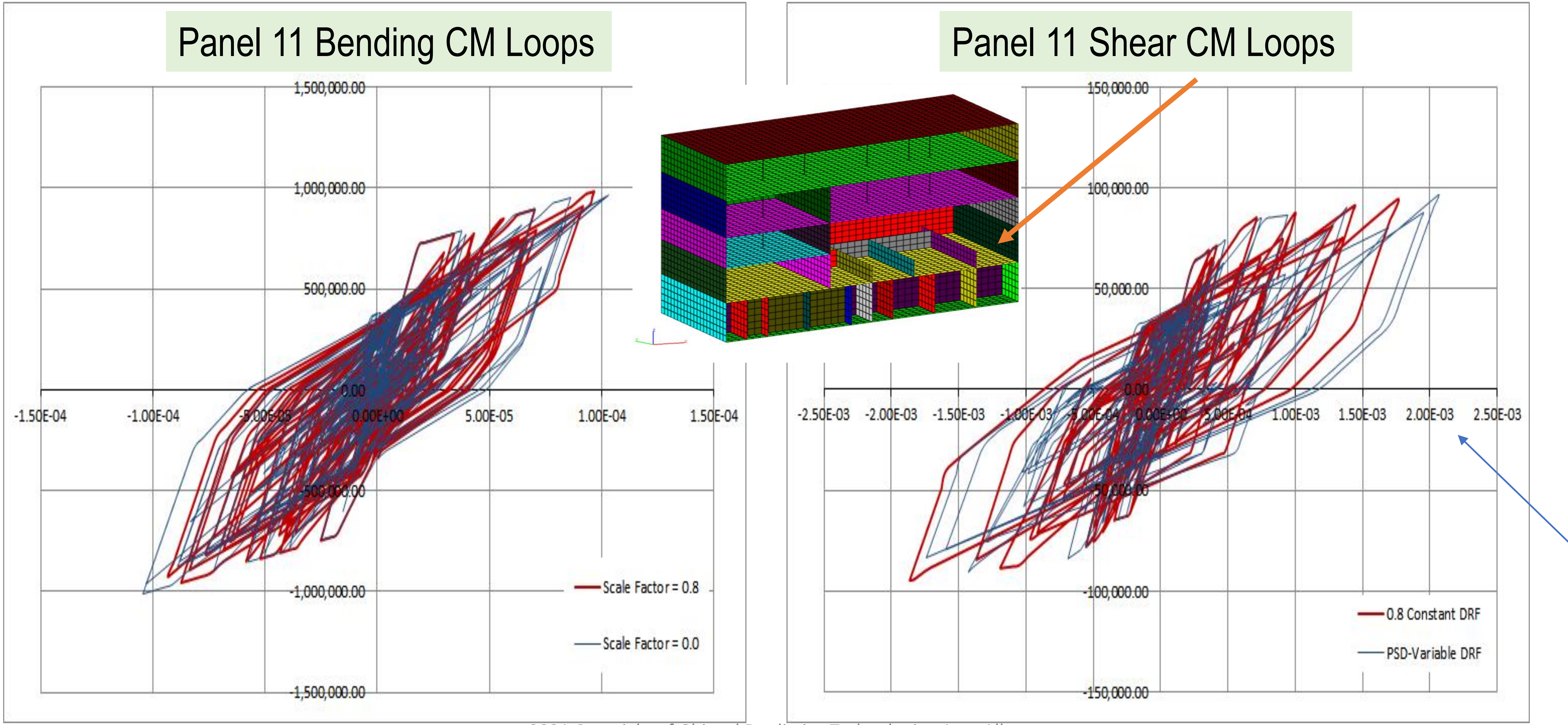
Iterative Equivalent Linearization Using Variable or Constant DRF

The *PSD-based DRF* is computed based on the frequency content of the PSD frequency computed for the nonlinear shear force or bending moment for each wall at each floor level and each iteration.

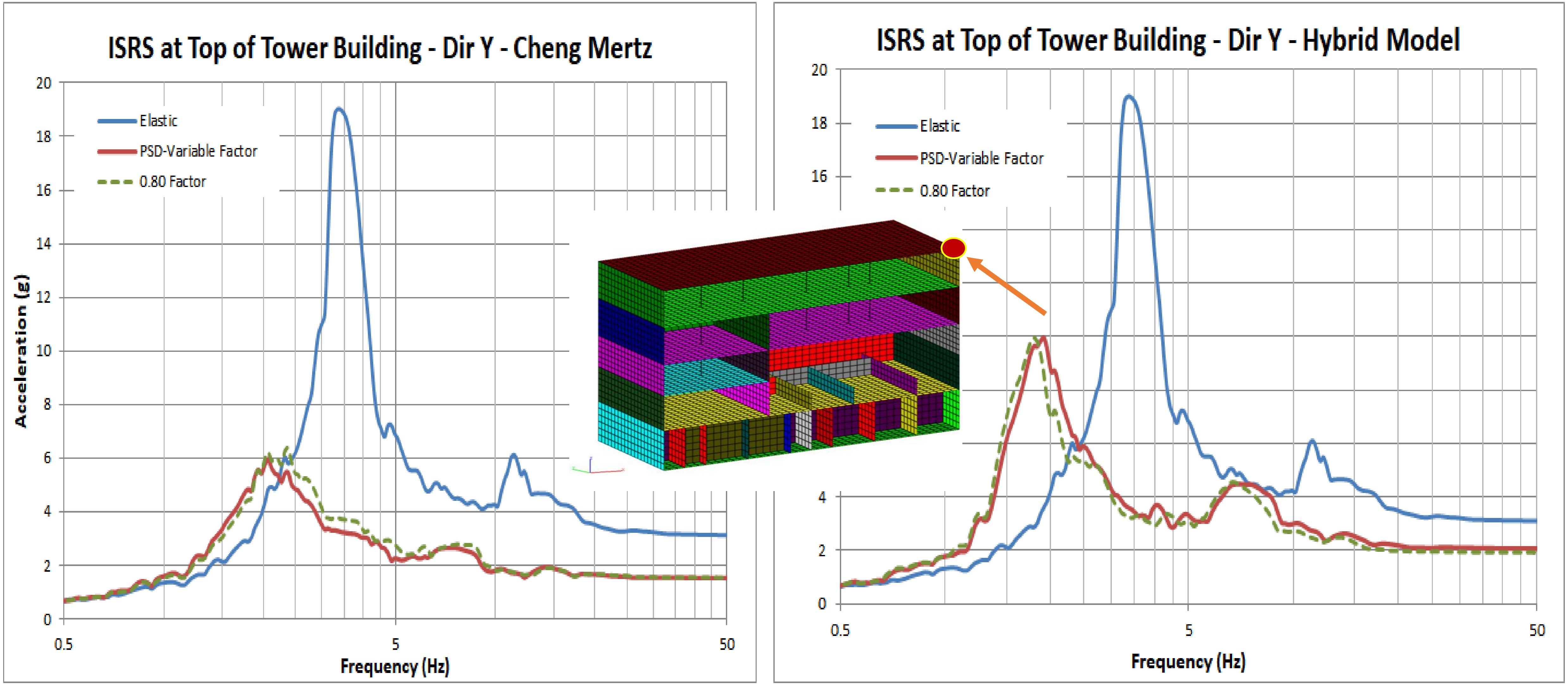
The DRF is computed based the PSD dominant frequency shifts at each iteration, as shown in the right-side figure.



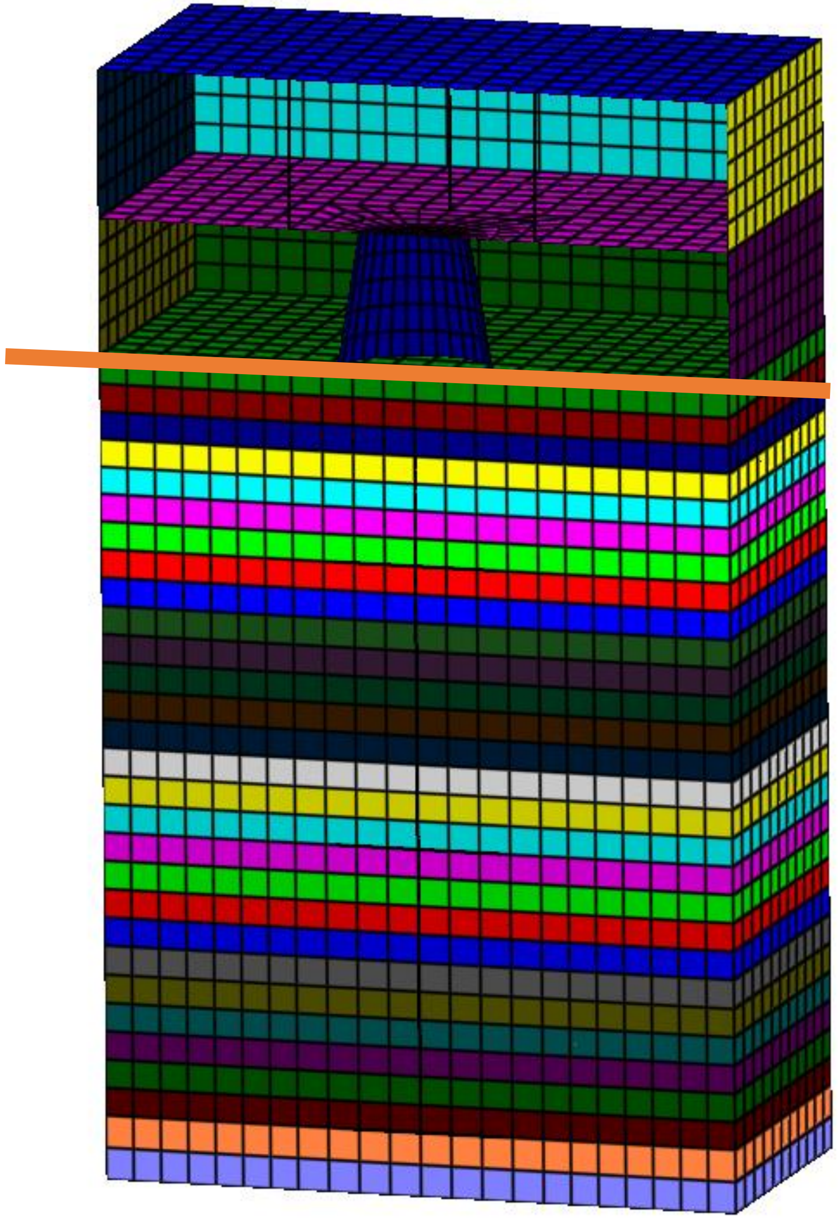
Hysteretic Responses for PSD-based Variable DRF vs. 0.80 DRF

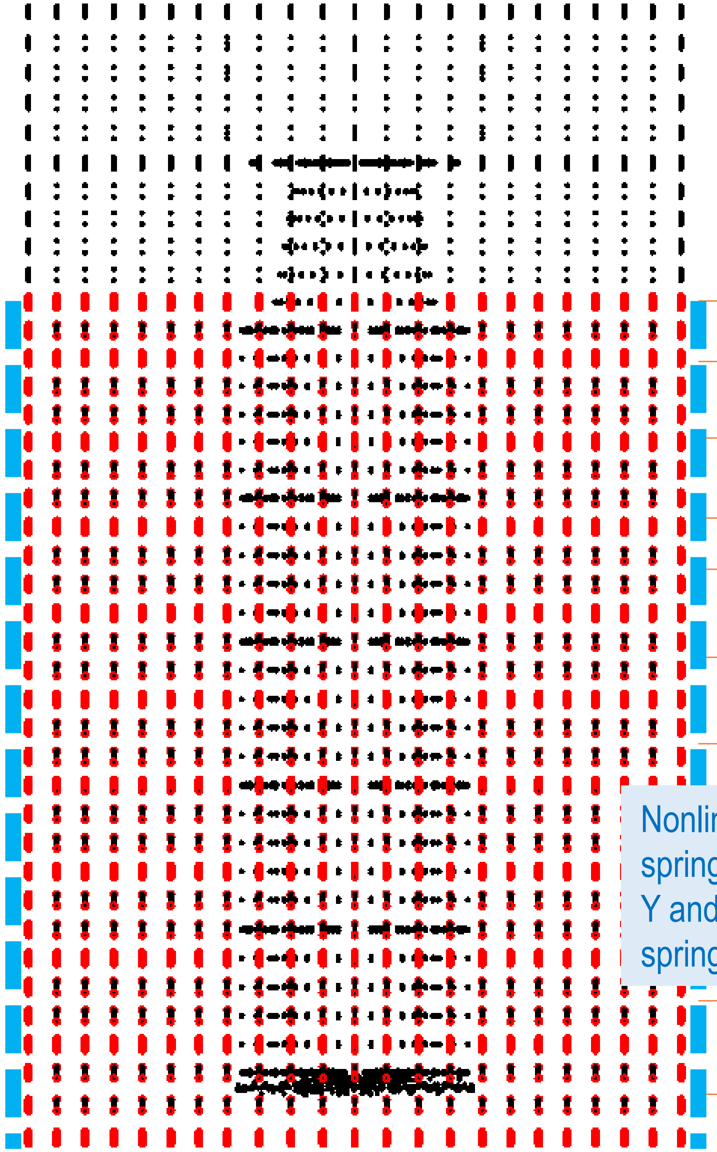


Nonlinear ISRS Computed Using PSD-based DRF vs. 0.80 DRF



SMR with Nonlinear RC Walls and Nonlinear Wall-Soil Interface

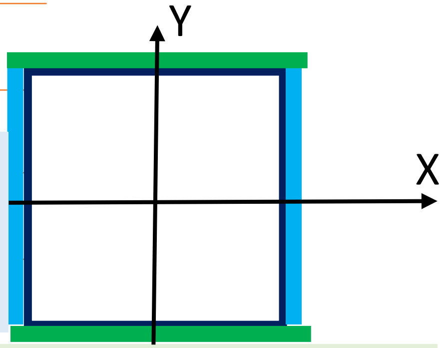




Side-soil interface modeled using nonlinear springs:

- **Nonlinear interface**

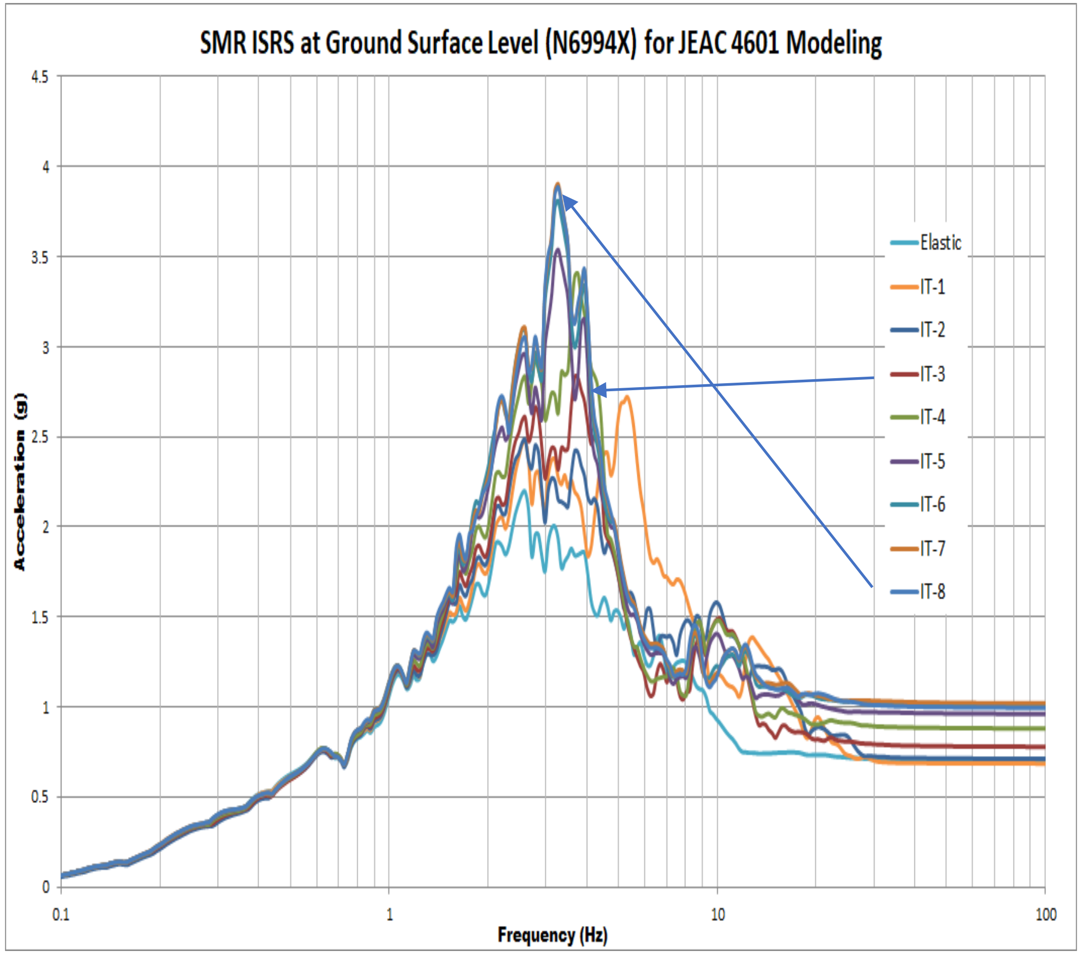
Rigid springs in normal directions X, Y and Z, and nonlinear springs in tangential directions X, Y and Z. Friction forces vary with depth.



Nonlinear side-soil springs in tangential Y and Z directions springs

Nonlinear side-soil springs in tangential X and Z directions springs

Embedded SMR SSI Response Convergence for Nonlinear RC Walls (Panels) and Nonlinear Wall-Soil Interface (Springs) for 0.60g



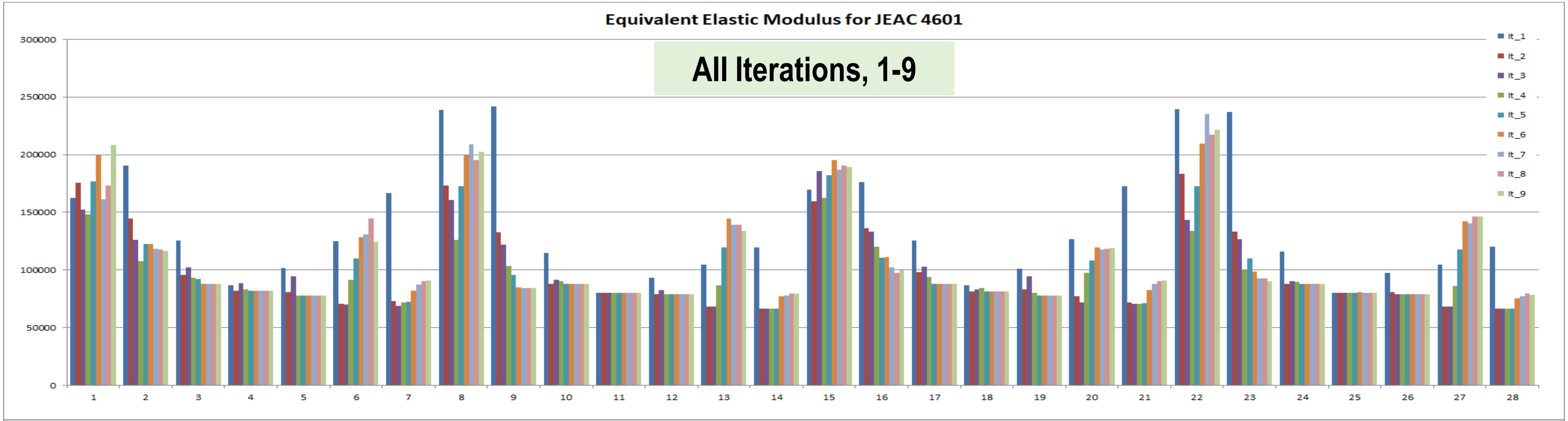
*Nonlinear-Response-Convergence-Checking - Notepad

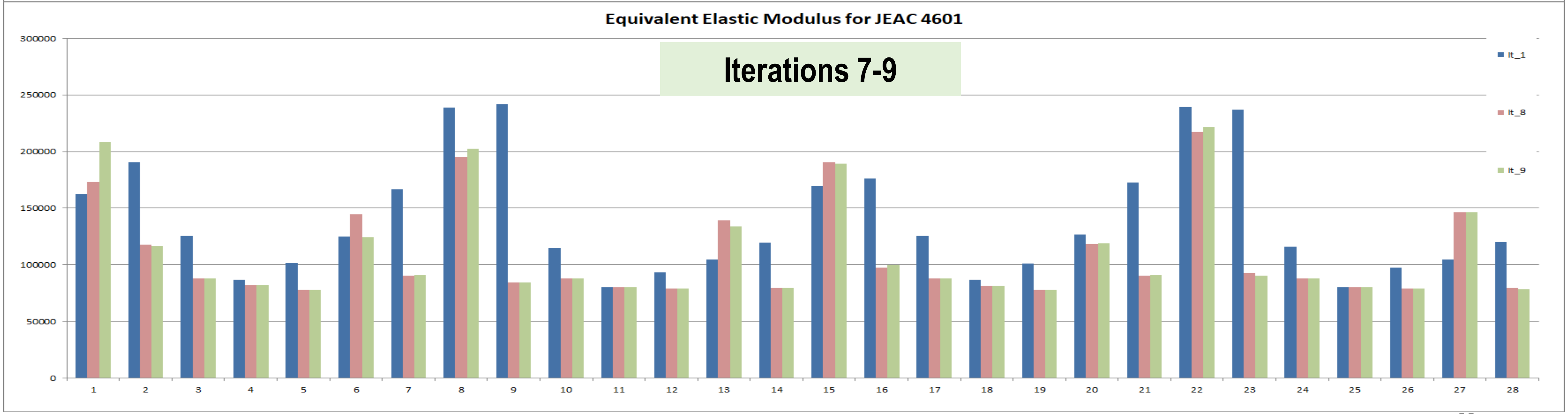
File Edit Format View Help

Convergence File

```
Number of Iteration =    1
Elastic E modulus relative difference between current and previous iteration =    70.738%
Damping ratio relative difference between current and previous iteration =    2.883%

Number of Iteration =    2
Elastic E modulus relative difference between current and previous iteration =    42.549%
Damping ratio relative difference between current and previous iteration =   17.882%

Number of Iteration =    3
Elastic E modulus relative difference between current and previous iteration =    54.342%
Damping ratio relative difference between current and previous iteration =   16.987%

Number of Iteration =    4
Elastic E modulus relative difference between current and previous iteration =    27.948%
Damping ratio relative difference between current and previous iteration =   11.733%

Number of Iteration =    5
Elastic E modulus relative difference between current and previous iteration =    15.903%
Damping ratio relative difference between current and previous iteration =    9.445%

Number of Iteration =    6
Elastic E modulus relative difference between current and previous iteration =    13.246%
Damping ratio relative difference between current and previous iteration =    6.800%

Number of Iteration =    7
Elastic E modulus relative difference between current and previous iteration =     7.132%
Damping ratio relative difference between current and previous iteration =    5.197%

Number of Iteration =    8
Elastic E modulus relative difference between current and previous iteration =     4.837%
Damping ratio relative difference between current and previous iteration =    3.408%

Number of Iteration =    9
Elastic E modulus relative difference between current and previous iteration =     4.251%
Damping ratio relative difference between current and previous iteration =    0.838%
```

Iterative Equivalent E Modulus Due to Shear Effects in SMR Walls





Hysteretic Models Library Available for Nonlinear RC Walls

The hysteretic model library includes 8 types of models applicable to the structure RC walls:

1-Cheng-Mertz Shear (CMS)

2-Cheng-Mertz Bending (CMB)

3-Takeda (TAK)

4-General Massing Rule (GMR)

5-Maximum Point-Oriented (PO) for Shear - per JEAC 4601 App. 3.6

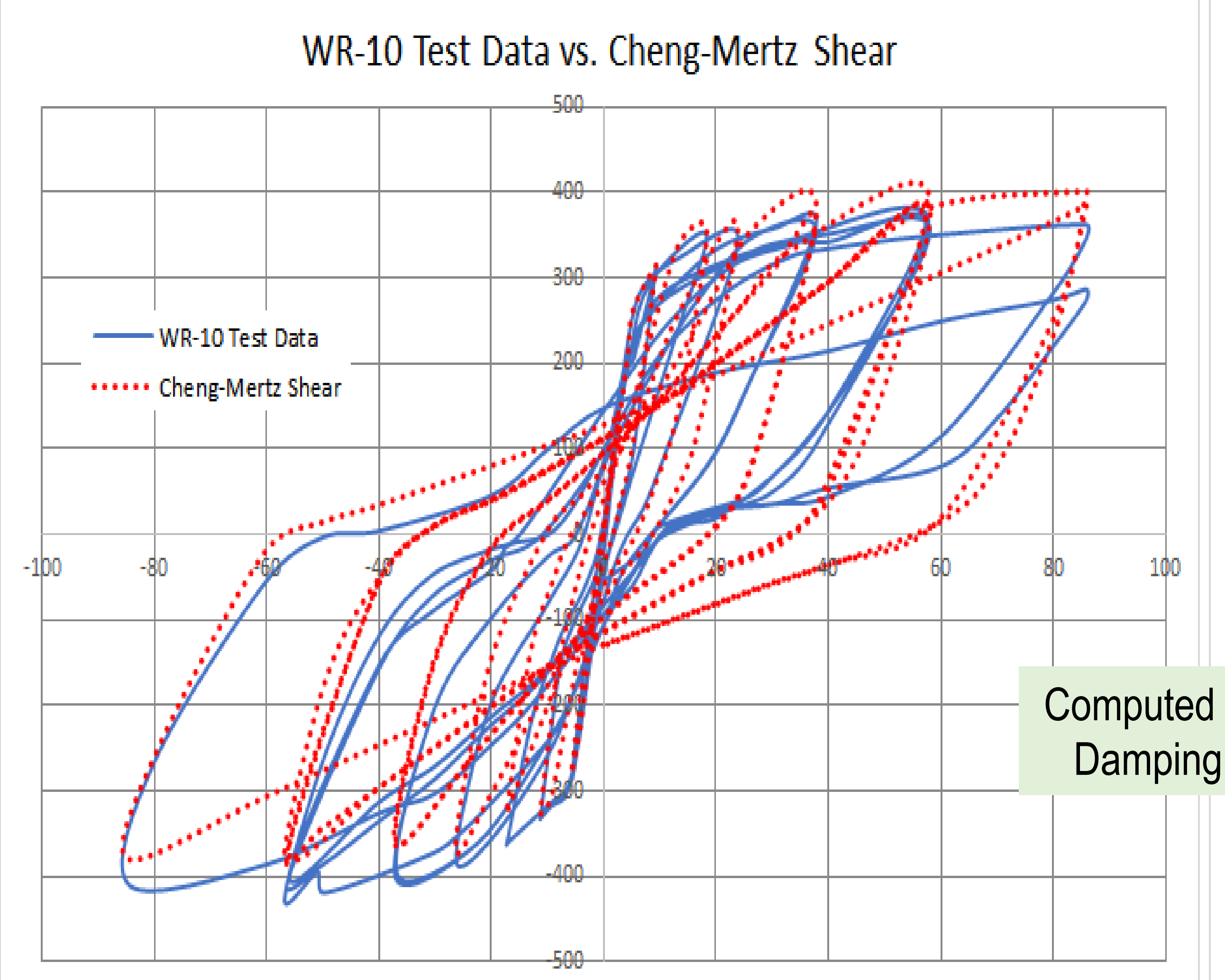
6-Maximum Point-Oriented Degrading Trilinear (PODT) for Bending - per JEAC 4601 App. 3.6

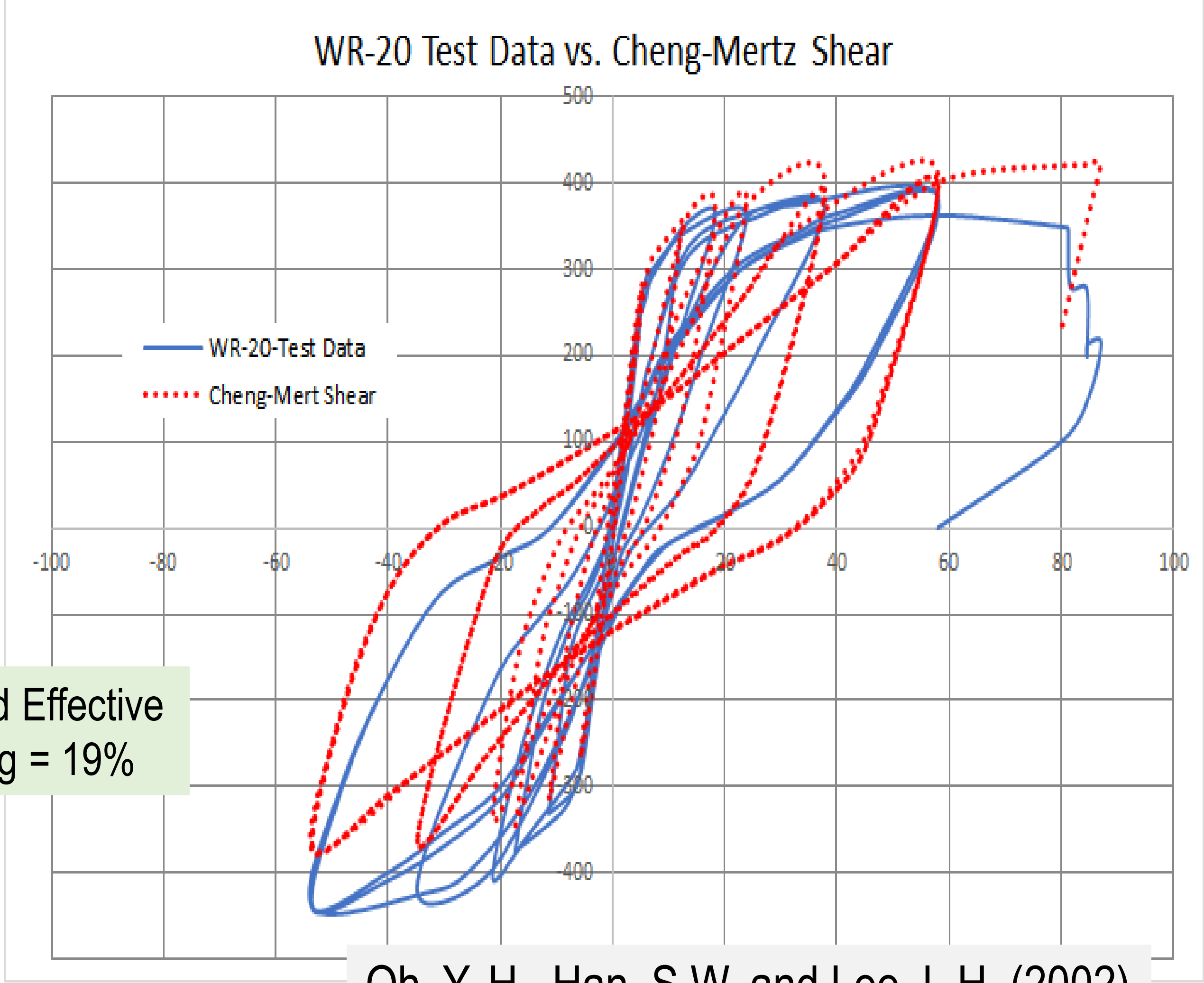
7-Hybrid Shear (HYS) – obtained by combining PO Shear and CMS models

8-Hybrid Bending (HYB) - obtained by combining PODT Bending and CMB models

Cheng-Mertz Shear Hysteretic Model Against HU Wall Test Data

Cheng-Mertz Shear Model (Model 1)



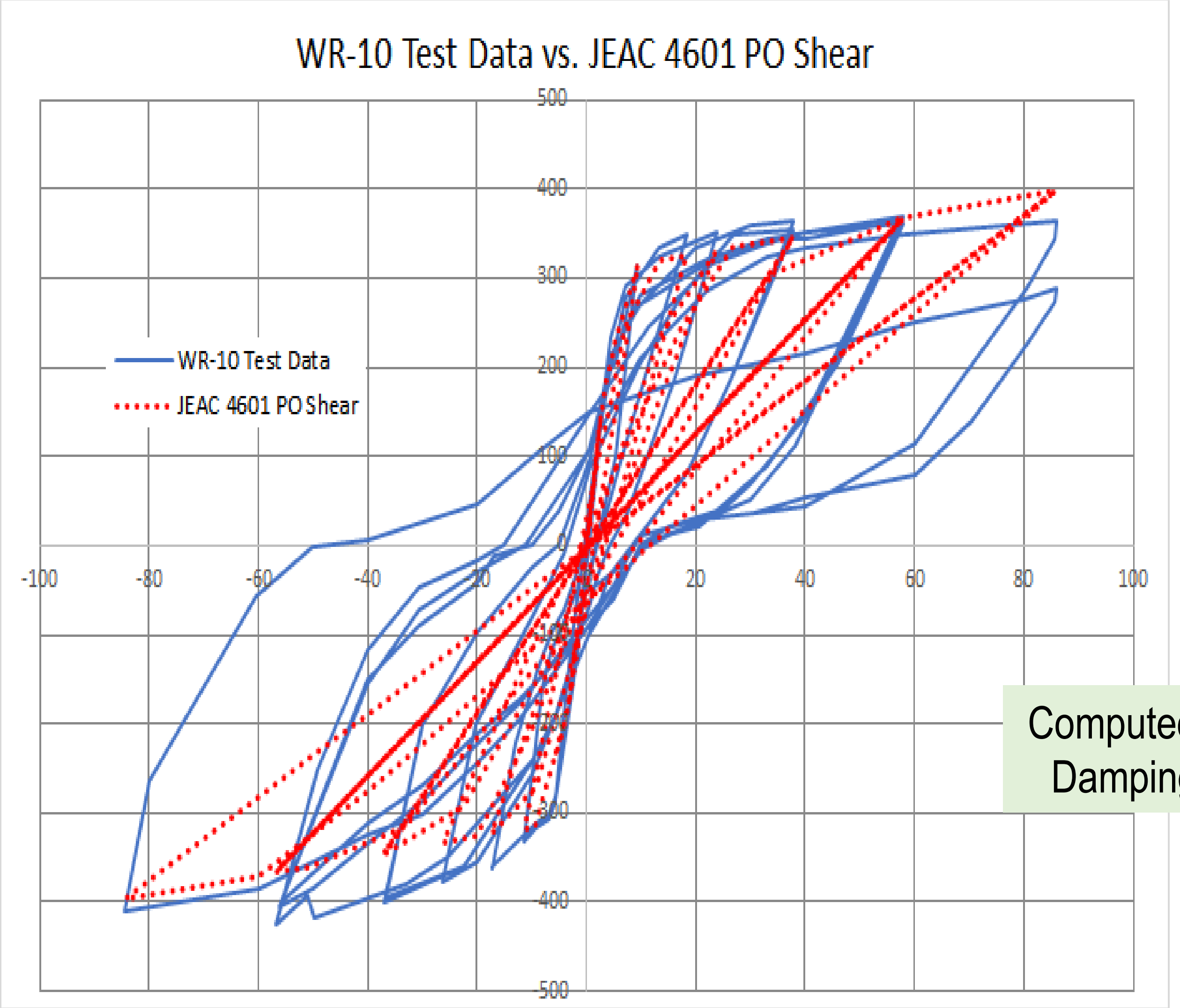


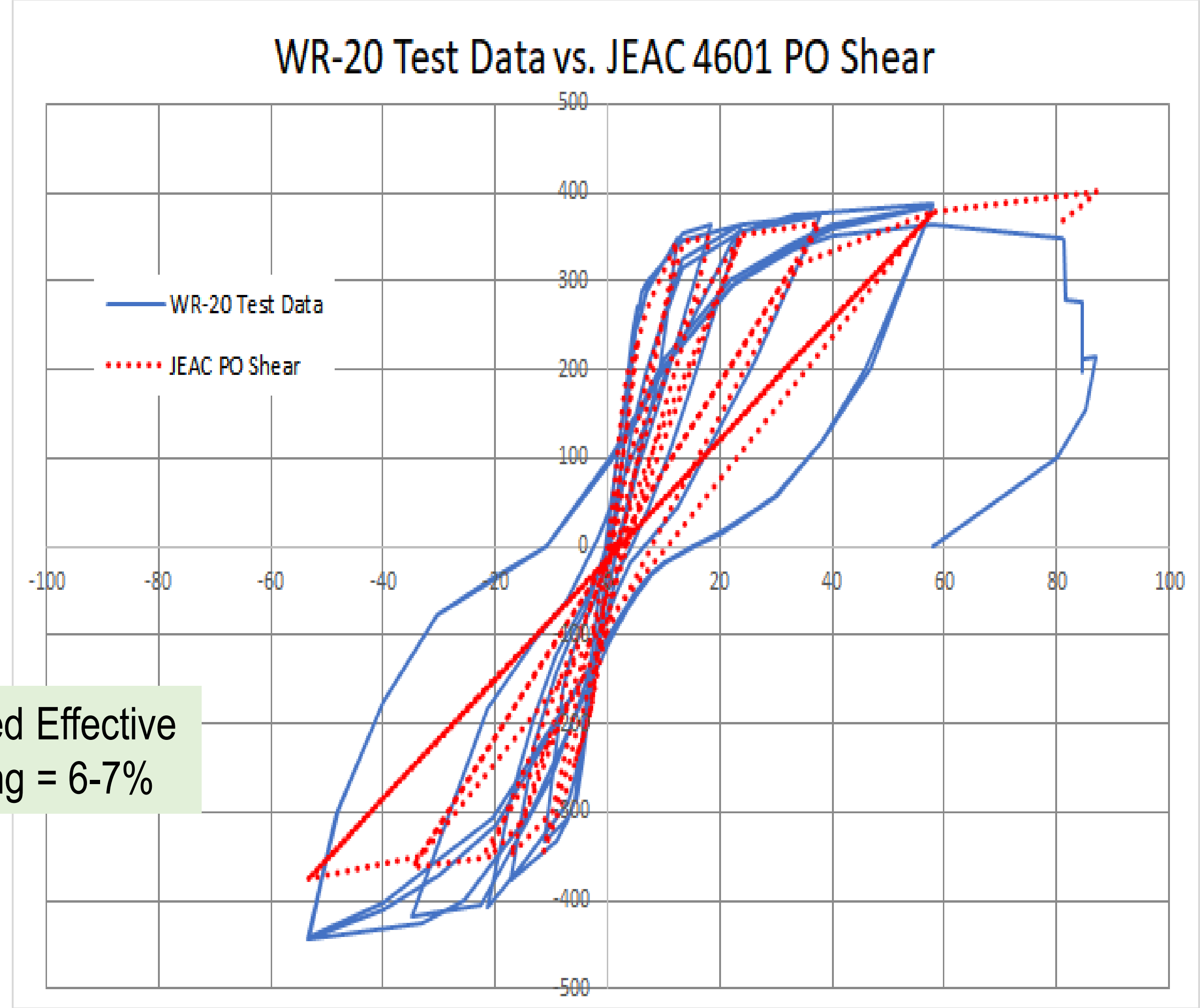
Computed Effective Damping = 19%

Oh, Y. H., Han, S.W. and Lee, L.H. (2002)

JEAC PO Shear Hysteretic Model Against HU Wall Test Data

JEAC 4601 Point-Oriented (PO) Shear Model (Model 5)

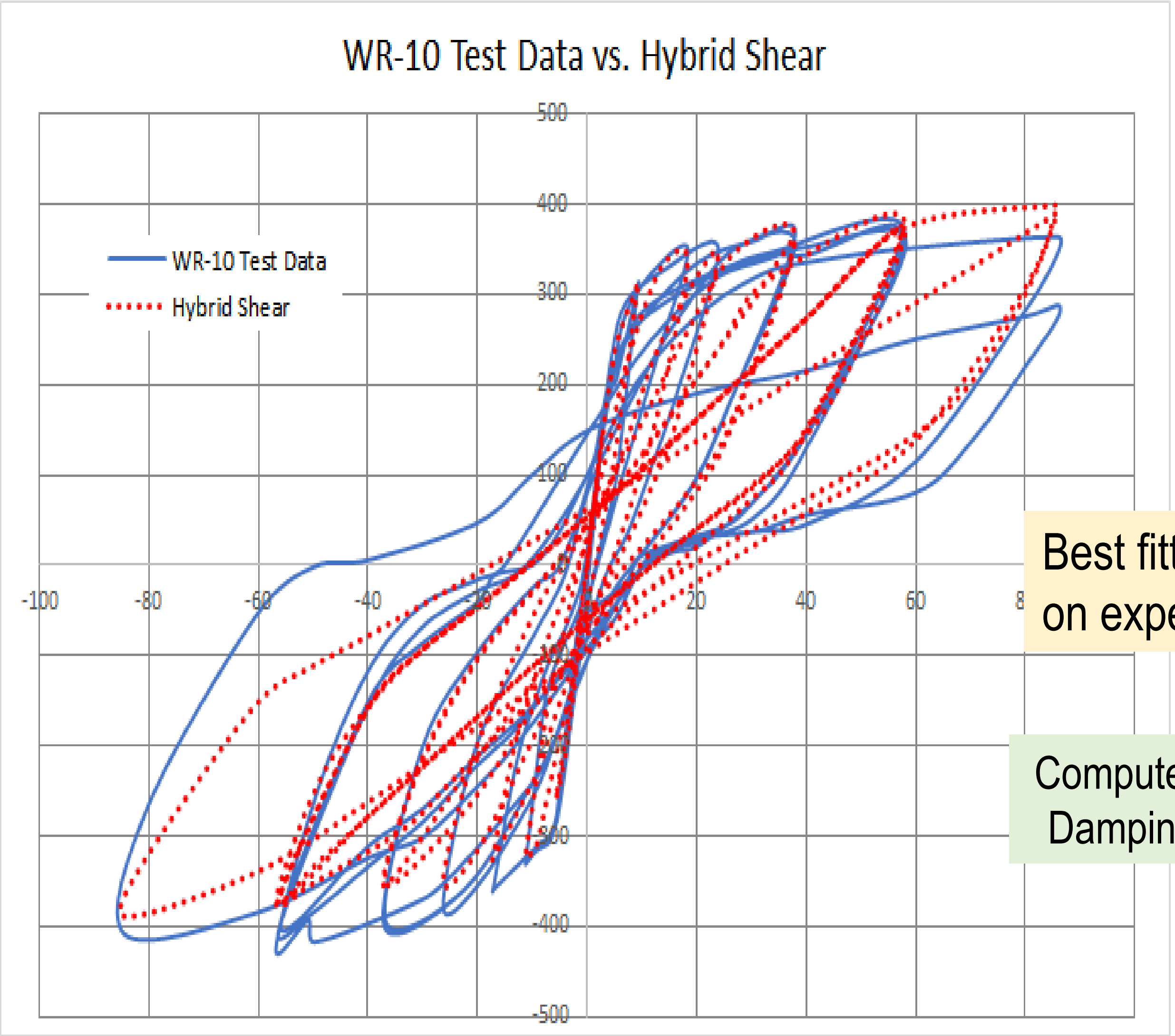




Computed Effective Damping = 6-7%

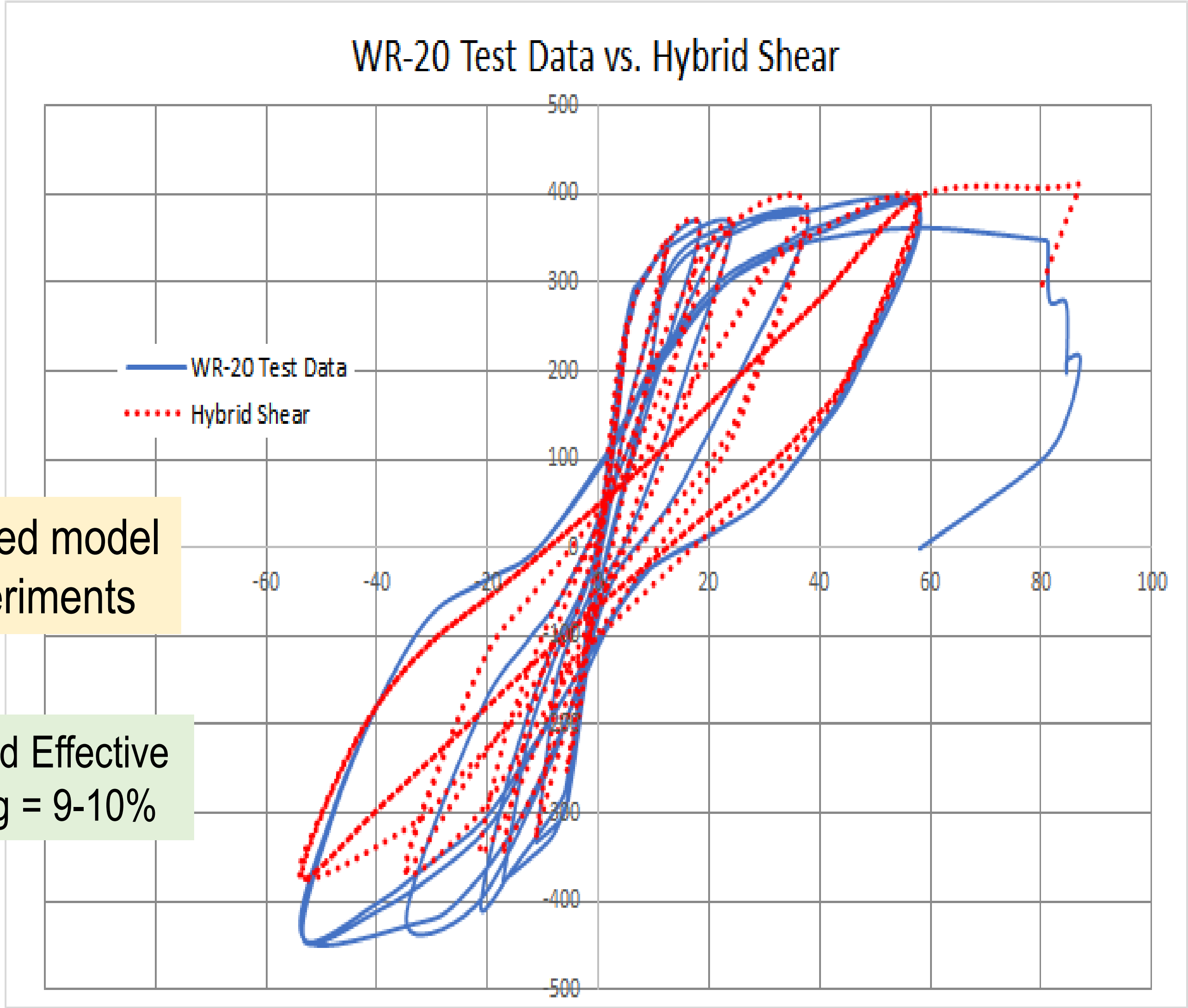
Hybrid Shear Hysteretic Model Against HU Wall Test Data

Hybrid Shear Model (Model 7)

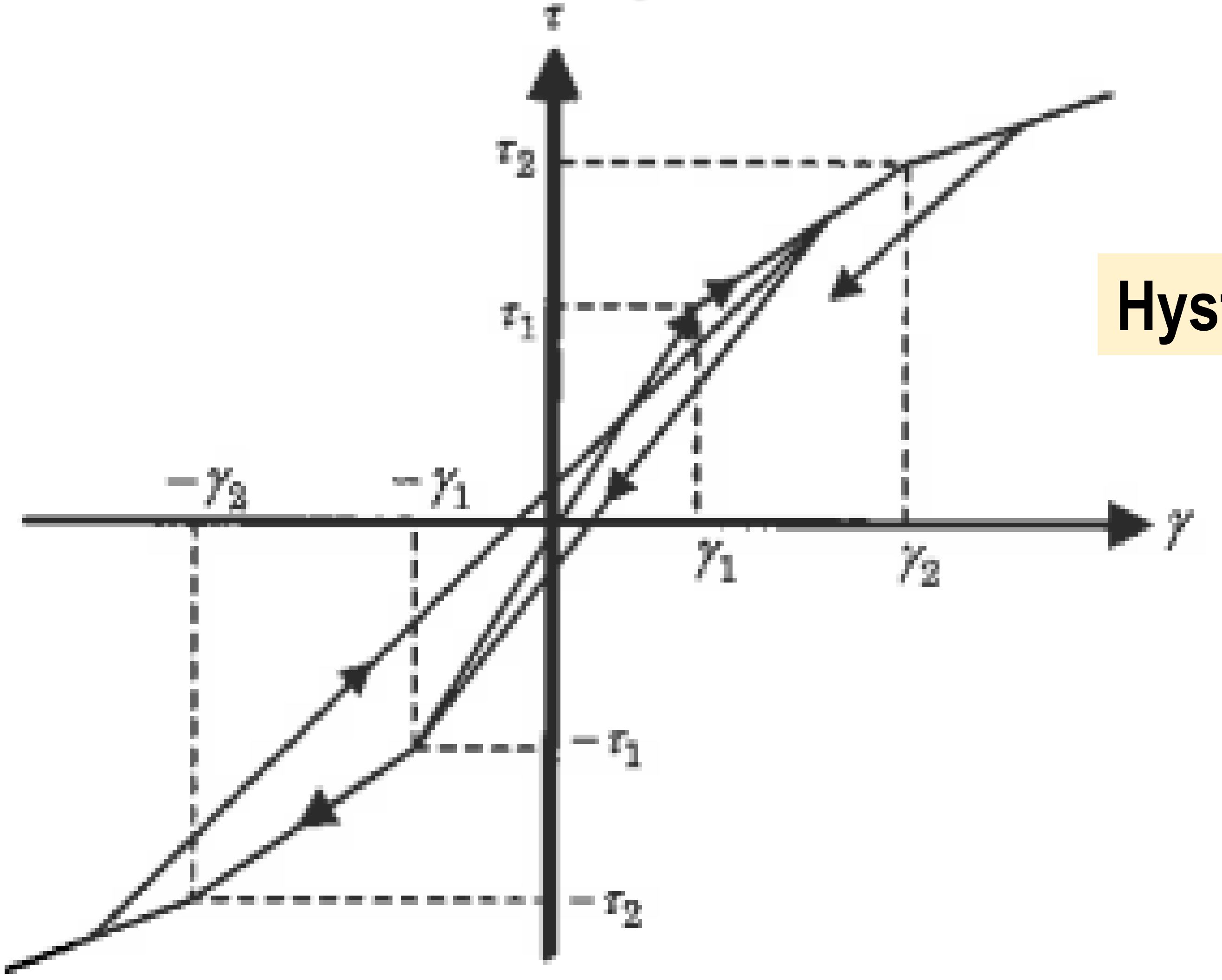


Best fitted model on experiments

Computed Effective Damping = 9-10%

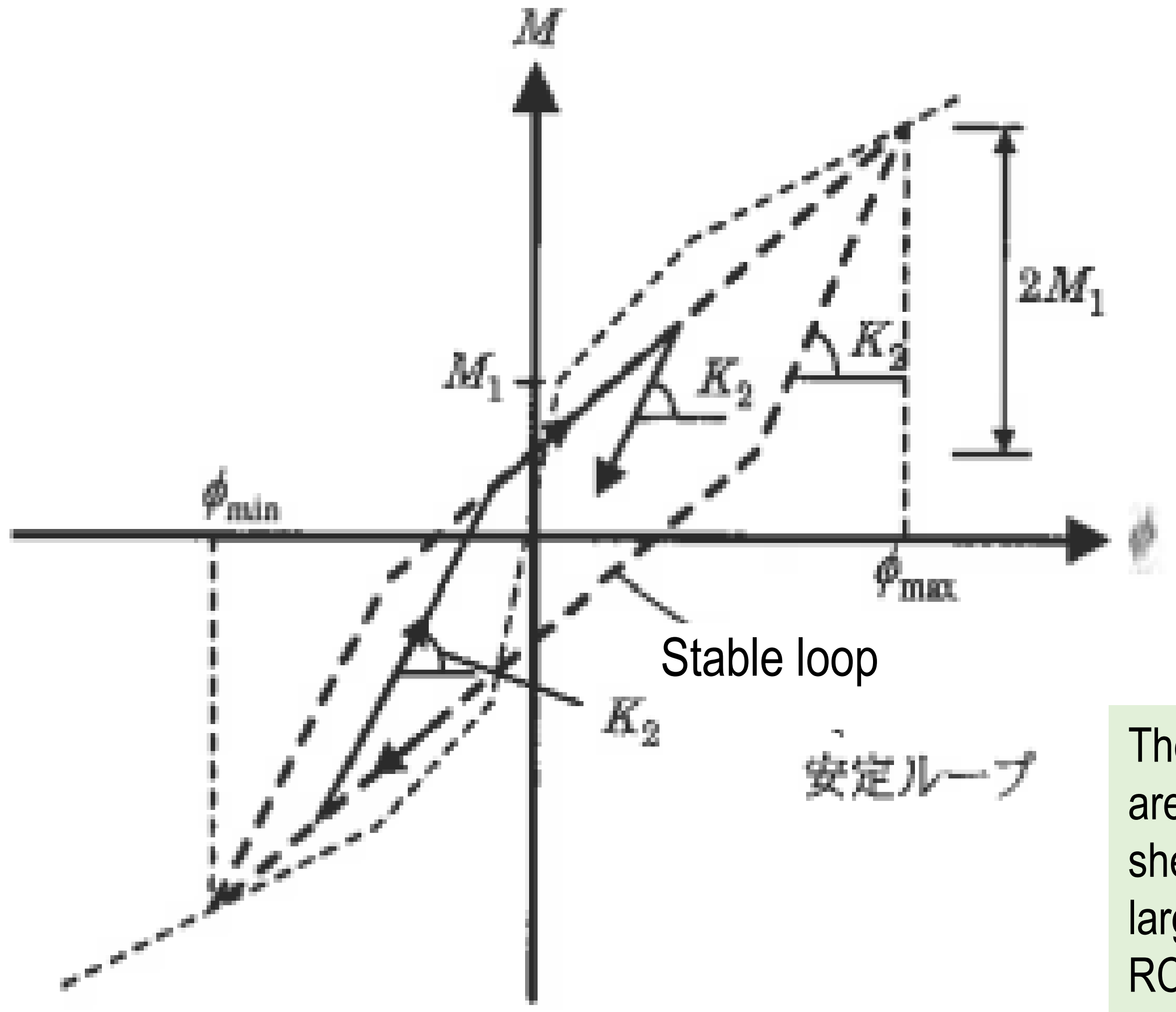


Remarks for JEAC 4601 Point-Oriented (PO) Shear Hysteretic Model

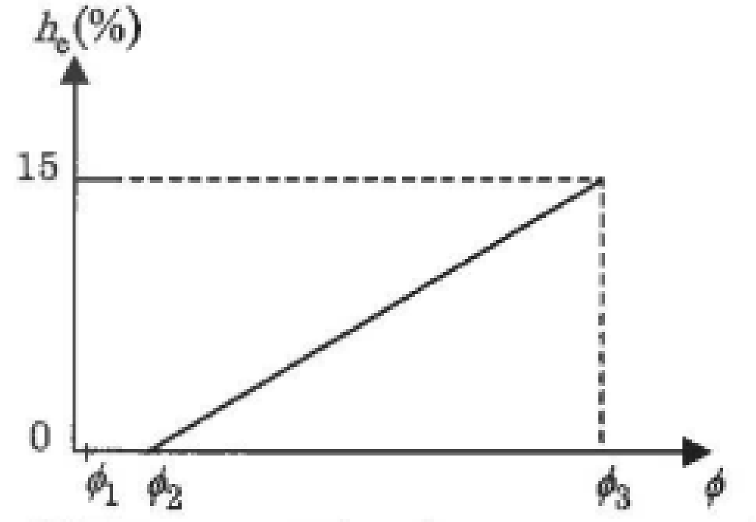


Hysteretic Damping is zero!

Remarks for JEAC 4601 Point-Oriented-Degraded-Trilinear (PODT) Bending Hysteretic Model

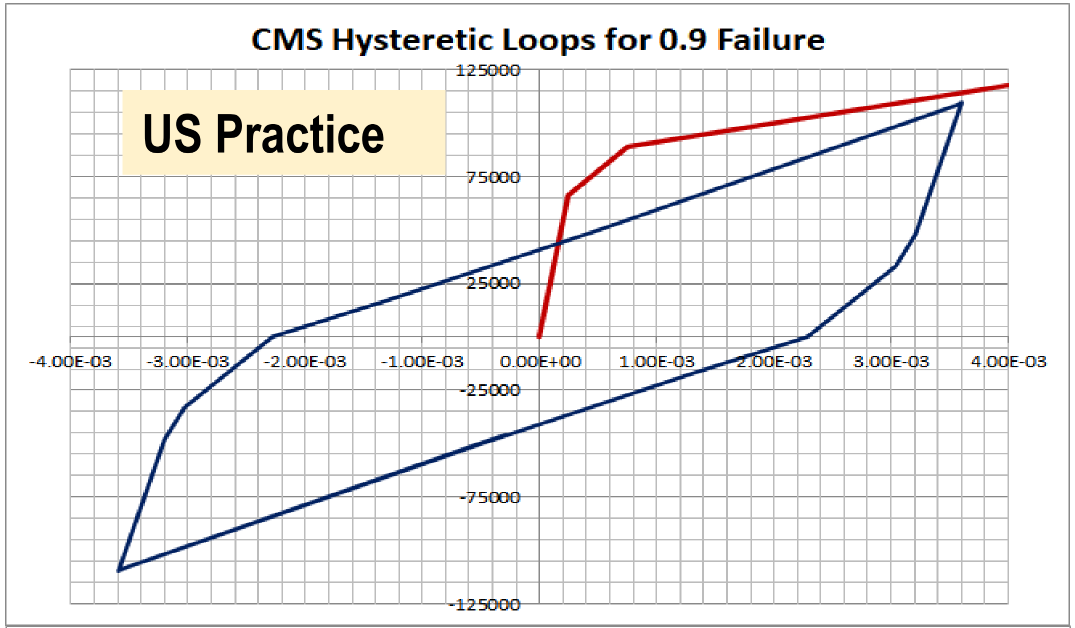


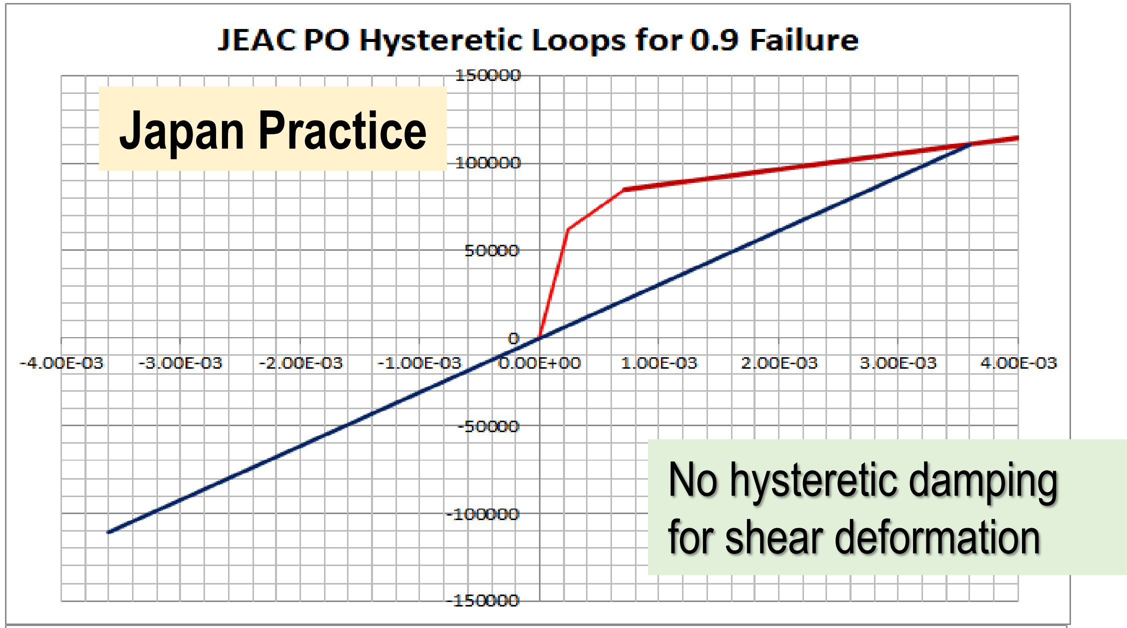
Hysteretic Damping varies from 0% to 15%; 0% at yielding and 15% at failure (ultimate).

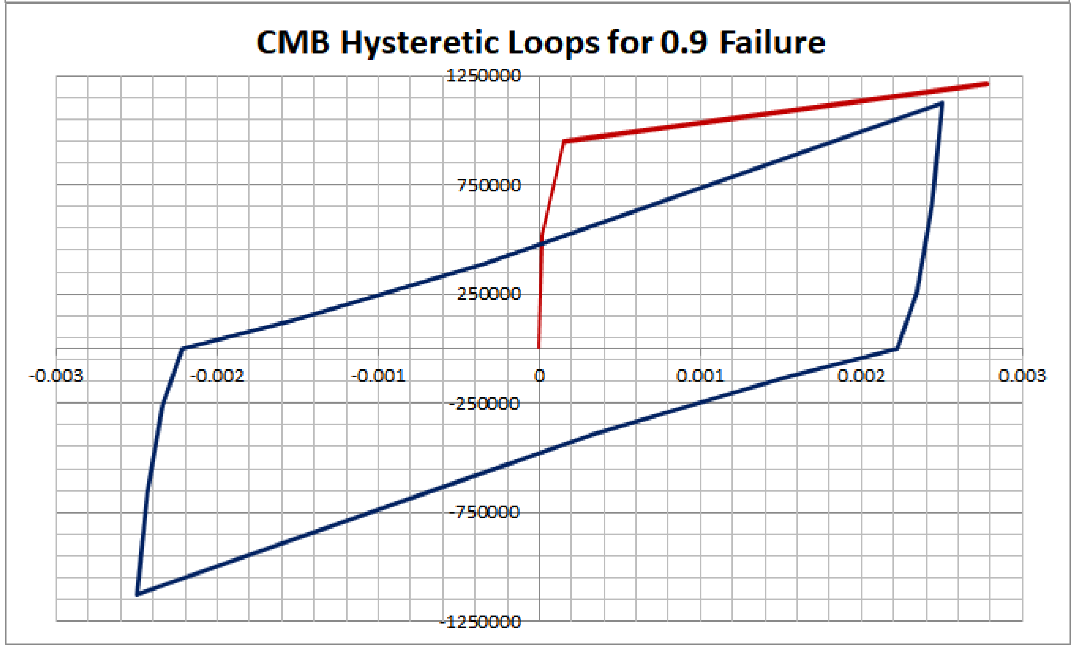


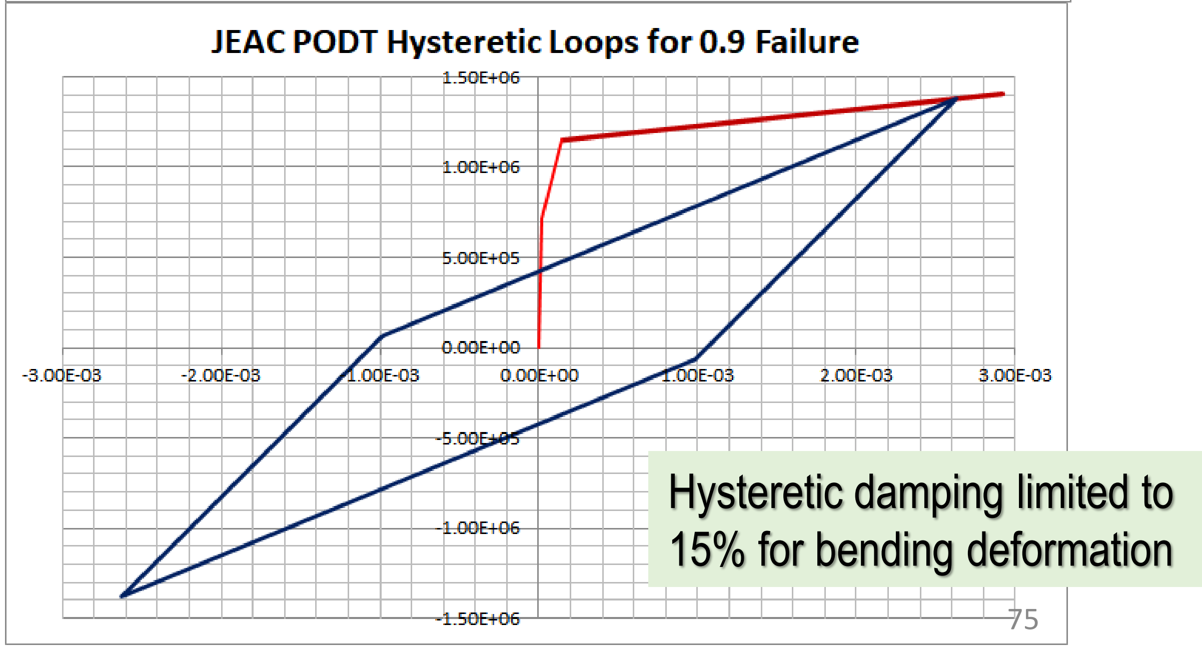
The low hysteretic damping values recommended in the JEAC 4601 are based on a series of experimental tests done for various shearwall configurations and typical NPP structure RC walls with larger thicknesses and reinforcement percentages than those of the RC walls in conventional structures (Taitokui report, 1987). These damping values are lower than those computed using FEA codes.

CM & JEAC 4601 PO Model Hysteretic Loops for Harmonic Inputs

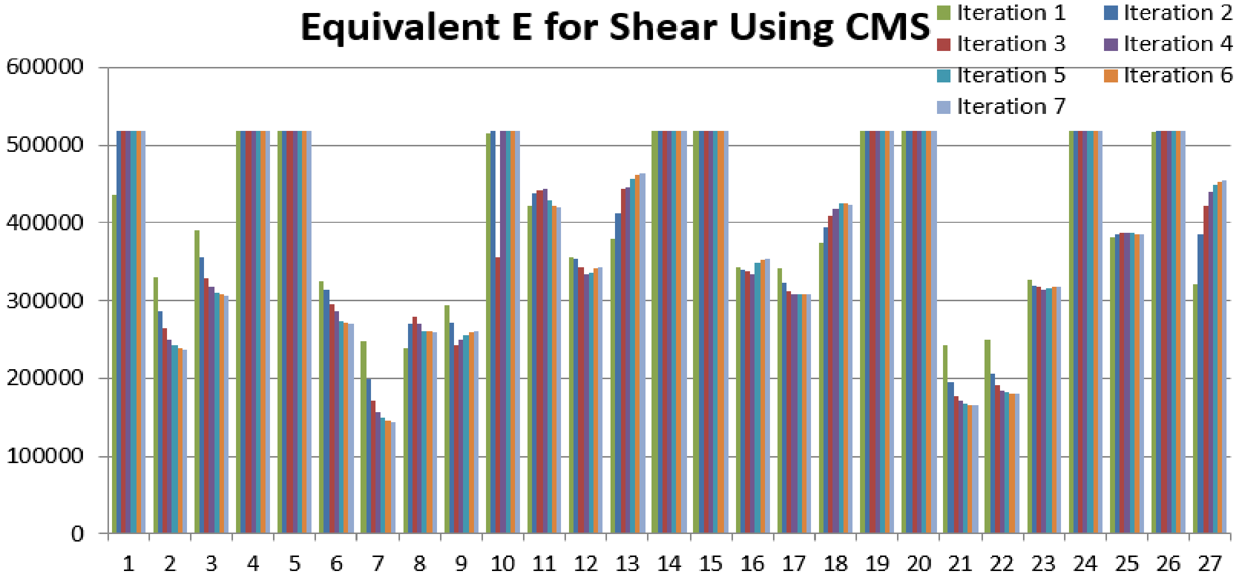


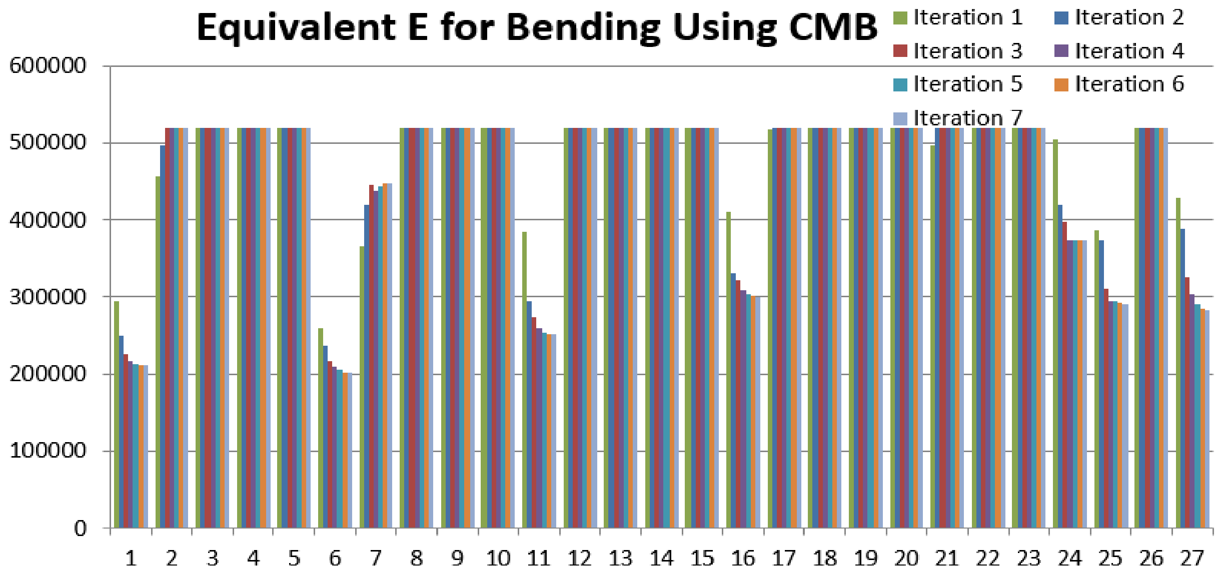


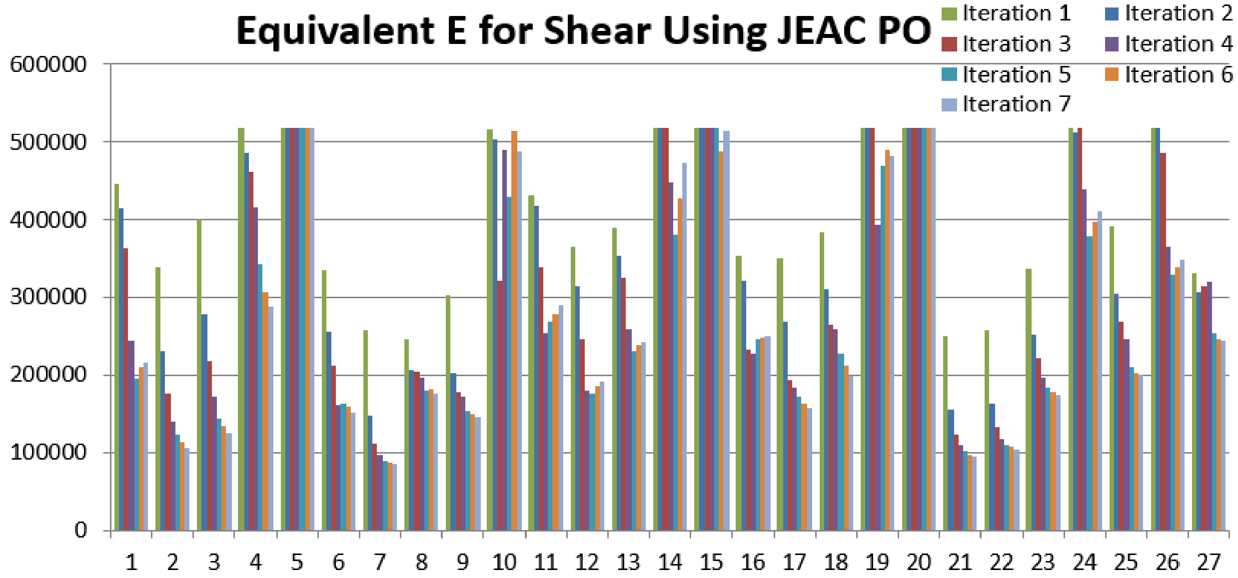


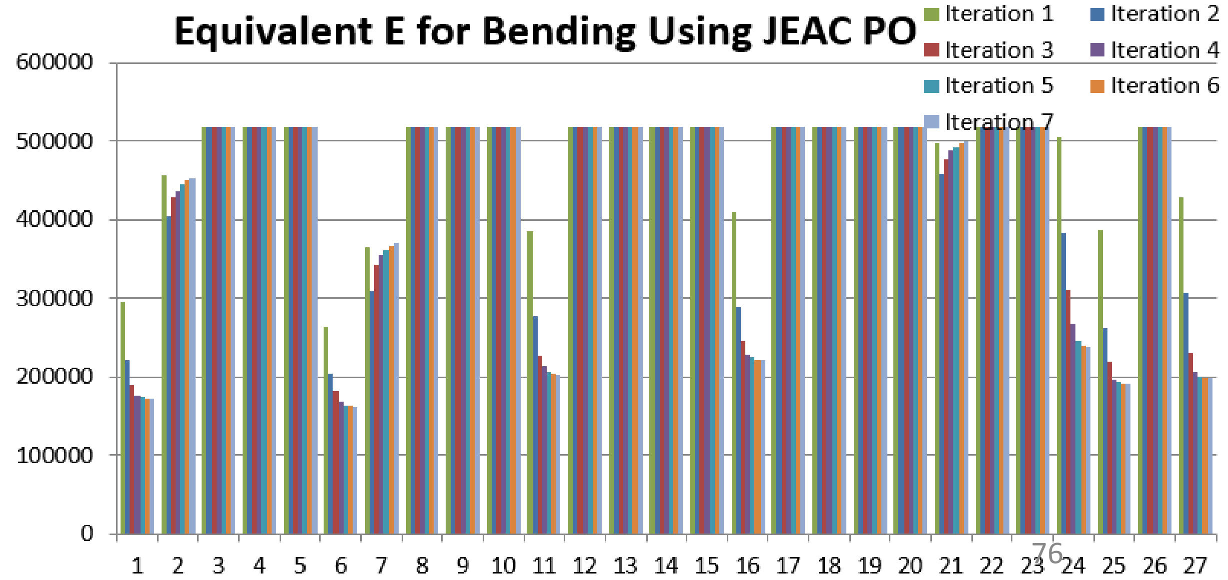


Comparisons of JEAC and ACI/ASCE 4/43 Model SSI Results from Separate Nonlinear SSI Analyses (Including Damping Effects)



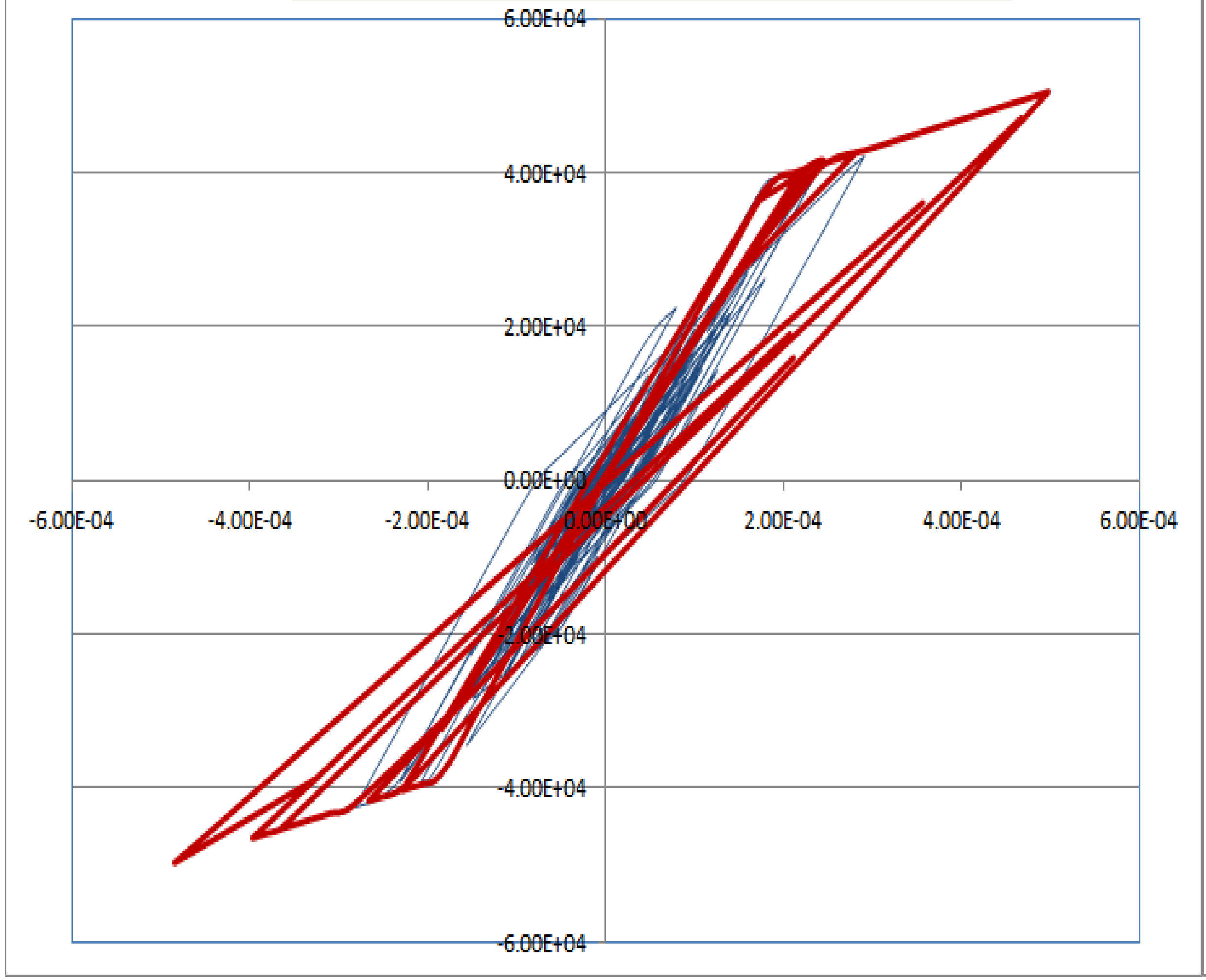




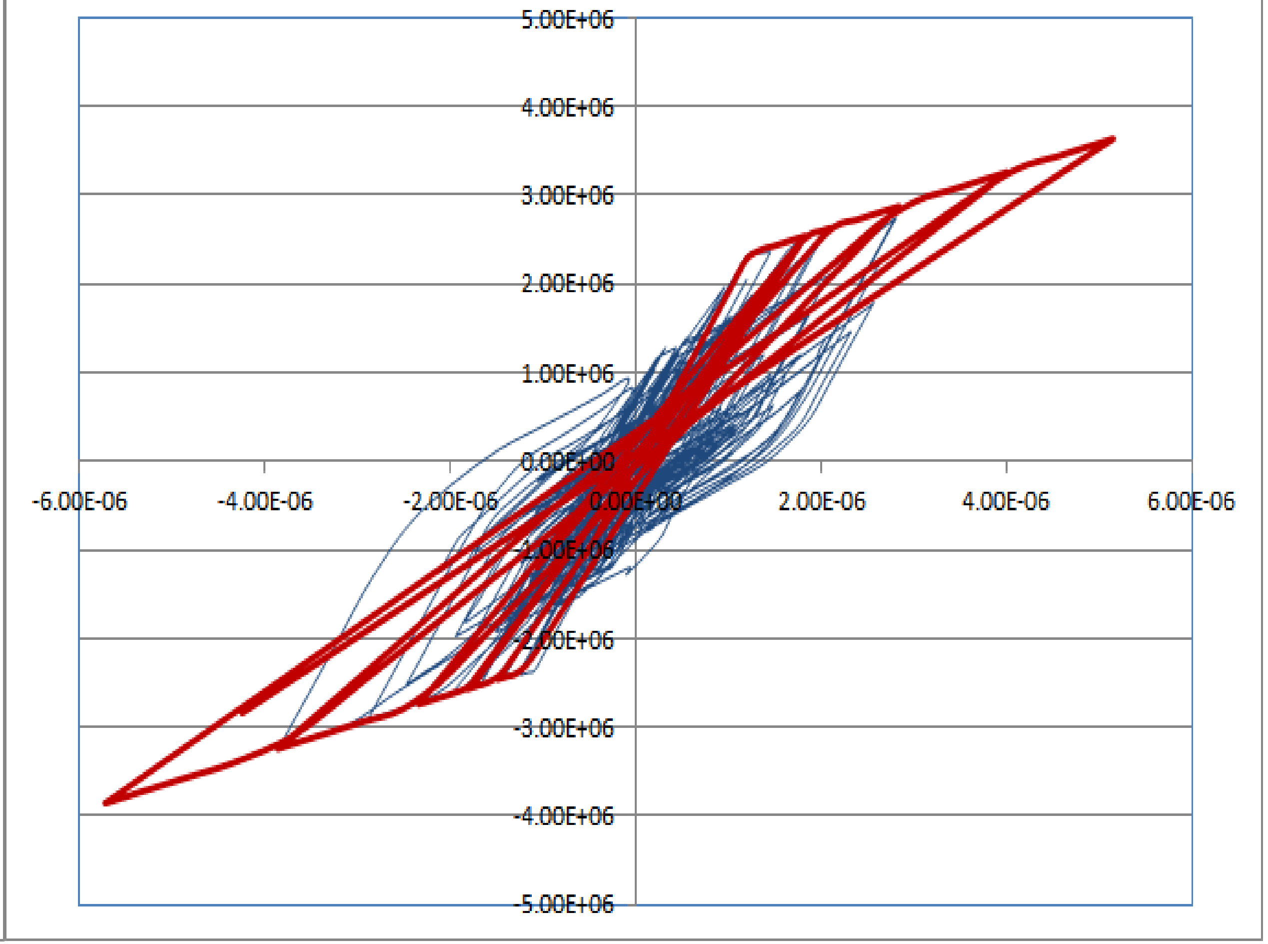


Comparisons of JEAC and ACI/ASCE 4/43 Model Loops Based on Separate Nonlinear SSI Analyses (Different Dynamic Effects)

Panel 11 Shear for CMS vs. JEAC PO



Panel 11 Bending for CMB vs. JEAC PODT



Nonlinear X, Y and Z Responses at Each SSI Iteration

In the current Option NON there is no hysteretic model for handling axial deformation in wall panels under vertical uniform forces. It should be noted that new ASCE 4-16 recommends to reduce only the shear and bending wall stiffnesses due to the concrete cracking, while the axial stiffness remains unchanged. The structure behaves nonlinearly under the horizontal input components and linearly elastic under the vertical seismic component.

The horizontal and vertical displacements computed at the corners of each wall panel shall include for each SSI iteration, the combined effects of the three seismic input components. This is achieved by using the COMB_XYZ_THD auxiliary program that is automatically included in the batch run file generated by the NONLINBAT, 1. The COMB_XYZ_THD.inp text file that is the input of the COMB_XYZ_THD auxiliary should be defined by the user (see example file for the COMB_XYZ_THD auxiliary program included on the installation DVD).

Option NON Simple:

For Low-Rise Shearwall Structures Dominated by Shear Deformation in RC Walls

(NONLINEAR Module)

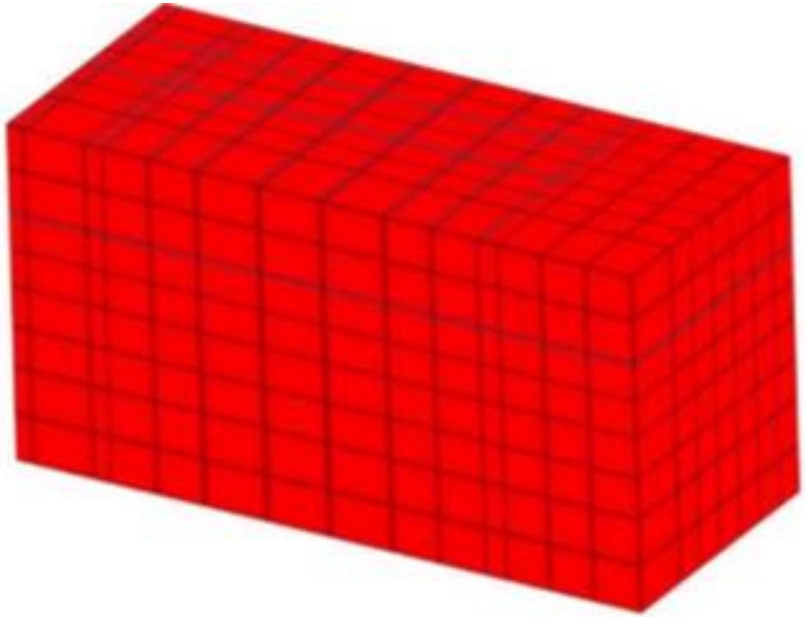
Applicable to Low-Rise Shearwall Structures

Based on the hysteretic behavior of each wall panel, the local equivalent-linear properties are computed after each SSI iteration. The stiffness reduction is applied directly to the elastic modulus for each panel. This implies, under the isotropy material assumption, that the shear, axial and bending stiffnesses suffer the same level of degradation. Poisson ratio is considered to remain constant.

The wall panel shear stiffness modification as a result on nonlinear behaviour is fully coupled with the bending stiffness. This is a reasonable assumption *only f*or the low-rise shearwalls for which the nonlinear behaviour is governed by the shear deformation, while bending effects play an insignificant role.

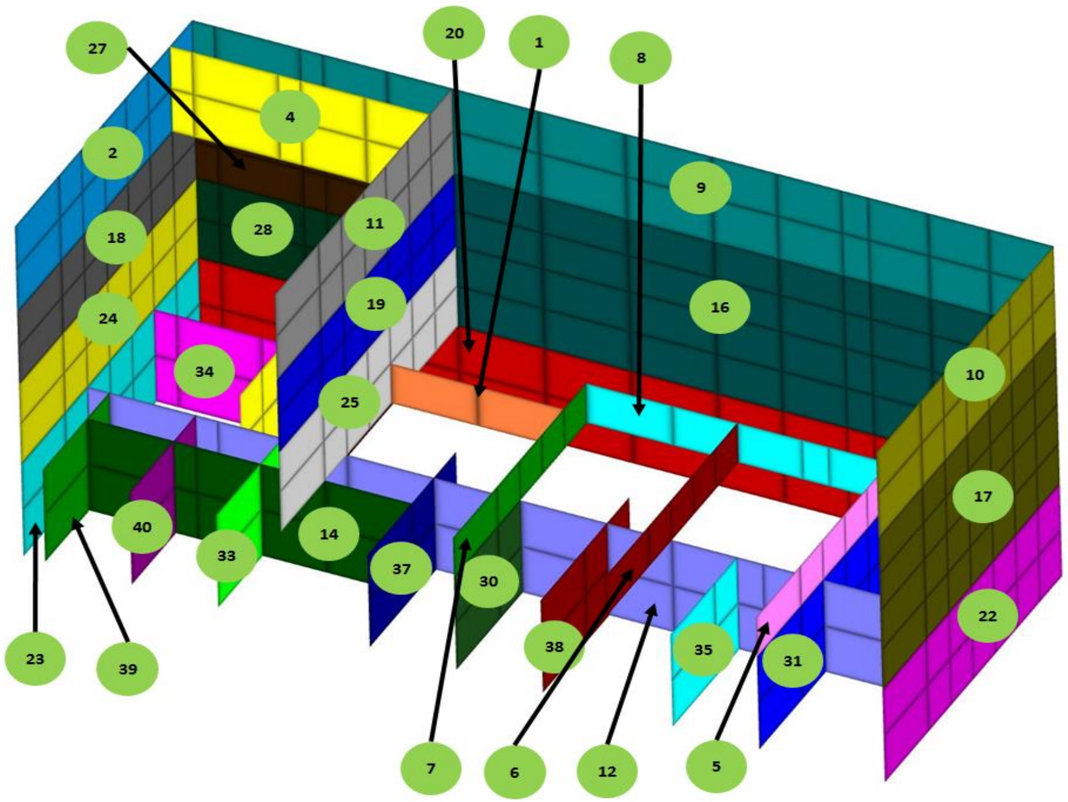
Based on various experimental tests done at Cornell University, Gergely points out in NUREG/CR 4123, 1984 that in the low-rise walls such as those that occur in the modern nuclear power plants, the flexural distortions and associated vertical yielding play a negligible role. This was also recognized by many other research studies, including the EPRI report on "Methodology for Developing Seismic Fragilities" (Reed and Kennedy, 1994).

Nonlinear Building Model Split in Simple Wall (Shear) Panels

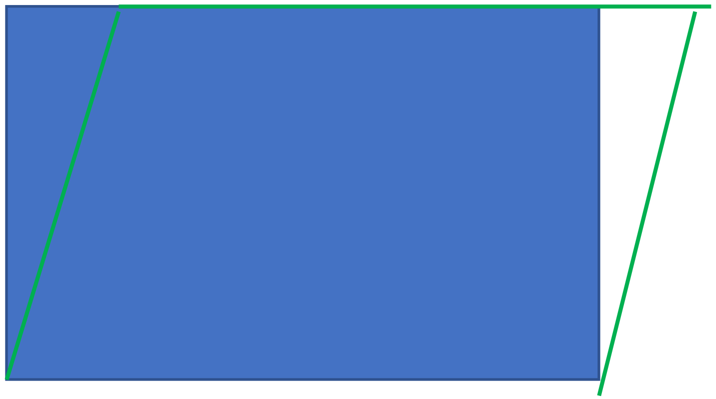


Nuclear building model split in nonlinear panels with different nonlinear properties. Many ACS SASSI User-Interface commands are available: WALLFLR, SPLITWALL, SEGWALLS, etc.

Each panel should be described by its elastic properties, BBC and hysteretic model for in-plane shear or bending deformation (Cheng-Mertz for Shear and Bending, and Takeda)



Experimental Test-Based Shear Wall Capacities for Squats

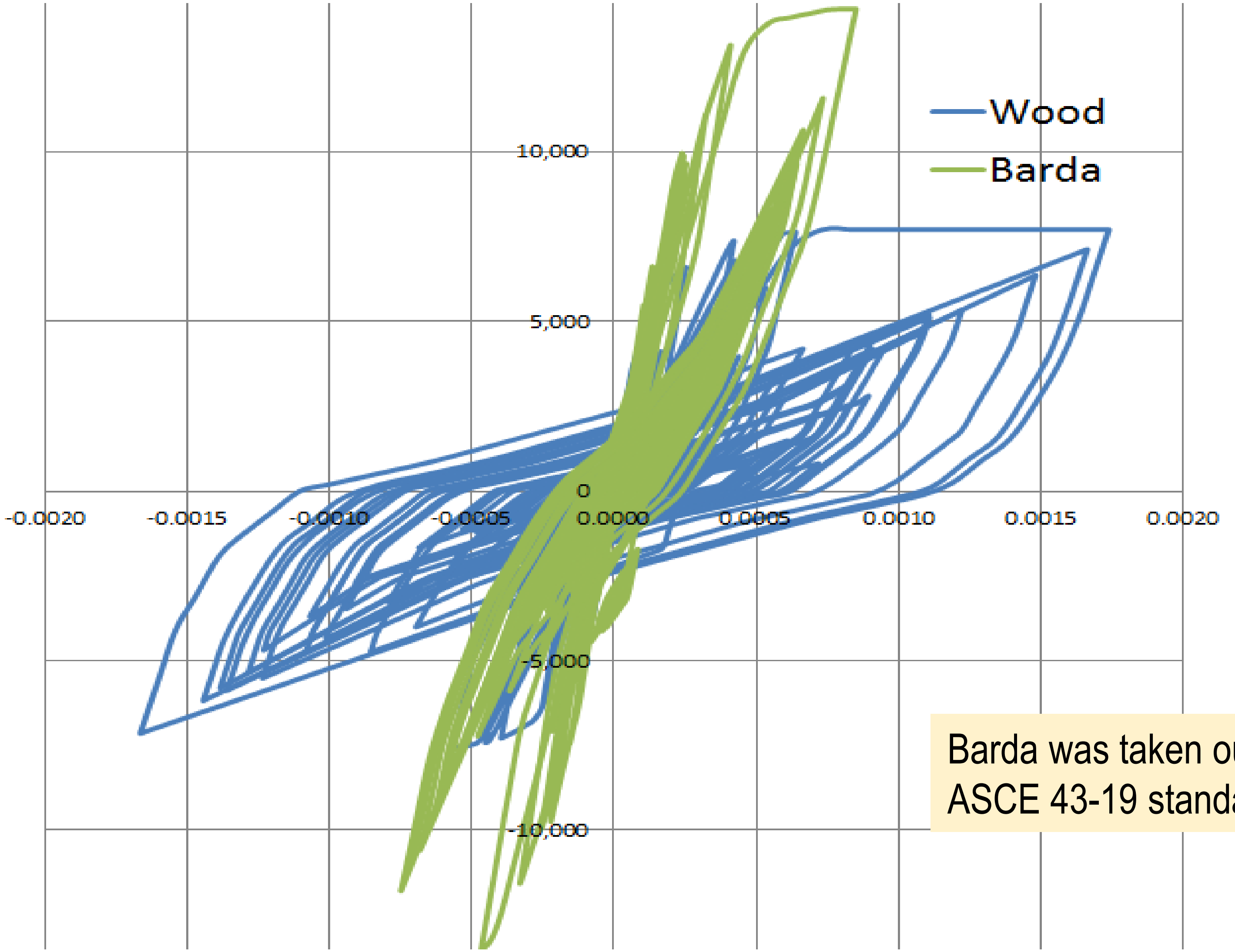


Walls have no openings!

Useful References for Peak Capacity Equations:

- Barda et al.,1977 in the 1994 EPRI Reports – could overly estimate
- ACI 349, 2006, Section 11.10, 21.4, based on Barda
- Wood, 1990 – small bias, typically less 10% lower, for median capacity
- Gulec and Whittaker, 2009, Eqs. 6.9-6.10, small bias for median capacity

Shearwall Panel 17 Hysteretic Behavior
Barda (1977) vs. Wood (1990) for 0.60g Input



Barda was taken out of new ASCE 43-19 standard draft

Automatic Generation of Backbone Curves (BBC)

BBCGEN,<Panel>,<Shear>,[fc],[fy],[Pn],[….], [CrackingForceLevel]

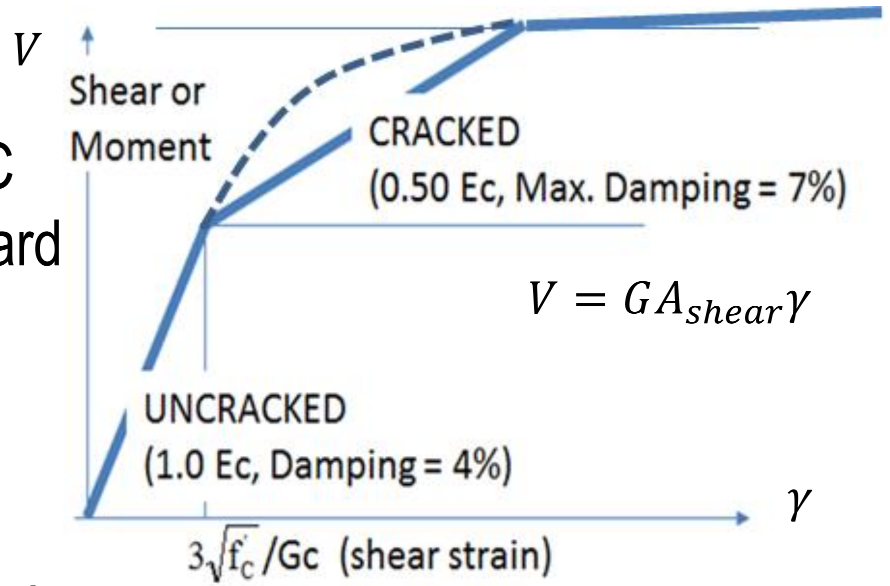
<Panel>

= 0, the BBC curves will be generated for all panels defined by the user assuming the same command parameters. The Panel = 0 option, it can be used for submodels to define properties of panels in a subset.

= K, the BBC will be generated only for Panel K.

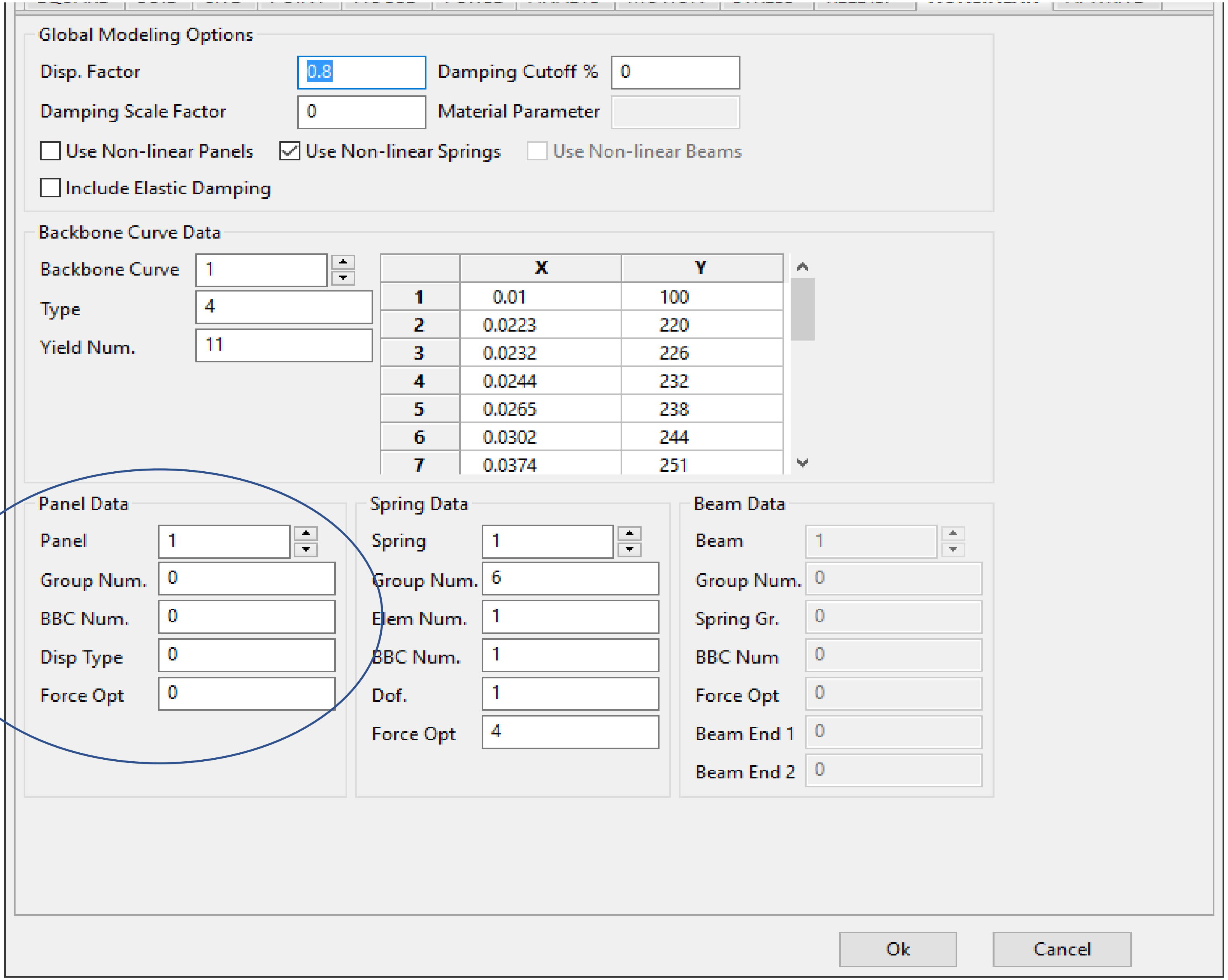
[CrackForceLevel]

= 0. Default option for building BBC curves uses new ASCE 4-16 standard recommendation in Section C.3.3.2 for defining the concrete cracking stress level by the value of $3\sqrt{f'_c}$



= Vcr/Vu alue in the [0.1 0.5] interval. Uses the cracking shear/ultimate shear force ratio to build the BBC curves.

Nonlinear Module Input Window



P Command

P,<num>,<group>,<bbc>,<disp>,<force>

This Option NON command defines wall panels for nonlinear structure SSI analysis.This command associates the SSI model data with a finite shell element group to create the wall panel. This command does not define any new groups or elements and no linear SSI model information is changed by this command. All shells in a panel group should be coplanar. Coplanar shell groups can be created by using the WALLFLR command.

- num - panel number
- group - group number
- bbc - back bone curve number
- disp - displacement type
- force - force option

WALLFLR Command (No Parameter)

The WALLFLR command will take the current active model and delete all of the non shell elements in the model. Then the command will attempt to separate all of the shells into different wall and floors groups based on a coplanarity test of the shell elements. If 5 or more elements are found to be coplanar, then these elements will be put into a new group. All of the elements that were not put into wall or floor groups because there were not enough coplanar shells to form a new wall or floor in a separate group.

SPLITWALLS Command (No Parameter)

This command splits walls (shell groups that are not perpendicular to the global Z axis) by using intersections with other floors (shell groups that are perpendicular to the global Z axis).

This command does not change floors groups. This command should be used before SEGWALLS in most cases.

Nonlinear Structure SSI Input .Pre File (P)

AB_SHEAR_NL.pre

```
L,11,5,0.15,200000,100000,0.01,0.01
L,12,5,0.15,200000,100000,0.01,0.01
L,13,5,0.15,200000,100000,0.01,0.01
L,14,5,0.15,200000,100000,0.01,0.01
L,15,5,0.15,200000,100000,0.01,0.01
L,16,5,0.15,200000,100000,0.01,0.01
* Real Property Table
R,1,11.111,0,0,17.387,10.288,10.288
R,2,13,0,0,22.316,9.75,20.343
R,3,2.849,0,0,28.958,0.333,28.292
* NonLinear
EQL,0.8,1,0,1,1
P,1,3,1,1,1
P,2,8,2,1,1
P,3,9,3,1,1
P,4,10,4,1,1
P,5,11,5,1,1
P,6,12,6,1,1
P,7,13,7,1,1
P,8,14,8,1,1
P,9,15,9,1,1
P,10,16,10,1,1
P,11,17,11,1,1
P,12,18,12,1,1
P,13,19,13,1,1
P,14,20,14,1,1
P,15,21,15,1,1
P,16,22,16,1,1
P,17,23,17,1,1
P,18,25,18,1,1
P,19,26,19,1,1
P,20,27,20,1,1
P,21,28,21,1,1
```

PANELGEN command

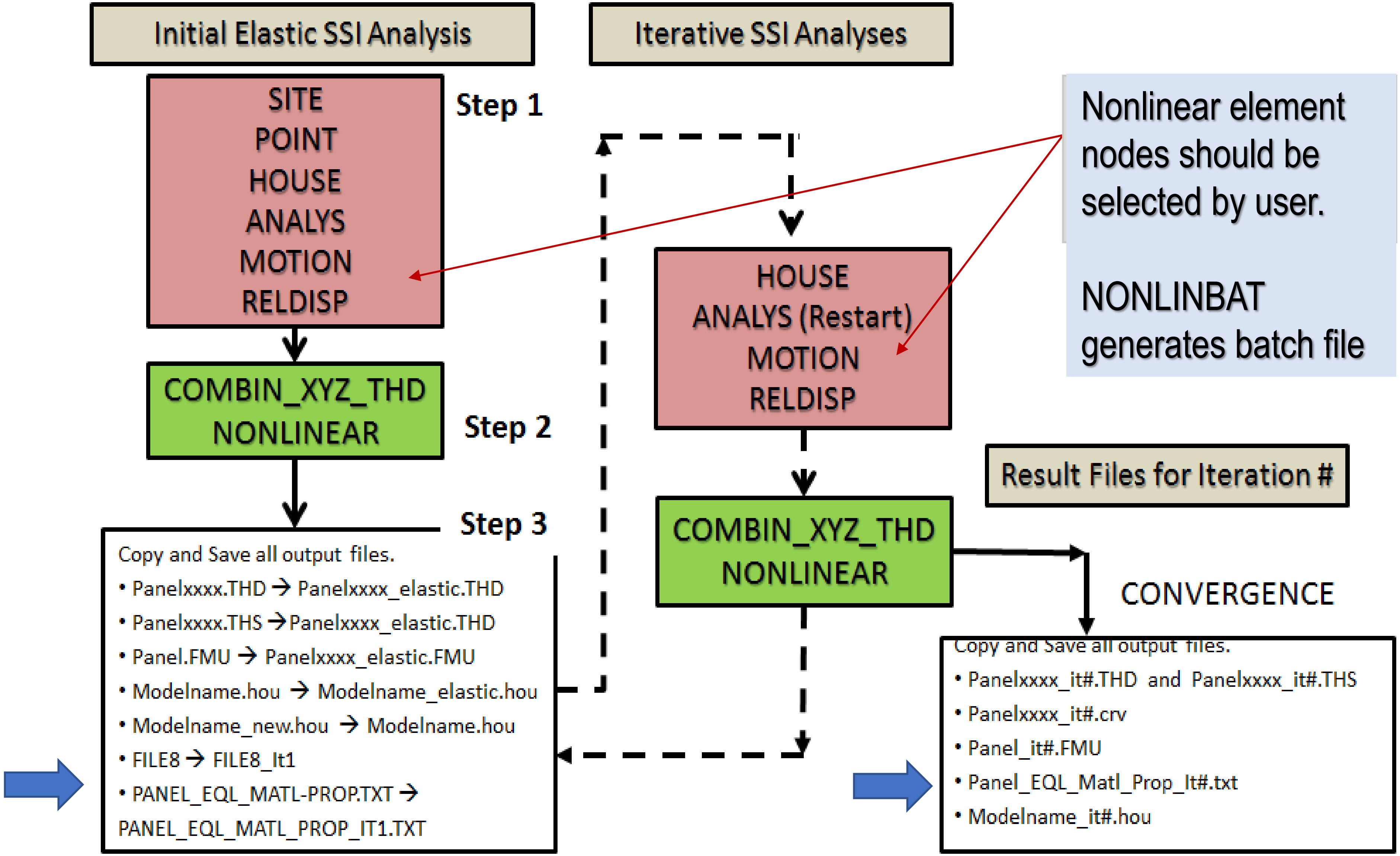
Nonlinear Structure SSI Input .Pre File (BBCP)

AB_SHEAR_NL.pre

```
P,106,113,106,1,1
P,107,114,107,1,1
P,108,115,108,1,1
P,109,116,109,1,1
P,110,117,110,1,1
P,111,118,111,1,1
P,112,119,112,1,1
P,113,120,113,1,1
BBCI,1,21,1
BBCP,1,1,0.00013825,2415.18
BBCP,1,2,0.000152075,2650.66
BBCP,1,3,0.0001659,2874.06
BBCP,1,4,0.000179724,3085.39
BBCP,1,5,0.000193549,3284.65
BBCP,1,6,0.000207374,3471.82
BBCP,1,7,0.000221199,3646.92
BBCP,1,8,0.000235024,3809.95
BBCP,1,9,0.000248849,3960.9
BBCP,1,10,0.000262674,4099.77
BBCP,1,11,0.000276499,4226.57
BBCP,1,12,0.000290324,4341.29
BBCP,1,13,0.000304149,4443.93
BBCP,1,14,0.000317974,4534.5
BBCP,1,15,0.000331799,4612.99
BBCP,1,16,0.000345624,4679.41
BBCP,1,17,0.000359449,4733.75
BBCP,1,18,0.000373274,4776.02
BBCP,1,19,0.000387099,4806.21
BBCP,1,20,0.000400924,4824.32
BBCP,1,21,0.000414749,4830.36
BBCP,1,22,0.02,4926.97
BBCI,2,21,1
BBCP,2,1,0.00013825,805.06
BBCP,2,2,0.000152075,883.554
BBCP,2,3,0.0001659,958.022
BBCP,2,4,0.000179724,1028.46
```

BBCI and BBCP Commands

Option NON Simple Application (NONLINEAR Module)



PANEL_EQL_MATL_PROP_IT# Text Files; Iteration 1,4,6

PANEL_EQL_MATL_PROP_IT1 - Notepa

File Edit Format View Help

00052	519100	0.040000
00053	444427	0.068024
00054	519100	0.040000
00055	428596	0.070299
00056	399598	0.075423
00057	357810	0.086960
00058	296877	0.099000
00059	519100	0.040000
00060	519100	0.040000
00061	519100	0.040000
00062	519100	0.040000
00063	519100	0.040000
00064	427612	0.070436
00065	430509	0.070032
00066	329360	0.099000
00067	497448	0.051490
00068	495021	0.052083
00069	486163	0.054532
00070	519100	0.040000
00071	492199	0.052772
00072	519100	0.040000
00073	332062	0.099000
00074	433258	0.069637
00075	428142	0.070362
00076	423965	0.070945
00077	323590	0.099000
00078	519100	0.040000
00079	493437	0.052469
00080	490450	0.053245
00081	305877	0.099000
00082	519100	0.040000
00083	432717	0.069715
00084	423693	0.070983

PANEL_EQL_MATL_PROP_IT4 - Note

File Edit Format View Help

00052	519100	0.040000
00053	434881	0.069402
00054	519100	0.040000
00055	379428	0.079385
00056	363115	0.084592
00057	202701	0.122536
00058	128331	0.172728
00059	519100	0.040000
00060	519100	0.040000
00061	519100	0.040000
00062	519100	0.040000
00063	504400	0.050117
00064	420231	0.071465
00065	408232	0.073671
00066	292851	0.099000
00067	484745	0.054958
00068	482626	0.055594
00069	470562	0.059843
00070	519100	0.040000
00071	477685	0.057287
00072	519100	0.040000
00073	175308	0.139764
00074	432823	0.069700
00075	436629	0.069150
00076	428687	0.070286
00077	276456	0.099000
00078	519100	0.040000
00079	475545	0.058055
00080	475535	0.058058
00081	161111	0.149287
00082	519100	0.040000
00083	422270	0.071181
00084	434374	0.069475

PANEL_EQL_MATL_PROP_IT6 - N

File Edit Format View Help

00052	519100	0.040000
00053	451663	0.067458
00054	519100	0.040000
00055	418371	0.071724
00056	411010	0.073102
00057	174688	0.140190
00058	108539	0.188412
00059	519100	0.040000
00060	519100	0.040000
00061	519100	0.040000
00062	519100	0.040000
00063	519100	0.040000
00064	439278	0.068768
00065	421897	0.071233
00066	355619	0.087938
00067	497556	0.051464
00068	496152	0.051806
00069	489277	0.053597
00070	519100	0.040000
00071	492105	0.052795
00072	519100	0.040000
00073	149501	0.157120
00074	461457	0.064862
00075	462244	0.064421
00076	451532	0.067469
00077	331961	0.099000
00078	519100	0.040000
00079	490346	0.053276
00080	491813	0.052866
00081	135058	0.167238
00082	519100	0.040000
00083	450253	0.067567
00084	457948	0.066831

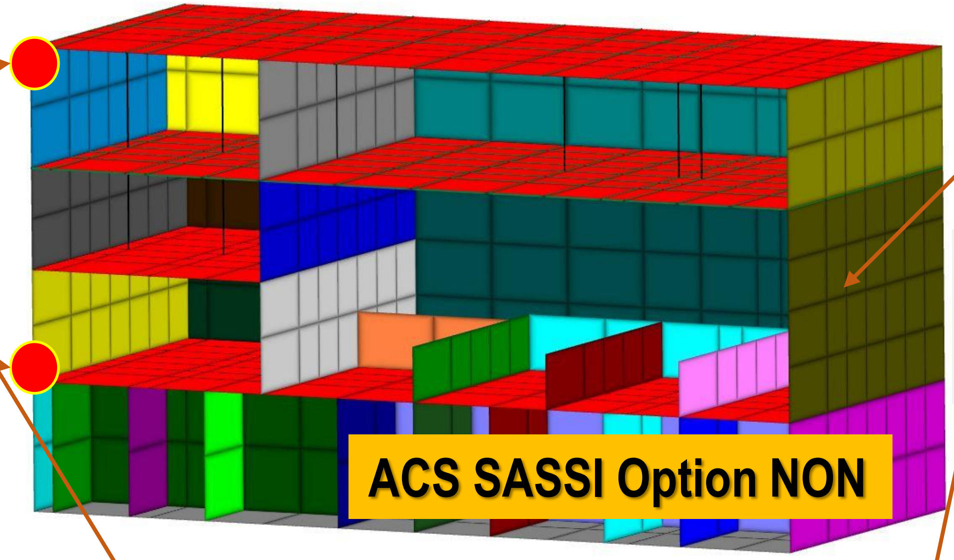
Comparative Nonlinear Results Vs. PERFORM3D Code for Low-Rise Shearwall Auxiliary Building (ABSHEAR)

Node 33

Node 243

Panel 17-Most Damaged

CM Shear Model for Wall Panels



AB is a low-rise building governed by nonlinear shear deformation effects.

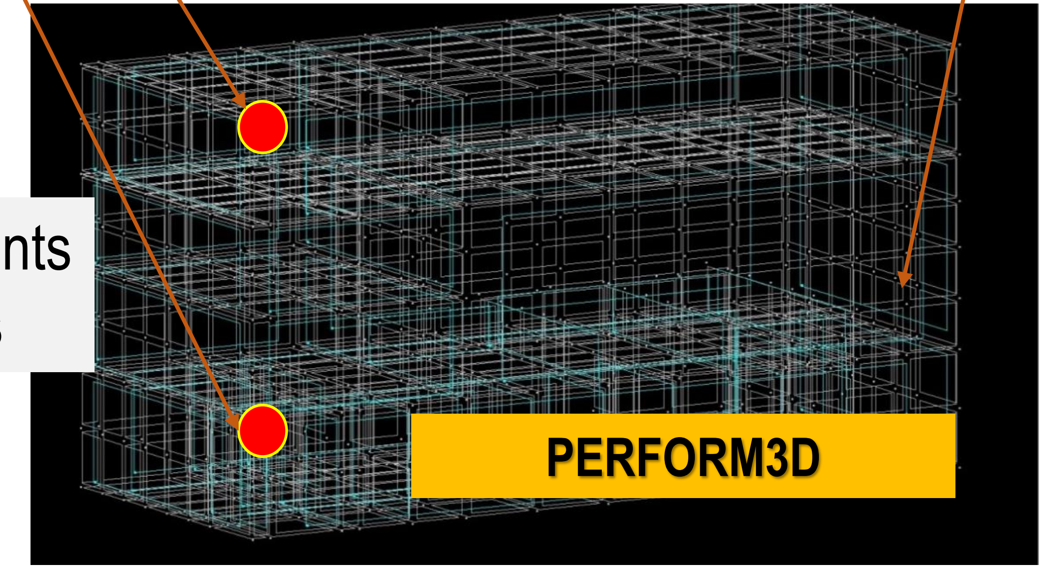
0.60g RG1.60 Input (2xDBE)

Rigid soil, fixed-base.

Same BBC for both models.

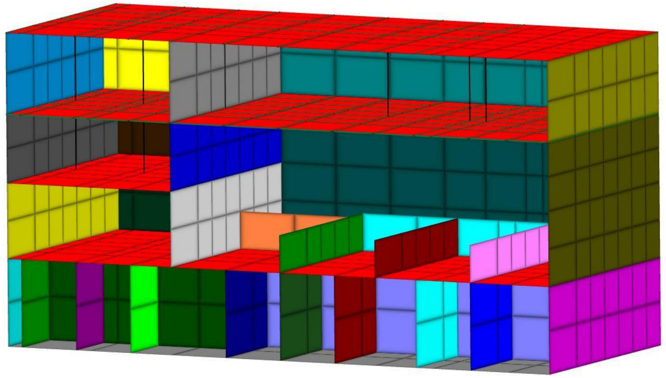
1% reinforcement for all walls

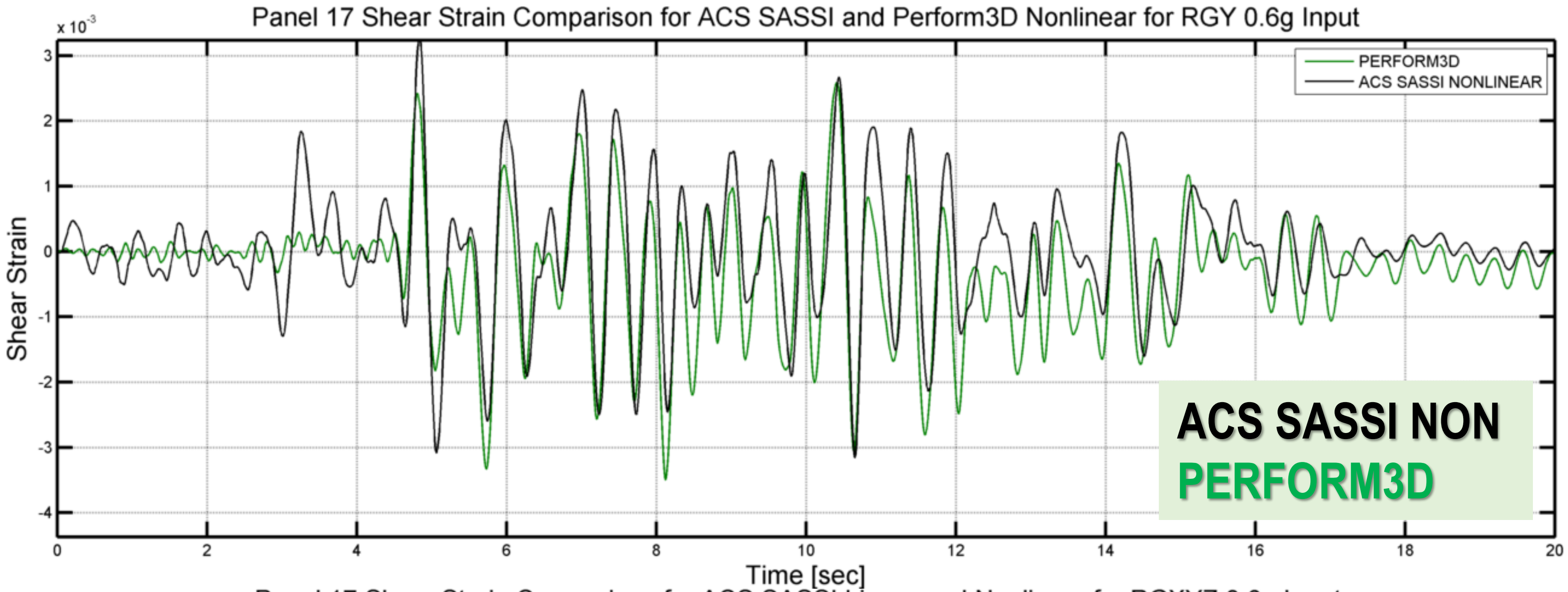
Shear Wall Elements for Wall Panels

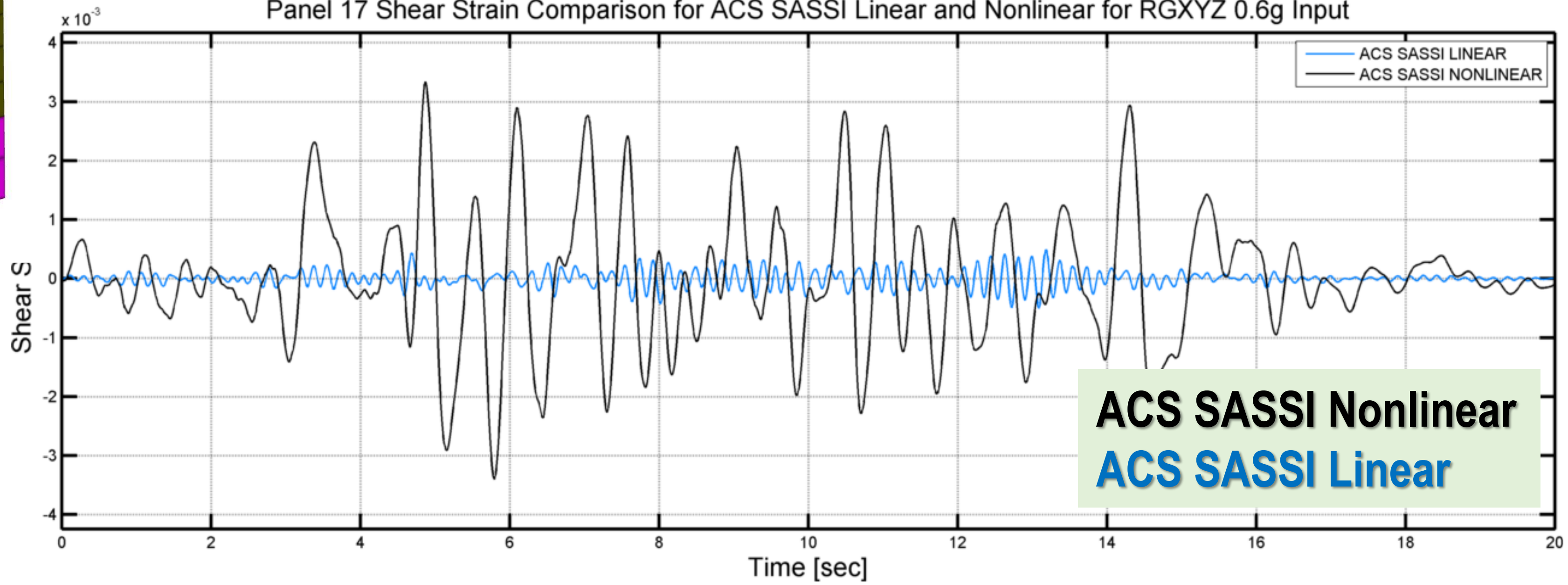


Comparative Nonlinear Shear Strain in Panel 17 for 0.60g (2xDBE)

Low-Rise Aux Building (AB)
Include Shear Effects Only

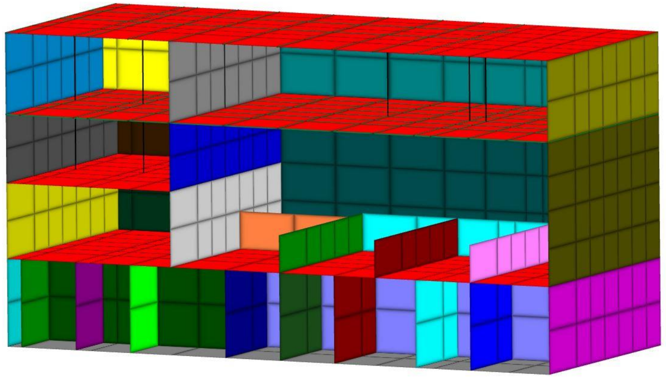




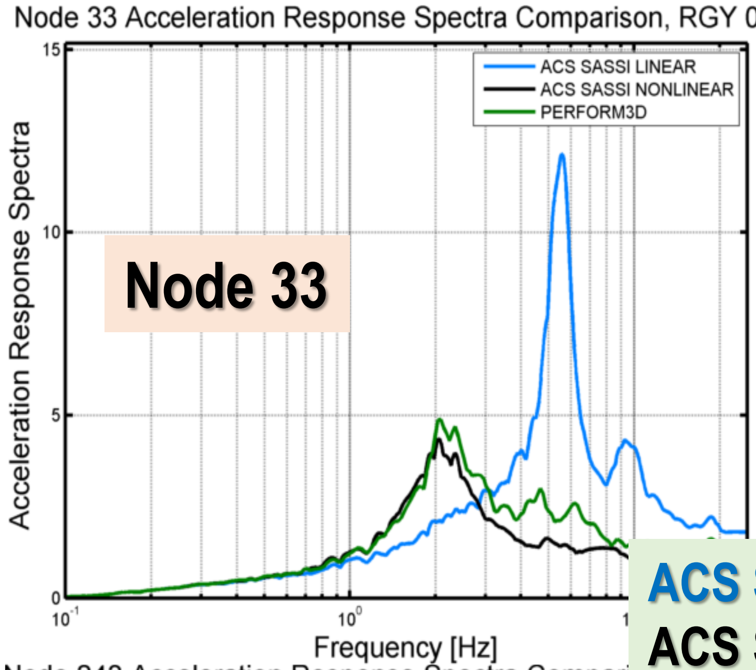


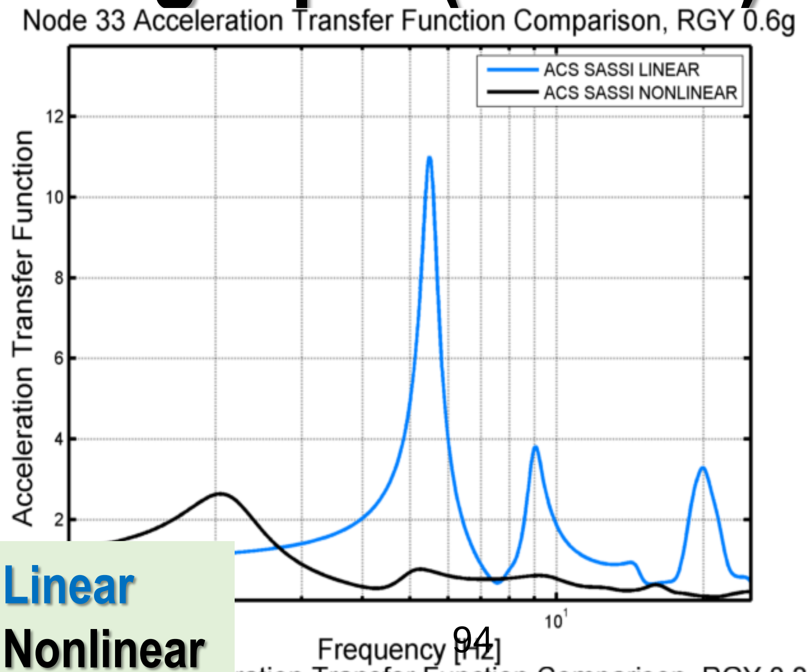
Comparative ISRS and ATF Results for 0.60g Input (2x DBE)

Low-Rise Aux Building (AB)
Include Shear Effects Only



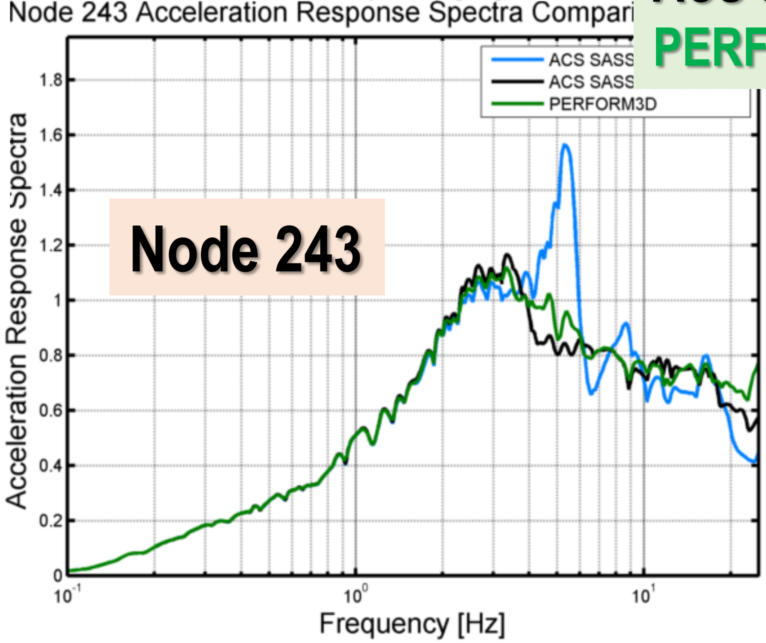
Node 33

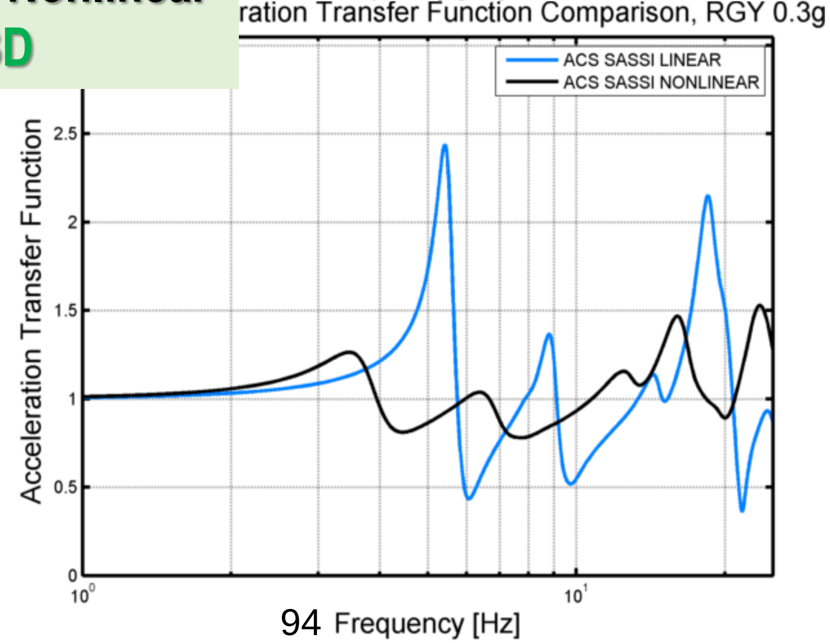




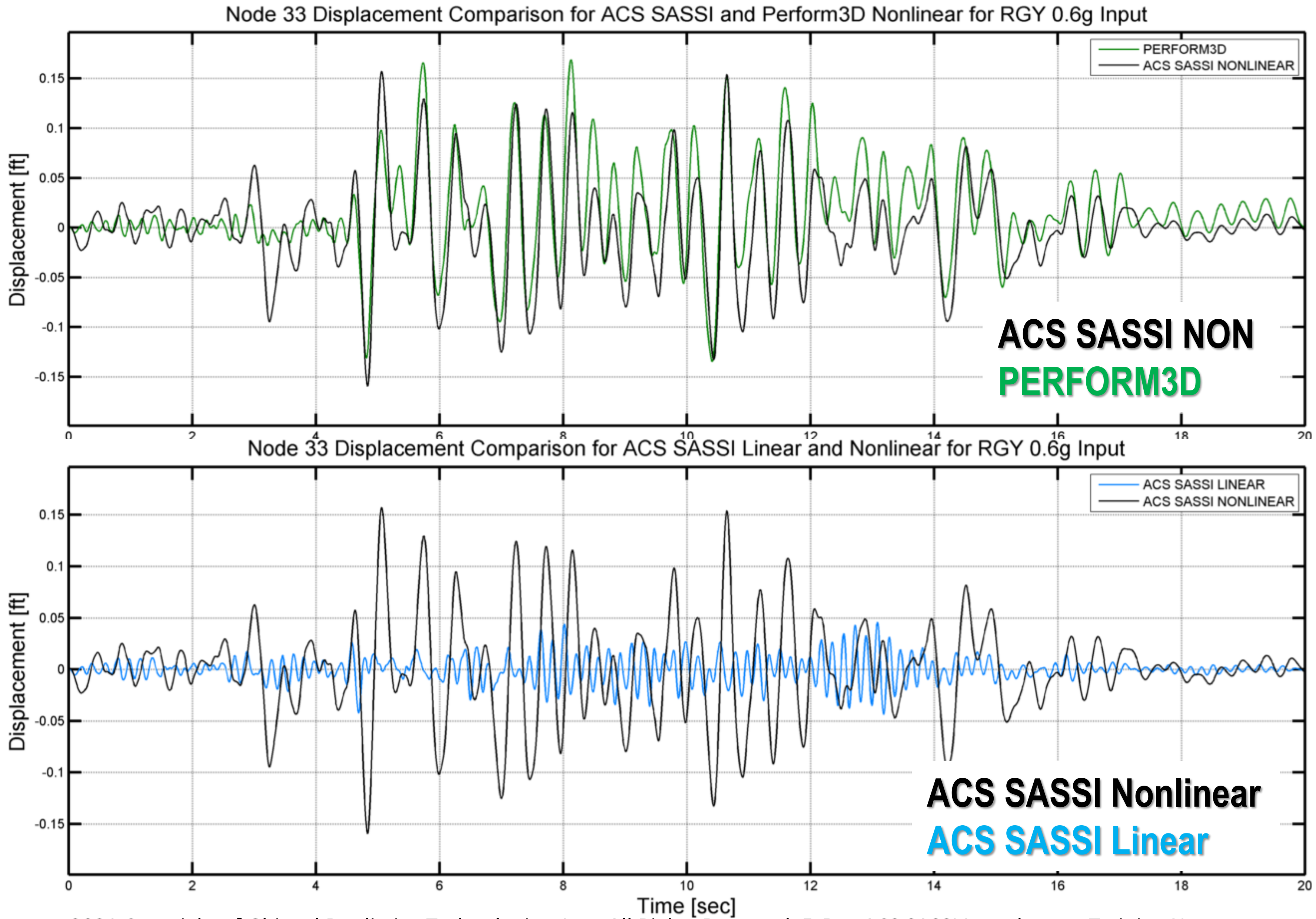
ACS SASSI Linear
ACS SASSI Nonlinear
PERFORM3D

Node 243





Node 33 (Top) Displacement for 0.60g Input



Option NON Advanced:

For RC Shearwall Structures With Interactive Shear & Bending Effects in Walls (Demo 18)

(NONLINEAR Module Plus Other Four Modules)

Main Steps of Option NON Advanced Nonlinear SSI Analysis Based on Standard Practices in US and Japan

Here are the main steps of the procedure:

1. Prepare *structure FE model.*
2. Create the *nonlinear RC wall FE submodels* from structure FE model
3. Perform *initial SSI analysis* for the gravity and seismic loads
4. Perform *automatic wall cross-section geometry identification and automatic section cuts* for each wall at each floor level for the gravity and seismic loads.
5. Compute *shear and bending BBCs* for each wall *per US or Japan standard recommendations*
6. Select *hysteretic wall models per US or Japan standard recommendations*
7. *Perform iterative nonlinear SSI analysis* using shear and bending hysteretic wall models and *combine shear and bending responses* at each iteration.
8. *Post-process the final SSI results* for the converged nonlinear response

Option NON Advanced Additional Modules

- ***The Section_Cuts_for_BBC module*** – Performs automatic wall section geometry identification and computes the wall section-cut forces for user-defined panels
- ***The BBC_JEAC_ACI_Fiber2D module*** – Computes shear and bending back-bone curves (BBC) for all the user-defined panels based on either the US standards or Japan standard recommendations, or 2D Fiber Model
- ***The Create_Flange_Materials module*** – Creates wall flange nonlinear materials for each wall panel which are used to create a new structure model .pre input file named *ModelName_NEW.pre* file.
- ***COMB_Shear_Bend module*** – This combines the nonlinear shear and bending interactive effects in wall panels after each SSI iteration.

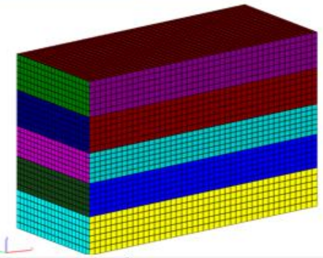
Option NON Advanced Implementation Flowchart

Build SSI Model

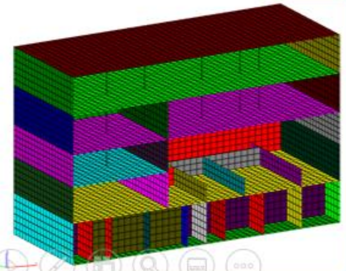
Analyst creates a 3DFEM for with element groups for each wall.

AB_Shearwall.pre Input File

Step 1 **(UI)**

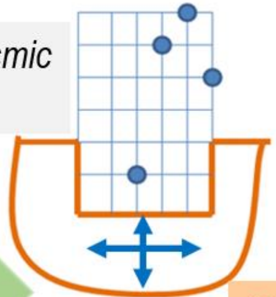


Analyst uses UI Section-Cut commands to split 3DFEM model into wall submodels

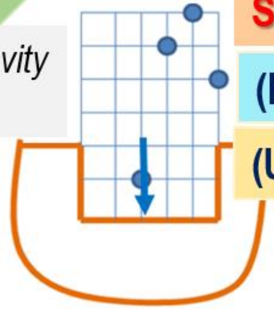


ACS SASSI SSI Analysis

Seismic SSI



Gravity SSI



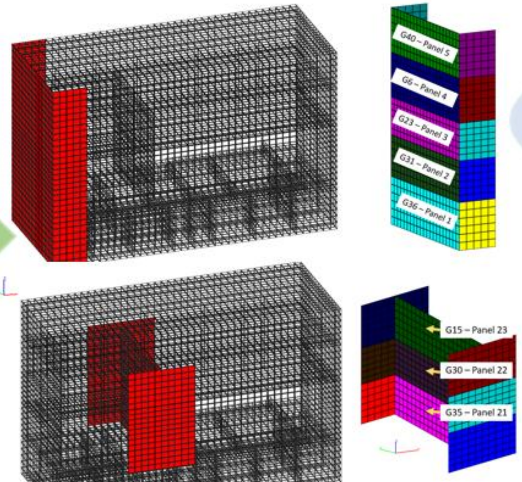
Step 3 **(B)** **(UI)**

STRESS Module Runs

STRESS Binary DB *for combined X, Y and Z Seismic inputs*

Step 4 **(B)** **(UI)**

STRESS Binary DB *for Gravity load in Z direction*



AB_ShearWall_Wall1.pre

⋮

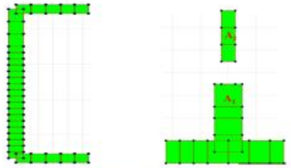
AB_ShearWall_Wall5.pre

Step 2 **(UI)**

Compute Shear and Bending BBCs

AB_Submodels_for_BBC.in file

Section_Cut_for_BBC Module Run



Automati... and secti... ident...

Options for Flanges:
1) Japan AIJ RC
2) US ACI-318

Step 5 ***Section_Data_for_BBC.out*** file **(B)**

User Section Review Adding RC Material Inputs

Step 6 ***Revised_Section_Data_for_BBC.in*** file **(EIF)**

Step 7 **BBC_JEAC_ACI_Fiber2D Run** **(B)**

BBC Shear.pre & BBC Bending....

Options for BBCs:
1) Japan JEAC 4601
2) US ACI-318ASCE 4

Step 8 **Create_Flange_Materials**

AB_ShearWall_New.pre fil...

Step 9 **UI is Used to AFWRITE New Inputs** **(UI)**

Generate New .Hou and .Eql Input Files

Option NON Nonlinear SSI Analysis (Batch Run)

Step 10 *Shear and bending effects are combined at each SSI iteration using* ***COMB_Shear_Bend*** *Module*

(B) ***File8*** files for converged SSI solution

Step 11 **Final SSI Post-Processing** **(B)** **(UI)**

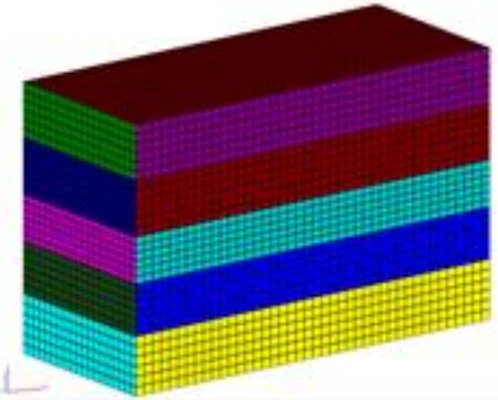
Steps 1-2: Prepare the 3DFEM with Separate Shell Groups for Walls

Build SSI Model

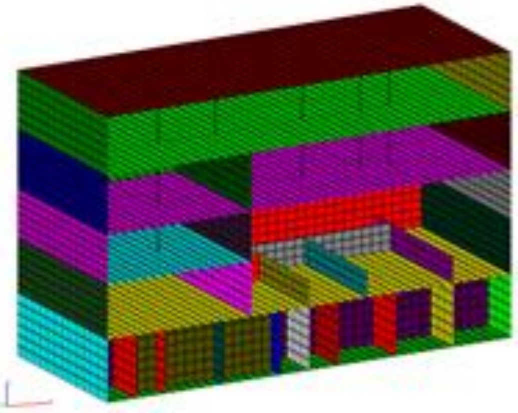
Analyst creates a 3DFEM for with element groups for each wall.

AB_Model.pre
Input File

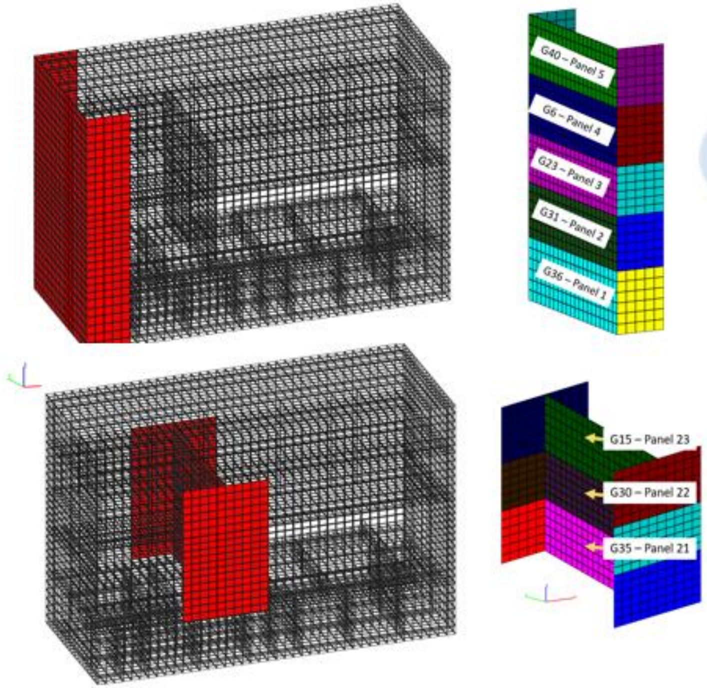
Step 1 (UI)



Analyst uses UI Section-Cut commands to split 3DFEM model into wall submodels



Use UI Section-cut commands to split the 3DFEM model in Wall submodels (Shell Groups). See Demos 18 and 19



AB_Model_Wall1.pre

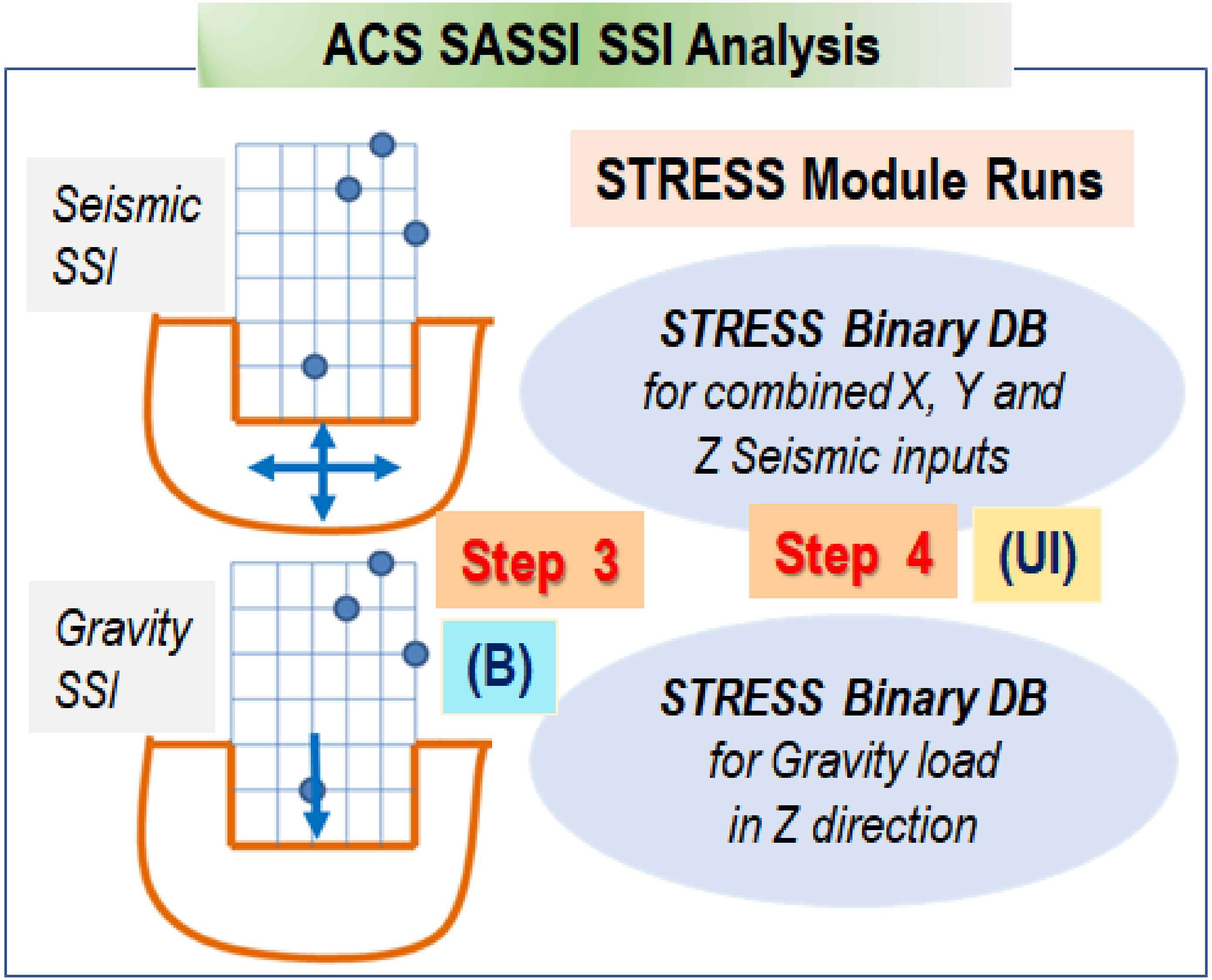
⋮

AB_Model_Wall5.pre

Step 2 (UI)

The 3DFEM and Wall submodel .pre file are used next to perform automatic section-cuts, section geometry identification for each wall submodel.

Steps 3-4: Perform SSI Analysis for Gravity and Seismic Loads



Step 3:

Perform SSI analysis (Batch)

1) Perform seismic ACS SASSI SSI analysis for the 3DFEM model using "Simultaneous Cases" ANALYS option to get FILE8s for post-processing

Step 4:

STRESS post-processing runs (Batch):

2) Run STRESS for the seismic inputs in X, Y and Z directions and create three binary DB for each input direction.

3) Run STRESS for the gravity (static) load for Z direction and create gravity binary DB

Combine X,Y,Z STRESS binary BD (B or UI):

4) Use COMBTHSDB to combine the seismic binary DBs for X, Y and Z in a single binary DB.

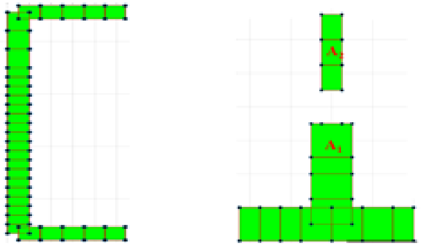
The Gravity and Seismic binary DBs are used in Step 5 for automatic section-cut calculations.

Step 5: Automatic Section Geometry Identification and Section-Cuts at Each Floor Level

Compute Shear and Bending BBCs

AB_Submodels_for_BBC.in file

Section_Cut_for_BBC Module Run



Automatic section-cut and section geometry identification

Step 5

Section_Data_for_BBC.out file

(B)

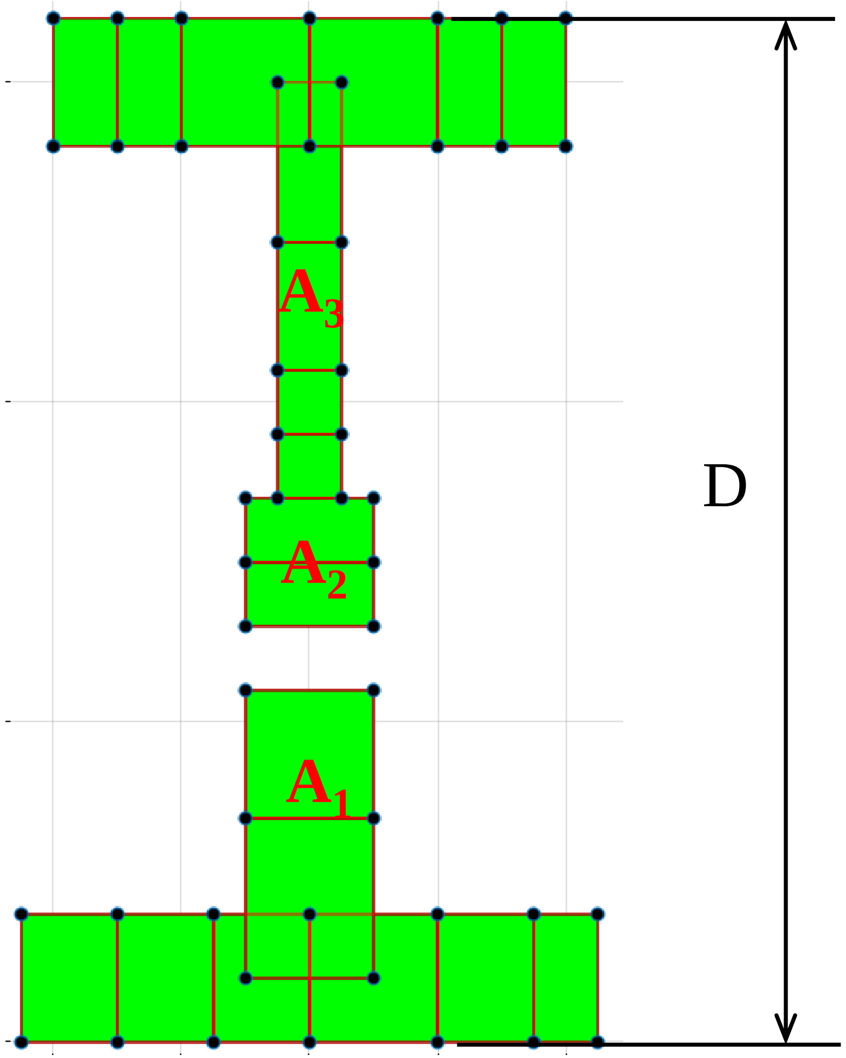
Step 5:

***Section_Cut_for_BBC* Module_runs (Batch):**
This module performs automatic section-cuts and identify the section geometries for all floor levels.

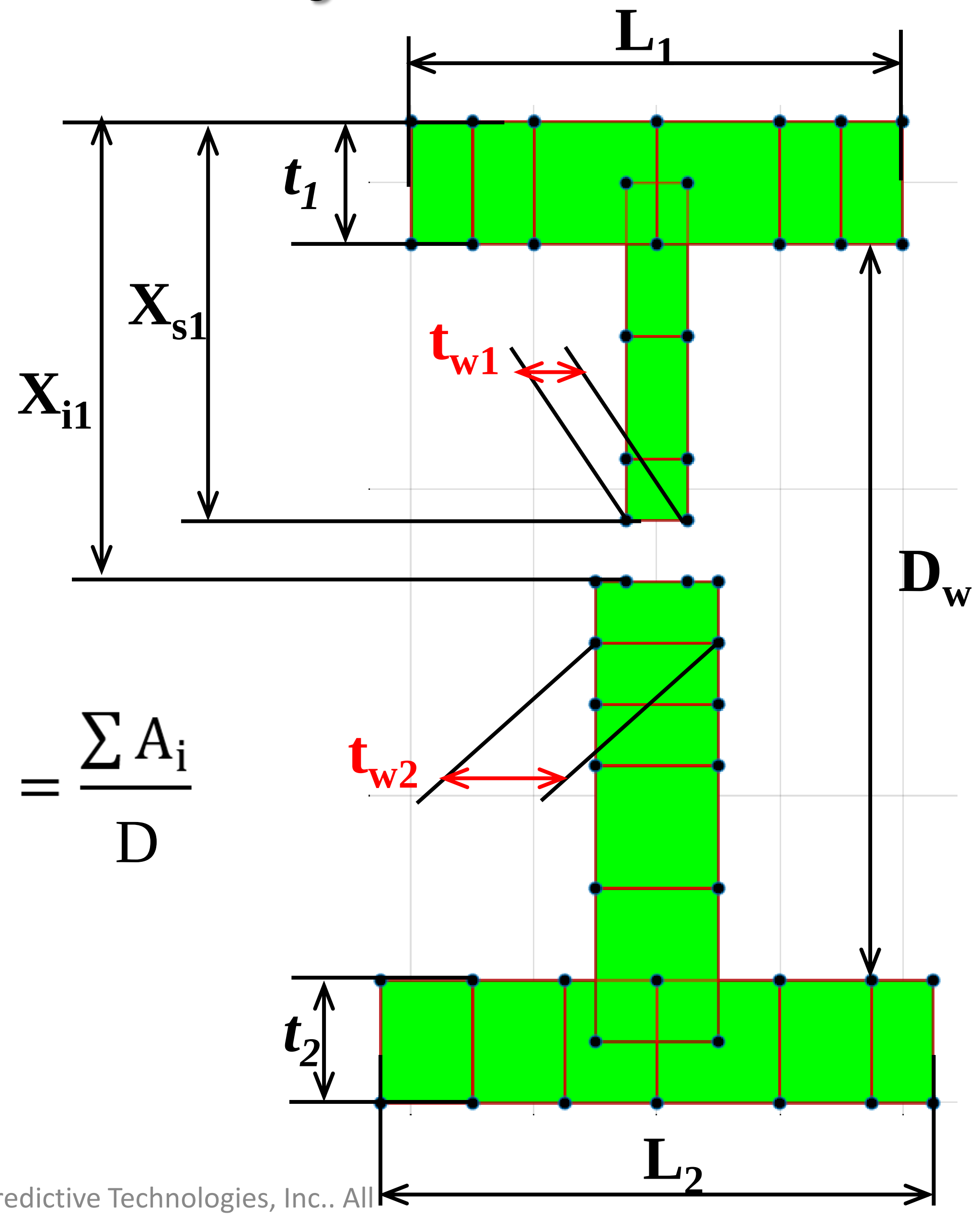
Output files:
The *Section_Data_for_BBC.out* output file produced by the run includes section-cut forces and geometry to be reviewed by the user in Step 6.

The *Modelname_Section_Data.out* as the general output file with input data and section geometry results.
The *Modelname_Section_Data.txt , output* file with the section data and other input data for next step

Examples of Section Geometry Identification

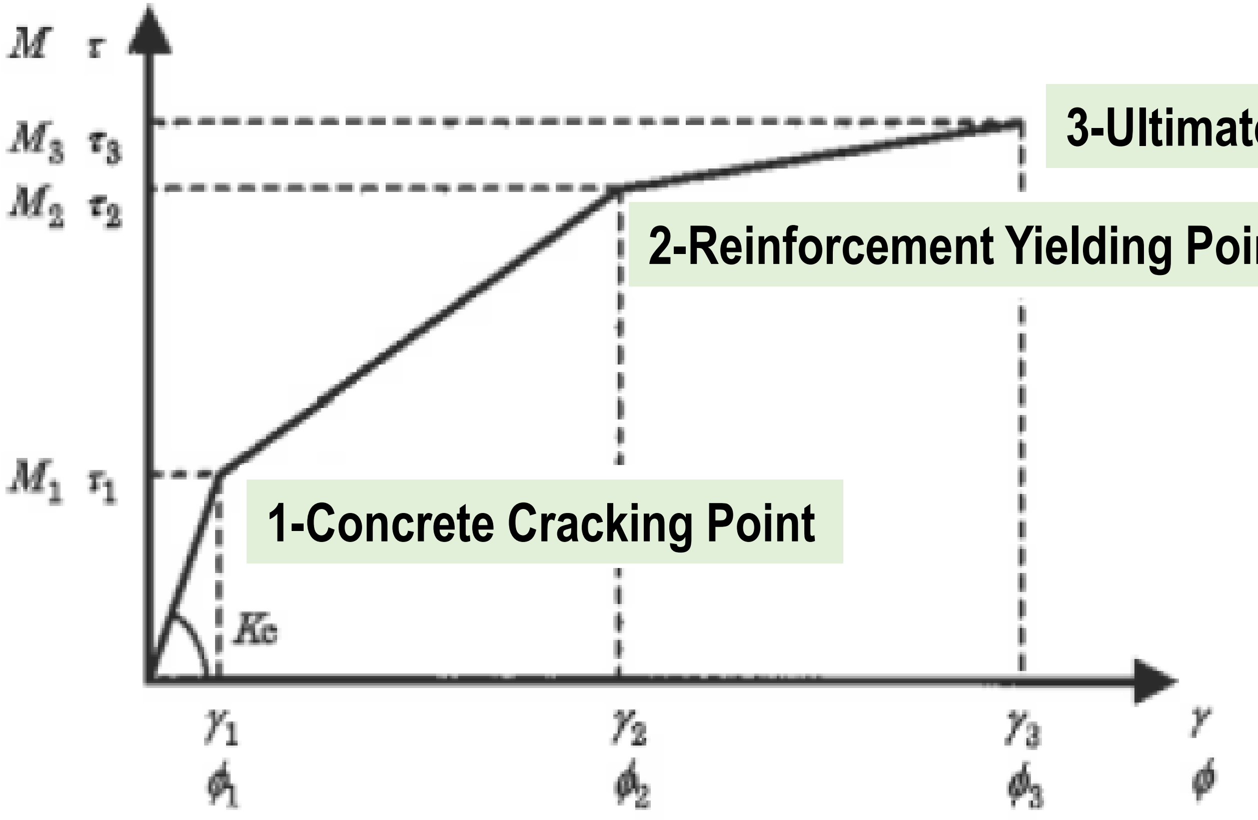


$$t_{eq} = \frac{\sum A_i}{D}$$



Nonlinear Modeling Wall Behavior Based on US & Japan Practice. Back-Bone Curves (BBC) and Hysteretic Models

BBC Curves: Are trilinear BBCs for both the shear and bending deformation following typical engineering practice, also recommended by the JEAC 4601-2015 Sect.3.5.6 (See figure below)



Exp. Figure 3.5.6-1 Trilinear skeleton curve.

Shear BBCs Computed per JEAC 4016-2015 Standard App.3.6

a. Shear deformation ($\tau\gamma$ relationship)

First turning point
$$\begin{cases} \tau_1 = \sqrt{0.31\sqrt{F_c}(0.31\sqrt{F_c} + \sigma_v} & (N/mm^2) \\ \gamma_1 = \tau_1 / G \end{cases}$$

Second turning point
$$\begin{cases} \tau_2 = 1.35\tau_1 & (N/mm^2) \\ \gamma_2 = 3\gamma_1 \end{cases}$$

Shear BBC at each floor level depends on the axial compression stress from gravity and the seismic bending moment by M/Q ratio (shear span ratio)

Ultimate point
$$\begin{cases} \tau_3 = \left(1 - \dfrac{\tau_s}{1.4\sqrt{F_C}}\right)\tau_0 + \tau_s & (N/mm^2) \quad \tau_s \le 1.4\sqrt{F_c} \\ \quad = 1.4\sqrt{F_c} & (N/mm^2) \quad \tau_s > 1.4\sqrt{F_t} \\ \gamma_3 = 4.0 \times 10^{-3} \end{cases}$$

where $\tau_0 = \left(0.94 - \dfrac{0.56M}{QD}\right)\sqrt{F_t}$ when $\dfrac{M}{QD} > 1$, $\dfrac{M}{QD} = 1$

$$\tau_s = \frac{(P_V + P_H)_s \sigma_y}{2} + \frac{(\sigma_v + \sigma_H)}{2}$$

Shear BBCs Computed Based on ASCE 4 & ACI 318 Standards

ACI 318-14 Section 18 for Shear Strength

$$V_n = A_{cv}(\alpha_c \lambda \sqrt{f_c'} + \rho_t f_y)$$

where the coefficient α_c is 3.0 for $h_w/\ell_w \le 1.5$, is 2.0 for $h_w/\ell_w \ge 2.0$, and varies linearly between 3.0 and 2.0 for h_w/ℓ_w between 1.5 and 2.0.

any one of the individual wall piers, V_n shall not be taken larger than $10A_{cw}\sqrt{f_c'}$, where A_{cw} is the area of concrete section of the individual pier considered.

Option NON BBC_GENERATION Module Implementation

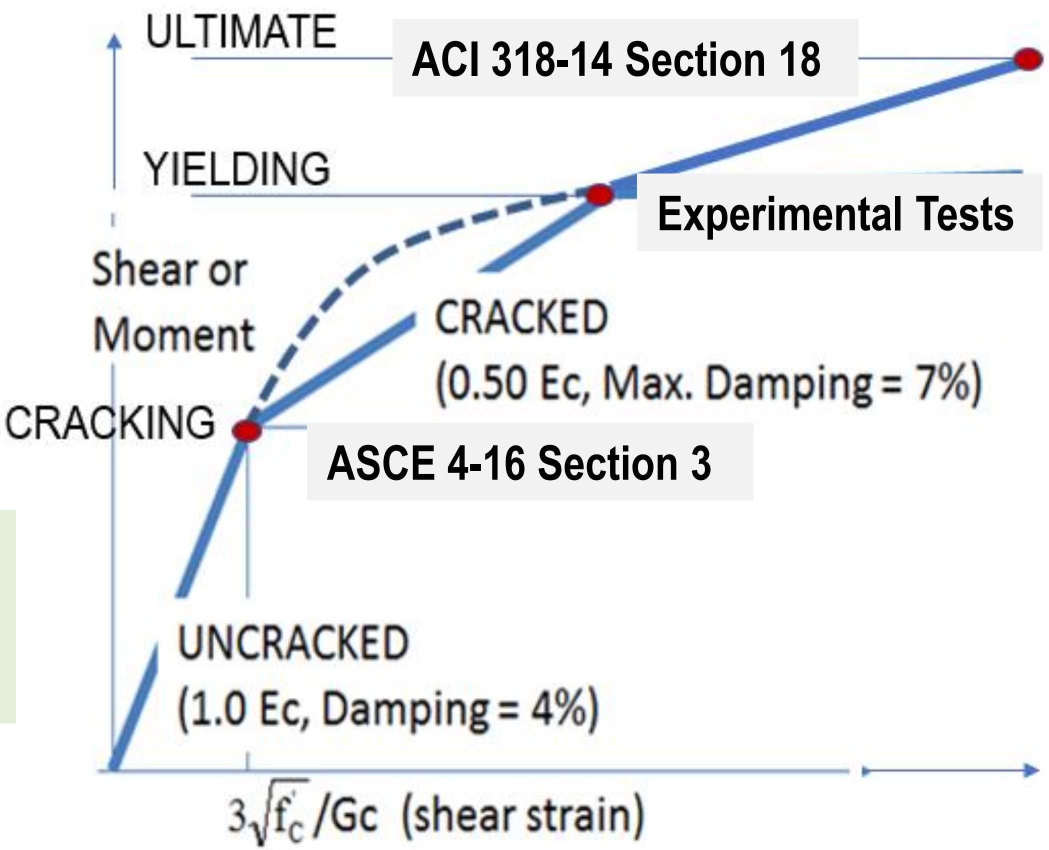
$$V = \left(\alpha_c \sqrt{f'_c} + \rho_H f_y\right) A_W \le 10\sqrt{f'_c}\, A_W$$

Does no depend on axial force or bending effects!

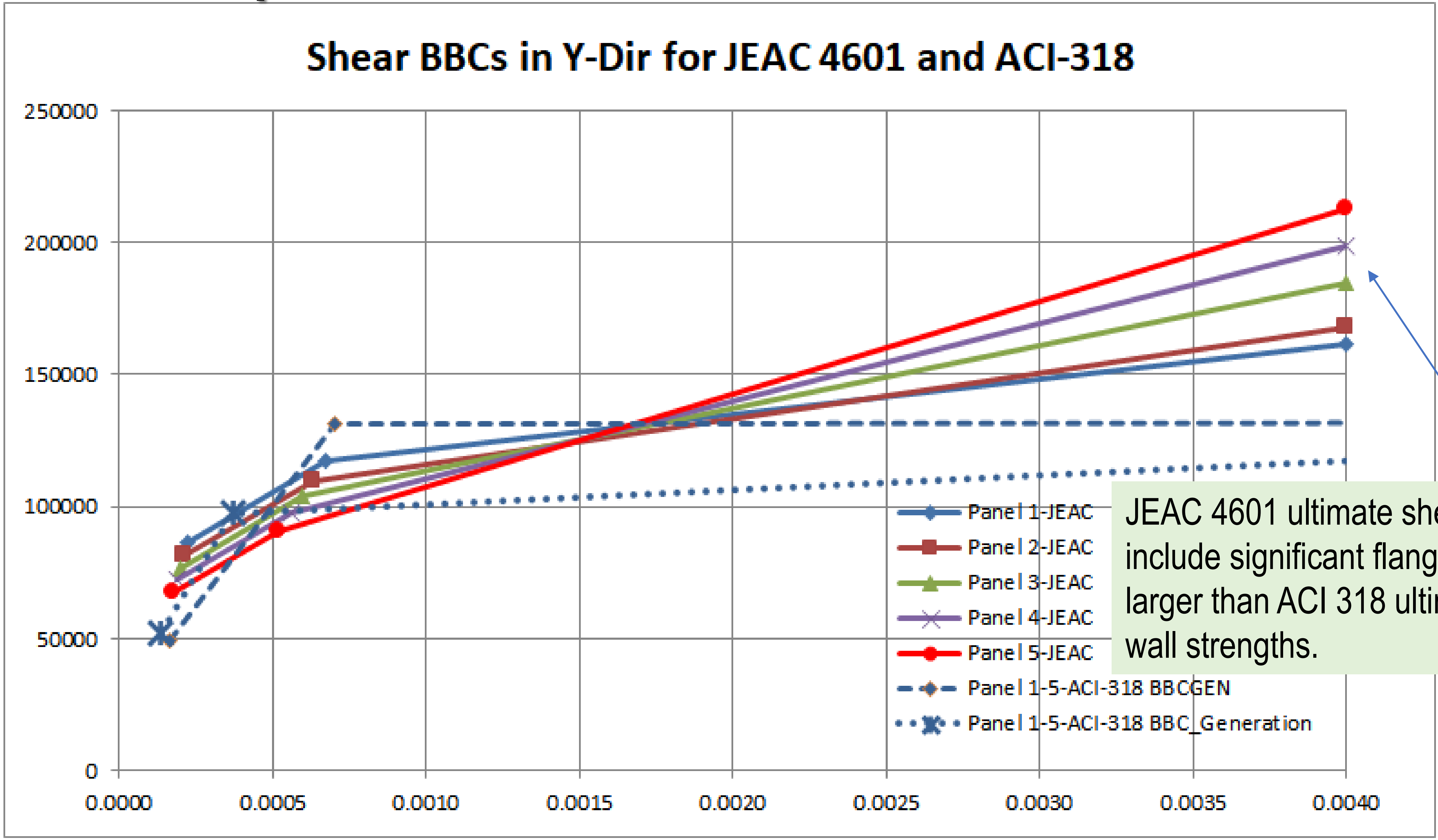
ASCE 4-16 Section 3

RC wall shear cracking occurs when the shear stress is larger than $3.\sqrt{f'_c}$

Trilinear Shear BBC Curve



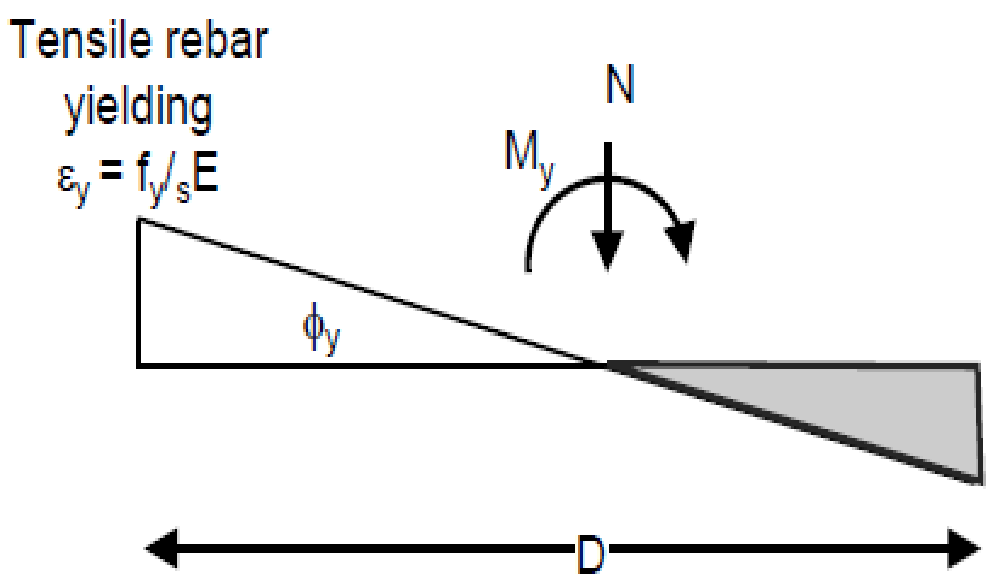
Computed Shear BBCs for TB RC Walls in Y-Dir



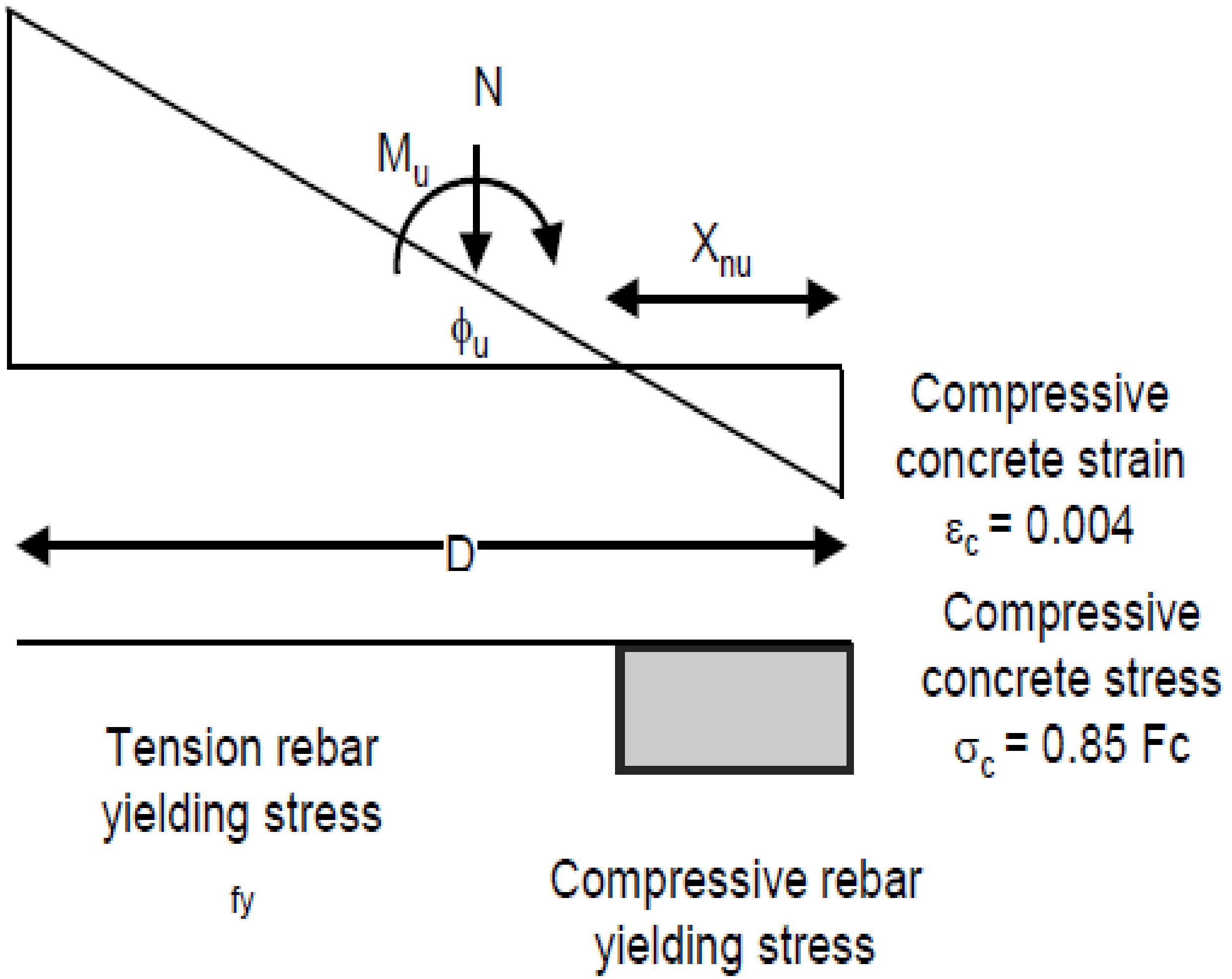
JEAC 4601 ultimate shear forces include significant flange effects; larger than ACI 318 ultimate shear wall strengths.

Bending BBCs Computed Based on US and Japan Standards

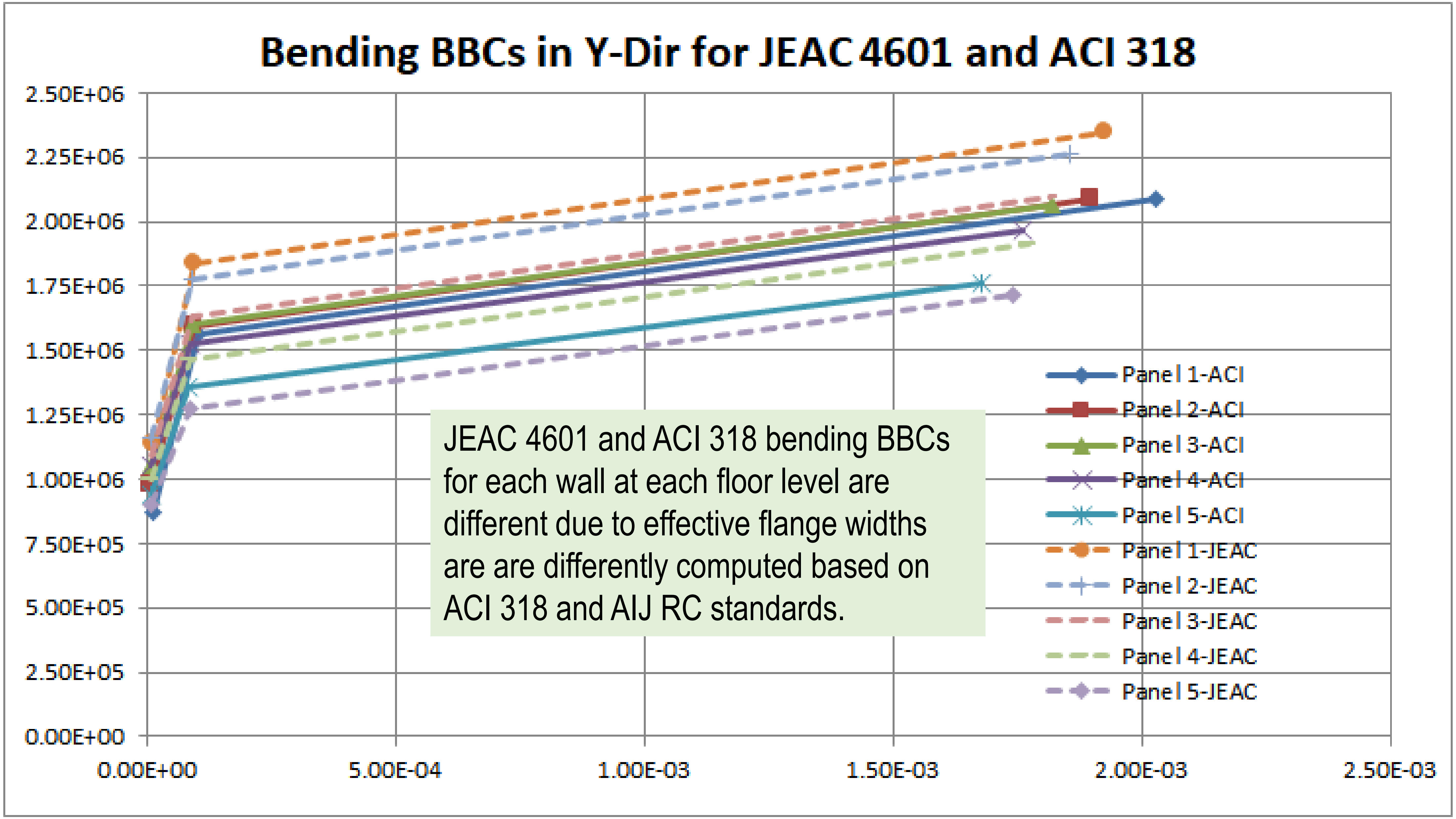
2-Reinforcement Yielding Point



3-Ultimate Point

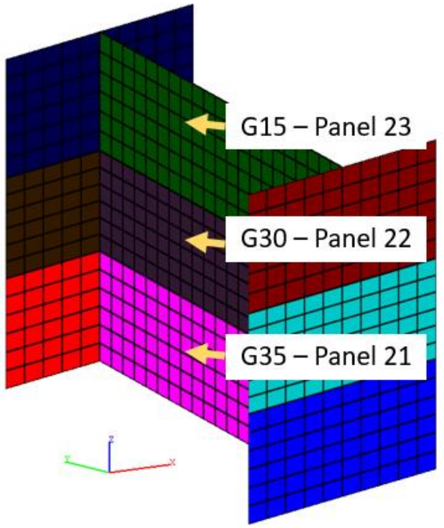


Computed Bending BBCs for TB RC Walls in Y-Dir



Section_Data_for_BBC.out File from *Section_Cuts_for_BBC Module* (Step5)

Example for Wall 5 Submodel with 3 Floors (and Sections)




```
3
-8.0264    16.764    40.843
1
  0.264106E+05   0.409823E+06   0.404182E+05
  4.7488     14.441     1.5240     14.441     1.5240     24.079     1.5240     13.801     13.801
  0
  0.0000     0.0000     0.0000     0.0000
  0.248546E+08   0.106216E+08   0.00E+00   0.00E+00   0.000E+00   0.000E+00   0.00E+00  0.00E+00
2
  0.228232E+05   0.188437E+06   0.358970E+05
 11.924     14.441     1.5240     14.441     1.5240     24.079     1.5240     17.650     17.650
  0
  0.0000     0.0000     0.0000     0.0000
  0.248546E+08   0.106216E+08   0.00E+00   0.00E+00   0.000E+00   0.000E+00   0.00E+00  0.00E+00
3
  0.124042E+05   0.371685E+05   0.215392E+05
 19.391     14.441     1.5240     14.441     1.5240     24.079     1.5240     20.946     20.946
  0
  0.0000     0.0000     0.0000     0.0000
  0.248546E+08   0.106216E+08   0.00E+00   0.00E+00   0.000E+00   0.000E+00   0.00E+00  0.00E+00
```

Automatic Section Cut results for N, M and V

Automatic Section Data

Concrete and steel material parameters, reinforcement ratios. To be completed by analyst after reviewing section geometries.

Step 6: Analyst Review of *Section_Data* Files To Prepare Nonlinear Input

User Section Review Adding RC Material Inputs

Step 6 *Revised_Section_Data_for_BBC.in* file

Step 6:
Analyst shall edit the *Section_Data_for_BBC.out* file for checking the automatic generated section-cut geometries (web and effective flanges sizes including floor openings effects). The analyst can modify section parameters based on engineering judgements and need to input concrete and steel nonlinear material parameters. Analyst should save the revised file as *Revised_Section_Data_for_BBC.in* file. This file is used as an input of Step 7.

Section data are provided in international units (kN and m)

Section_Data_for_BBC.out (Step 5)

```
 5
-8.0264     0.0000     16.764
 1
  0.949395E+05  0.336417E+07  0.153516E+06
-3.3650     7.9827     1.5240     7.9827     1.5240     24.079     1.5240     5.0674     5.0674
 0
 0.0000     0.0000     0.0000     0.0000
  0.248546E+08  0.106216E+08  0.0000E+00  0.0000E+00   0.000000E+00   0.000000E+00  0.0000E+00
 2
  0.964498E+05  0.223449E+07  0.193119E+06
 4.7488     7.9827     1.5240     7.9827     1.5240     24.079     1.5240     7.3088     7.3088
 0
 0.0000     0.0000     0.0000     0.0000
  0.248546E+08  0.106216E+08  0.0000E+00  0.0000E+00   0.000000E+00   0.000000E+00  0.0000E+00
```

Revised_Section_Data_for_BBC. in (Step 6)

```
 5
-8.0264     0.0000     16.764
 1
  0.949395E+05  0.336417E+07  0.153516E+06
-3.3650     7.9827     1.5240     7.9827     1.5240     24.079     1.5240     5.0674     5.0674
 0
 1.1950     1.5980     1.2460     0.95300
  0.248546E+08  0.330000E+05  0.2000E-02  0.4000E-02   0.205000E+09   0.345000E+06  0.1850E-02
 2
  0.964498E+05  0.223449E+07  0.193119E+06
 4.7488     7.9827     1.5240     7.9827     1.5240     24.079     1.5240     7.3088     7.3088
 0
 1.1950     1.5980     1.2460     0.95300
  0.248546E+08  0.330000E+05  0.2000E-02  0.4000E-02   0.205000E+09   0.345000E+06  0.1850E-02
```

Revised_Section_Data_for_BBC.in File Data Description

Line 6: NOPN, X0S, X0I, Z0S, Z0I, X1S, X1I, Z1S, Z1I (Openings explicitly defined) where
NOPEN = Total number of openings of the wall
X0S, X1S, … = Superior X coordinates at the top of each wall panel opening
X0I, X1I, … = Inferior X coordinates at the bottom of each wall panel opening
Z0S, Z1S, … = Superior Z coordinates at the top of each wall panel opening
Z0I, Z1I , … = Inferior Z coordinates at the bottom of each wall panel opening

Line 7: PVf1, PVf2, PVw, PHw (Wall Reinforcement Percentage)
PVf1 = Reinforcement percentage for Flange 1 (top)
PVf2 = Reinforcement percentage for Flange 2 (bottom)
PVw = Reinforcement percentage for Web (vertical)
PHw = Reinforcement percentage for Web (horizontal)

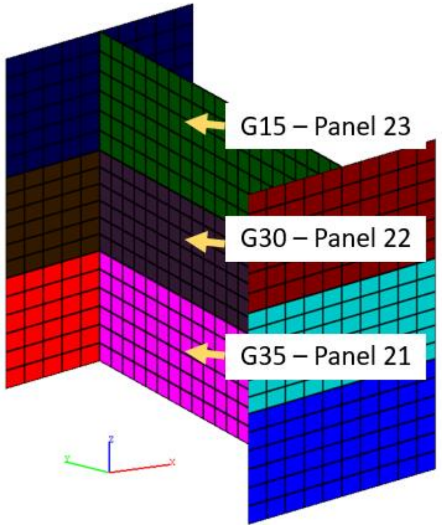
Line 8: Ec, Fc, Epsc_y, Epsc_u, Es, Fs, Epss_y, Epss_u
Ec = Concrete E modulus
Fc = Concrete Fc strength
Epsc_y = Concrete Yielding strain
Epsc_u = Concrete Ultimate strain
Es – Steel E modulus
Fs – Steel Fy yielding
Epss_y – Steel Yielding strain
Epss_u – Steel Ultimate strain

These are parameters shall be input by analyst for each wall Submodel and each floor level

Repeat line 3 to line 8 for all the sections of the wall.

Revised_Section_Data_for_BBC.in Input for *BBC_JEAC_4601_2015 Module*

Example for Wall 5 with 3 Floors (and Sections)




```
 3
-8.0264    16.764    40.843
 1
  0.264106E+05   0.409823E+06   0.404182E+05
4.7488     14.441     1.5240     14.441     1.5240     24.079     1.5240     13.801     13.801
 0
1.1950     1.5980     1.2460     0.95300
0.248546E+08   0.3300E+05  0.200E-02  0.400E-02   0.205000E+09   0.3450E+06  0.185E-02  0.500E-01
 2
  0.228232E+05   0.188437E+06   0.358970E+05
11.924     14.441     1.5240     14.441     1.5240     24.079     1.5240     17.650     17.650
 0
1.1950     1.5980     1.2460     0.95300
0.248546E+08   0.3300E+05  0.200E-02  0.400E-02   0.205000E+09   0.3450E+06  0.185E-02  0.500E-01
  3
  0.124042E+05   0.371685E+05   0.215392E+05
19.391     14.441     1.5240     14.441     1.5240     24.079     1.5240     20.946     20.946
 0
1.1950     1.5980     1.2460     0.95300
0.248546E+08   0.3300E+05  0.200E-02  0.400E-02   0.205000E+09   0.3450E+06  0.185E-02  0.500E-01
```

Section data are provided only in International system (kN and m)

If D=0, use 2D Fiber Model

BBC_JEAC_ACI_Fiber2D Module Run and Files (Step 7)

Step 7:

Batch *BBC_JEAC_4601_2015* Module V4.3.1 or *BBC_JEAC_ACI_Fiber2D* Module V4.3.2 run – for Directional Walls:

This module computes the shear and bending BBC for each Wall submodel based on JEAC 4601-2015 App.3.7 approaches. For the shear BBC, the ultimate state shear stress is computed for both exterior walls (App.3.7 equations) and internal walls (Ref.App.3.7-14).

Output files for each RC Wall Submodel:

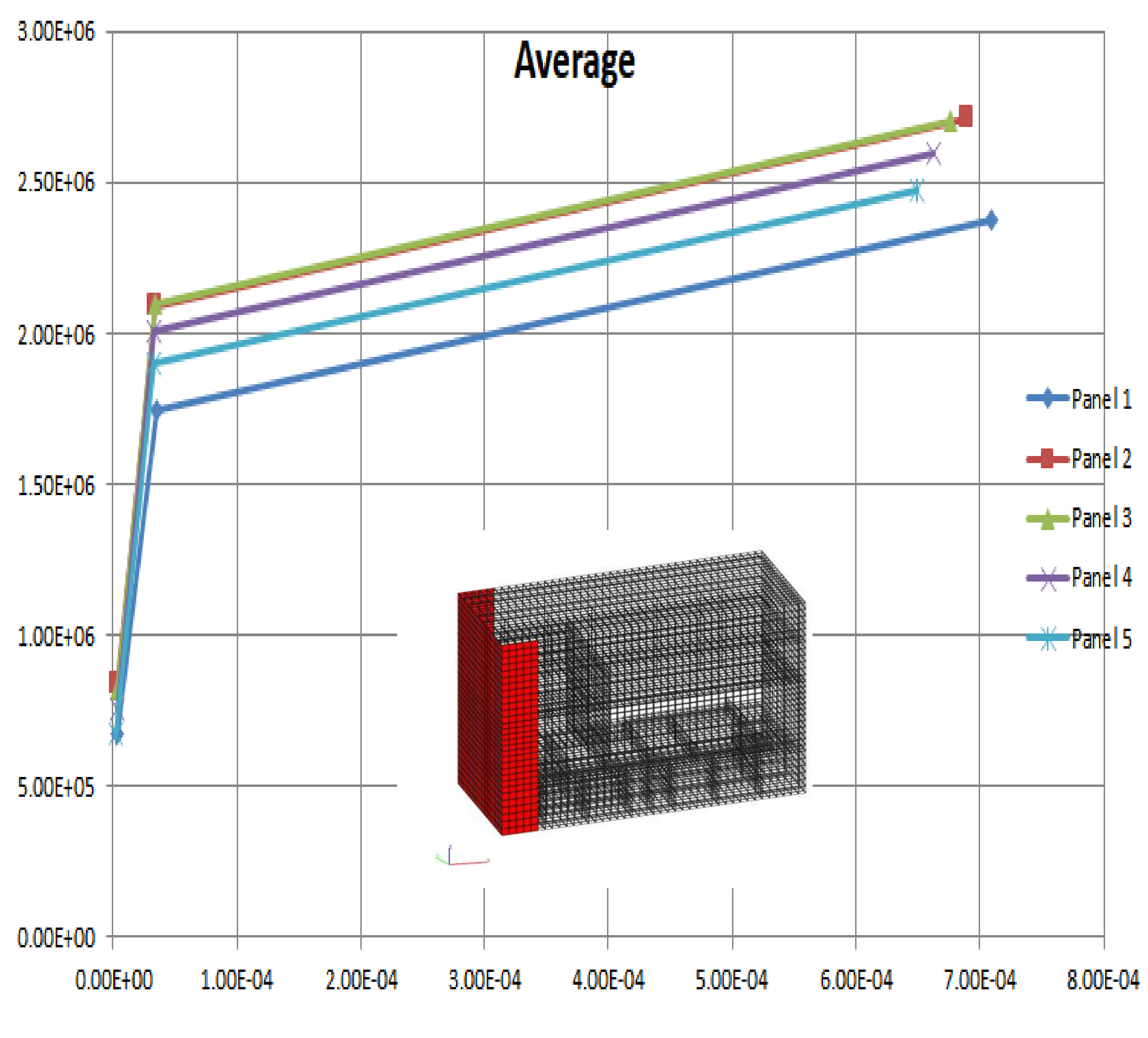
The computed shear BBC are saved in two text files, namely *Modelname_BBC_.out and BBC_Shear.pre.* The units in the output file are N and mm for International units or Kip and ft for British units and it depends on how the 3DFEM model is defined in the .pre input file. The output file contains the computed shear stress in N/mm2, while in the BBC_Shear.pre file the shear force in given kN or Kip.

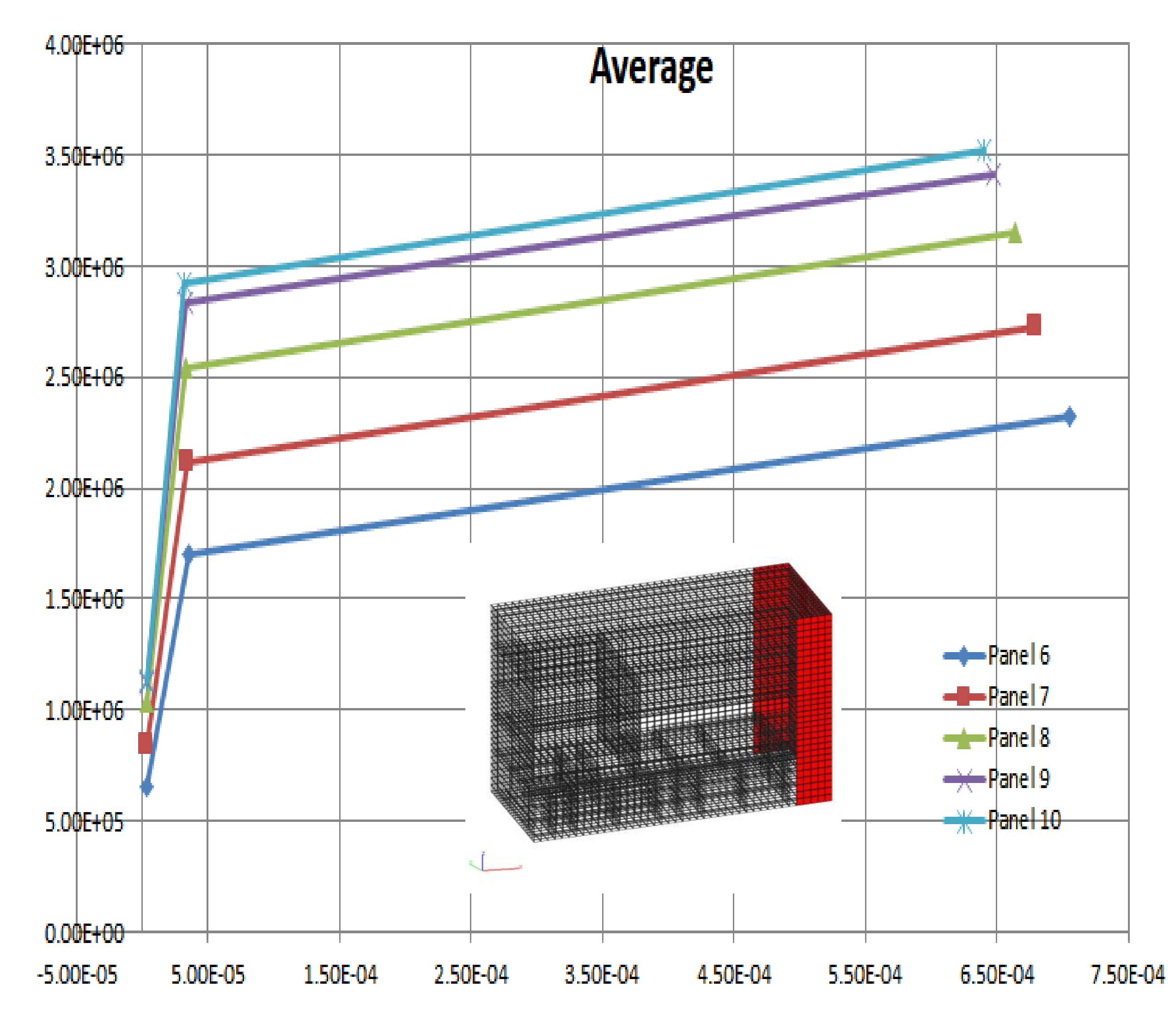
The computed bending BBC are saved in three files, the *BBC_JEAC_4601_Data.out* file *and two BBC_Bending.pre* files, one .pre file for minimum moments and one .pre file for average moments. The minimum and average moments are computed based on two cases: 1) Flange 1 is in compression and 2) Flange 1 is in tension. The moment units in the output file are kN-m, while in the *BBC_Bending.pre files* is given in kN-m or Kip-ft. Analyst has to decide if uses minimum or average moments.

***BBC_JEAC_ACI_Fiber2D* Module V4.3.2 run – for Non-directional Walls (closed sections, circular, square, composite):**

Computes the shear and bending BBC for each Wall submodel based on 2D Fiber model and shear area numerical integration for non-planar walls (without flanges). The 2D Fiber Model is launched when Dw=0, i.e. the flange and web identification fails.

Bending BBCs Computed for External and Internal Walls Using BBC_JEAC_ACI_Fiber2D Module (per JEAC 4601)





Modelname_Wall#.txt File With Section Data Based on JEAC 4601 (Step 7)

Section Geometries and Material Properties

Geometric Input Data:

Z Coordinate of Wall Bottom: -8.026

				Web			North Flange			South Flange		
Section#	Z-Location	Ic	Ie	Dw	Tw	Pvw	L1	T1	Pvf1	L2	T2	Pvf2
		(m4)	(m4)	(mm)	(mm)	(%)	(mm)	(mm)	(%)	(mm)	(mm)	(%)
1	-3.3650	3149.	3444.	24079.	1524.	1.246	5067.	1524.	1.195	5067.	1524.	1.598
2	4.7488	4019.	4400.	24079.	1524.	1.246	7309.	1524.	1.195	7309.	1524.	1.598
3	11.9240	4318.	4728.	24079.	1524.	1.246	8078.	1524.	1.195	8078.	1524.	1.598
4	19.3910	4502.	4931.	24079.	1524.	1.246	8553.	1524.	1.195	8553.	1524.	1.598
5	27.4960	4610.	5050.	24079.	1524.	1.246	8832.	1524.	1.195	8832.	1524.	1.598

Material Properties:

Section#	AS	Fc	Ec	Gc	epsc_y	epsc_u	Fs	Es	Gs	epss_y	epss_u	sigm_v	Pv	Ph
	(m2)	(N/mm2)	(N/mm2)	(N/mm2)	(%)	(%)	(N/mm2)	(N/mm2)	(N/mm2)	(%)	(%)	(N/mm2)	(%)	(%)
1	32.05	33.00	24855.	10356.	0.200	0.400	379.50	205000.	85417.	0.185	5.000	1.821	1.246	0.953
2	32.05	33.00	24855.	10356.	0.200	0.400	379.50	205000.	85417.	0.185	5.000	1.635	1.246	0.953
3	32.05	33.00	24855.	10356.	0.200	0.400	379.50	205000.	85417.	0.185	5.000	1.184	1.246	0.953
4	32.05	33.00	24855.	10356.	0.200	0.400	379.50	205000.	85417.	0.185	5.000	0.574	1.246	0.953
5	32.05	33.00	24855.	10356.	0.200	0.400	379.50	205000.	85417.	0.185	5.000	0.021	1.246	0.953

Shear BBC Information

Shear Force Outputs:

Section#	M/QD	Tao1	Gamma1	Tao2	Gamma2	Tao3 (EX.)	Tao3 (IN.)	Gamma3
		(N/mm2)		(N/mm2)		(N/mm2)	(N/mm2)	
1	0.910	2.5325	0.2445E-03	3.4189	0.7336E-03	5.9927	4.0886	0.4000E-02
2	0.481	2.4665	0.2382E-03	3.3298	0.7145E-03	6.4529	4.1516	0.4000E-02
3	0.322	2.2978	0.2219E-03	3.1020	0.6656E-03	6.5434	4.1287	0.4000E-02
4	0.208	2.0479	0.1977E-03	2.7647	0.5932E-03	6.5666	4.0803	0.4000E-02
5	0.191	1.7914	0.1730E-03	2.4184	0.5189E-03	6.4796	4.0320	0.4000E-02

Section data are provided only in International system (N and mm) per JEAC 4601 App. 3.7 equations

Modelname_Wall#.txt File With Section Data Based on JEAC 4601 (Step 7)

Bending BBC Information

Bending Moment Outputs:

M1 Calculation)

Section#	Load Direction	Cx (mm)	Ze (m3)	Mc (KN*M) Value	Mc (KN*M) Average	Mc (KN*M) Minimum	Phi (1/m) Value	Phi (1/m) Average	Phi (1/m) Minimum
1	1	12040.	286.	0.1145E+07			0.1464E-04		
1	2	12040.	286.	0.1145E+07	0.1145E+07	0.1145E+07	0.1464E-04	0.1464E-04	0.1464E-04
2	1	12040.	365.	0.1395E+07			0.1397E-04		
2	2	12040.	365.	0.1395E+07	0.1395E+07	0.1395E+07	0.1397E-04	0.1397E-04	0.1397E-04
3	1	12039.	393.	0.1322E+07			0.1232E-04		
3	2	12039.	393.	0.1322E+07	0.1322E+07	0.1322E+07	0.1232E-04	0.1232E-04	0.1232E-04
4	1	12040.	410.	0.1129E+07			0.1009E-04		
4	2	12040.	410.	0.1129E+07	0.1129E+07	0.1129E+07	0.1009E-04	0.1009E-04	0.1009E-04
5	1	12040.	419.	0.9245E+06			0.8068E-05		
5	2	12040.	419.	0.9245E+06	0.9245E+06	0.9245E+06	0.8068E-05	0.8068E-05	0.8068E-05

M2 Calculation

Mc (KN*M) Value	Mc (KN*M) Average	Mc (KN*M) Minimum	Phi (1/m) Value	Phi (1/m) Average	Phi (1/m) Minimum
0.2873E+07			0.1119E-03		
0.2628E+07	0.2751E+07	0.2628E+07	0.1102E-03	0.1110E-03	0.1102E-03
0.3424E+07			0.1083E-03		
0.3060E+07	0.3242E+07	0.3060E+07	0.1064E-03	0.1073E-03	0.1064E-03
0.3365E+07			0.1052E-03		
0.2959E+07	0.3162E+07	0.2959E+07	0.1033E-03	0.1042E-03	0.1033E-03
0.3100E+07			0.1017E-03		
0.2671E+07	0.2885E+07	0.2671E+07	0.9971E-04	0.1007E-03	0.9971E-04
0.2811E+07			0.9840E-04		
0.2360E+07	0.2586E+07	0.2360E+07	0.9665E-04	0.9752E-04	0.9665E-04

M3 Calculation

Mc (KN*M) Value	Mc (KN*M) Average	Mc (KN*M) Minimum	Phi (1/m) Value	Phi (1/m) Average	Phi (1/m) Minimum
0.3791E+07			0.2006E-02		
0.3534E+07	0.3662E+07	0.3534E+07	0.2204E-02	0.2105E-02	0.2006E-02
0.4331E+07			0.2166E-02		
0.3943E+07	0.4137E+07	0.3943E+07	0.2127E-02	0.2147E-02	0.2127E-02
0.4239E+07			0.2104E-02		
0.3809E+07	0.4024E+07	0.3809E+07	0.2065E-02	0.2085E-02	0.2065E-02
0.3933E+07			0.2035E-02		
0.3479E+07	0.3706E+07	0.3479E+07	0.1994E-02	0.2014E-02	0.1994E-02
0.3601E+07			0.1968E-02		
0.3135E+07	0.3368E+07	0.3135E+07	0.1933E-02	0.1950E-02	0.1933E-02

Create_Flange_Materials Module Run (Step 8)

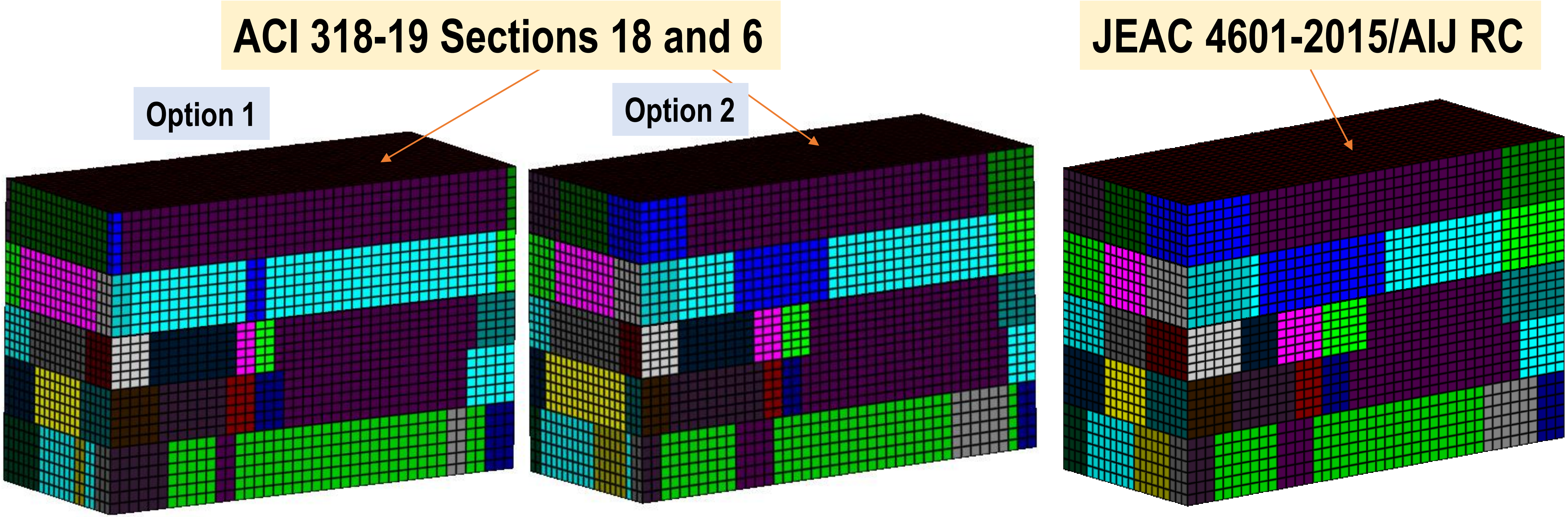
Step 8 — **Create_Flange_Materials Run** (B)

AB_ShearWall_New.pre file

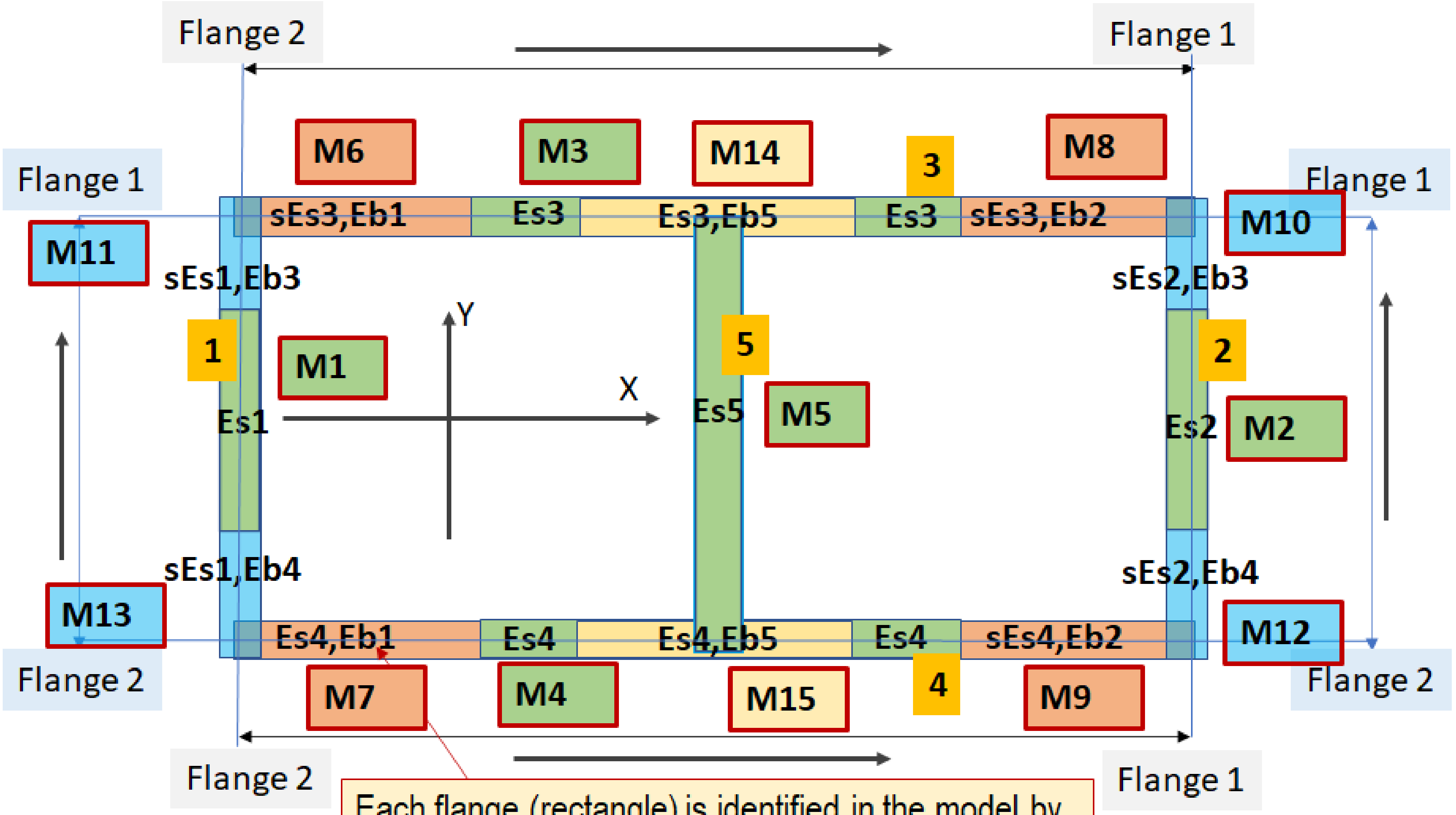
Step 8:

***Create_Flange_Material* Module** is run to create a new FE model including additional effective flange width materials.

Creates a new structure FEA model, *Modelname_New.pre*



Include New Flange Materials for Nonlinear Modeling – Case 3



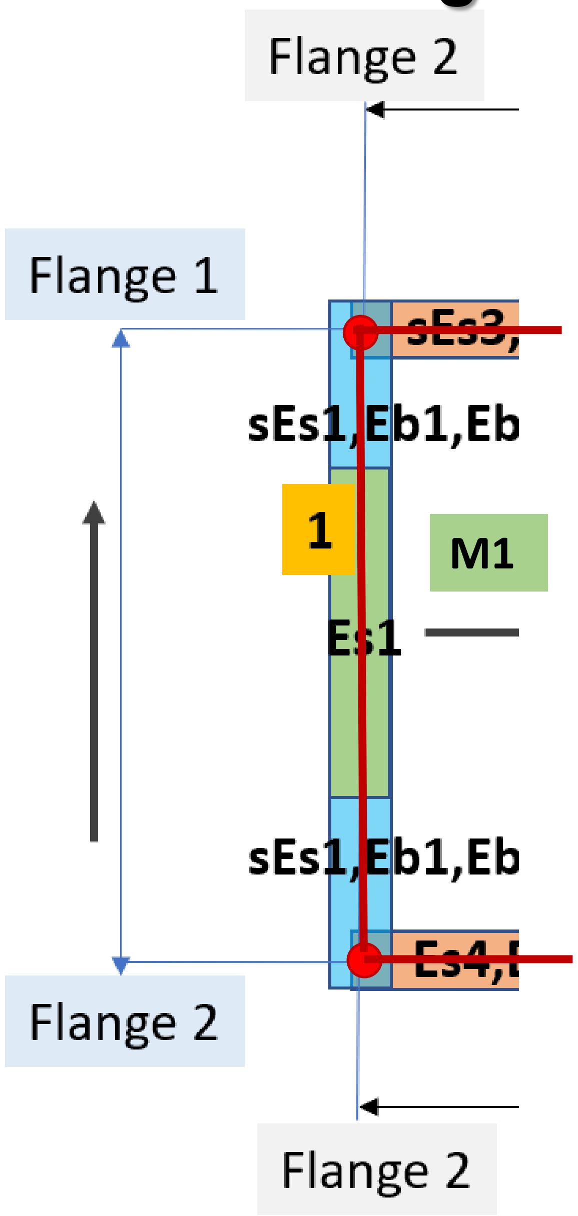
S =1 is for including shear effective Es for corner flanges (exterior walls)

S = 0 (default) is for not including Es for corner flanges.

For separate inputs, the wall perpendicular to input direction remain elastic (M3, M4 for Y input, and M1, M2 for X input)

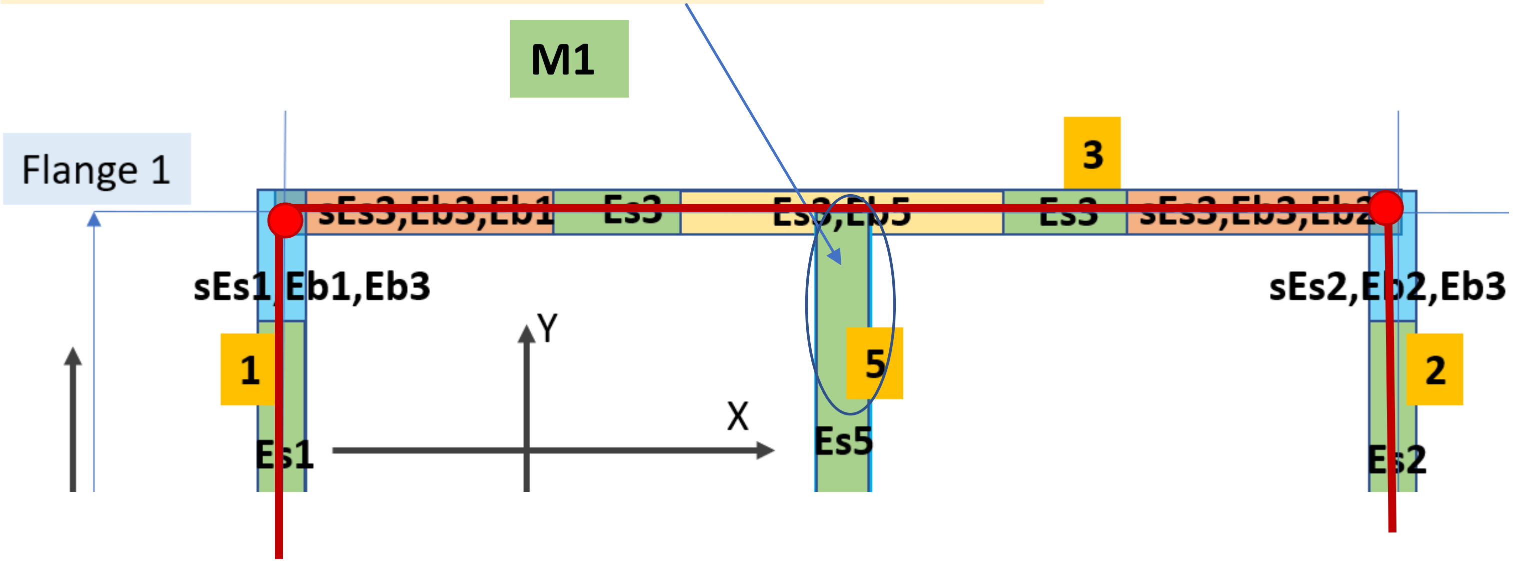
Each flange (rectangle) is identified in the model by the group # and the new assigned material # (to be used to change HOUSE input after each iteration)

E & D Material Changes For Flanges 1 & 2 Are at Two Wall Web Ends



Nodes of elements of Flanges 1 and 2 which are the two web ends should include the Panel 4 corner nodes!

This criterion was used to eliminate any other intermediate flanges in the middle of the web (not at the two ends) that should not be included for the E material shear-bending combination in COMB_SHEAR_BEND.



Effective Flange Size Calculations Implemented in Option NON

ACI 318 Option 1 in NON

The first ACI 318 option (Section 18.10.5.2) is recommended by the standard for modeling the RC walls, and the effective wall flange widths computation is based on the shear lag effects on the stress distribution in perpendicular walls at the intersection with the parallel walls. The shear lag effect is larger for larger nonlinear story drifts and axial loads.

ACI 318 Option 2 in NON

The second ACI 318 option (Section 6.3.2.1) is not recommended by the standard for modeling the RC walls. This option uses the effective wall flange width equations for the effective beam flange widths, basically, assuming that the beams represent the verticall RC walls. This assumption is conceptually consistent with the Japanese AIJ RC standard requirements for computing the effective wall flange widths.

JEAC 4601 Option in NON

Per the JEAC 4601-2015 standard implementation in practice based on the SR/Stick models and the Case 4 directional approach, the effective wall flange widths are determined using the effective beam flange widths computed per the AIJ RC standard equations. The effective flange widths conceptually reflect the variation with height of the effective bending stiffneses for RC wall parallel to the input direction.

Effects of Effective Flange Sizes on Nonlinear SSI Responses

The new Option NON implementation uses three calculation options per JEAC 4601 and ACI 318:

ACI 318-19 for Wall Effective Flange Widths Per Section 18.10.5.2 (ACI recommended)

ACI 318 Option 1

For the two-*sided* wall flanges, B1 and B2, then, the effective wall flanges are computed based on the side clearances, AR and AL, at each floor level I, as follows:

$$B1_i = min\begin{cases}\frac{1}{2}ARi\\ 0.25\ (H - Zsect, i)\end{cases}$$

$$B2_i = min\begin{cases}\frac{1}{2}ALi\\ 0.25\ (H - Zsect, i)\end{cases}$$

Where H-Zsect, i is the height of the structure above the i section level, Zsect,i

$$L1C_i = B1_i + Tw_i + B2_i$$

$$L2C_i = L1C_i$$

The above equations are also used for the one-sided wall flanges.

JEAC 4601-2015/AIJ RC Standard:

JEAC 4601/AIJ RC

$$B1_i = \begin{cases}\left(0.5 - 0.3 * \left(\frac{AR}{L_i}\right)\right) * AR, & if\ AR < L_i\\ 0.2 * L_i, & else\end{cases}$$

$$B2_i = \begin{cases}\left(0.5 - 0.3 * \left(\frac{AL}{L_i}\right)\right) * AL, & if\ AL < L_i\\ 0.2 * L_i, & else\end{cases}$$

$$L1C_i = B1_i + Tw_i + B2_i$$

$$L2C_i = L1C_i$$

ACI 318-19 for Wall Effective Flange Widths Per Section 6.3.2.1 (conceptually similar to AIJ RC modeling requirements, but not ACI recommended)

ACI 318 Option 2

a. If the panel i has *two side* flanges, B1i and B2i.

$$B1_i = min\begin{cases}\frac{1}{2}AR\\ 8\,Tw_i\end{cases}$$

$$B2_i = min\begin{cases}\frac{1}{2}AL\\ 8\,Tw_i\end{cases}$$

Check also $B1_i + B2_i \leq \frac{1}{4} L_i$

b. If the panel has only *one side* flange`

$$B1_i = min\begin{cases}\frac{1}{12}L_i\\ 6\,Tw_i\\ \frac{1}{2}AR\end{cases}$$

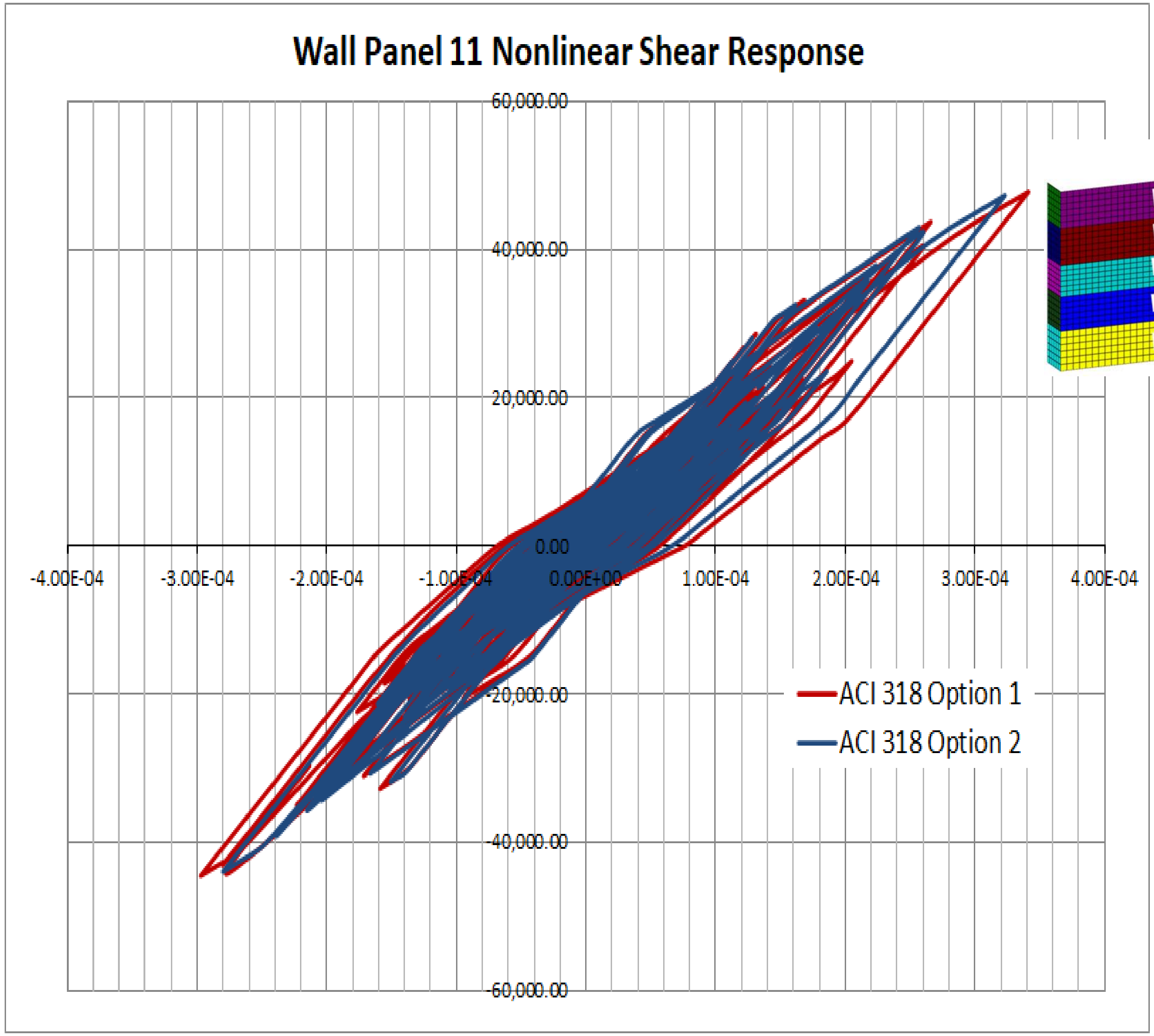
$$B2_i = min\begin{cases}\frac{1}{12}L_i\\ 6\,Tw_i\\ \frac{1}{2}AL\end{cases}$$

$$L1C_i = B1_i + Tw_i + B2_i$$

$$L2C_i = L1C_i$$

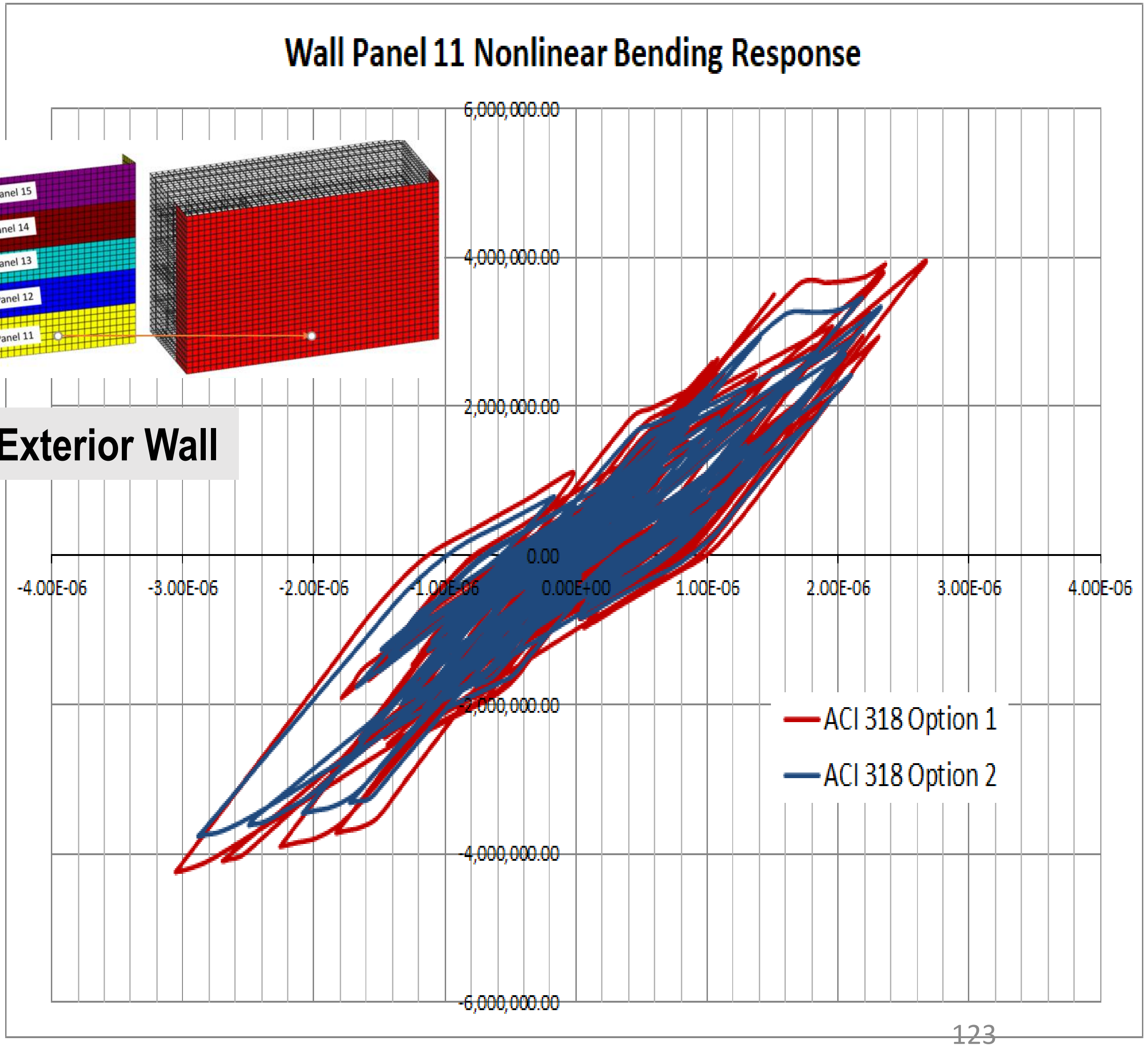
ACI Effective Flange Sizes Impact on Wall Response for 0.70g Input

SHEAR



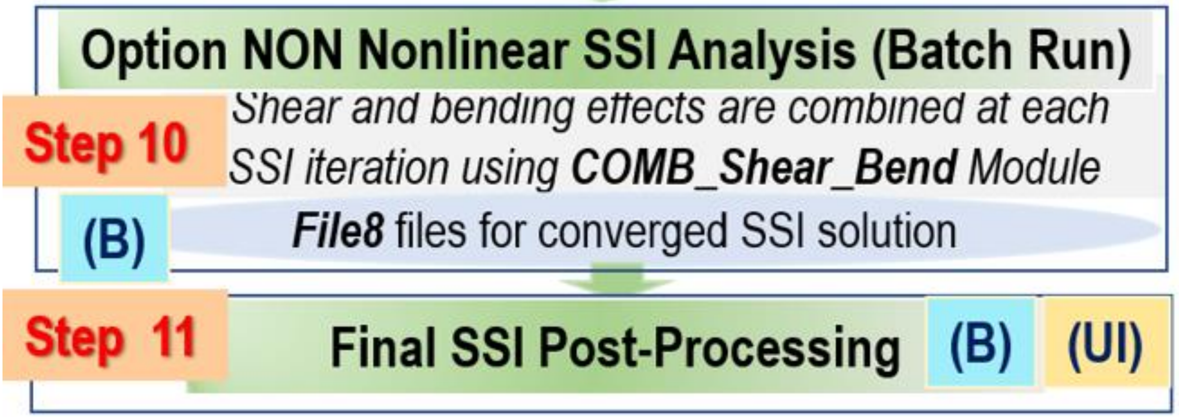
Exterior Wall

BENDING



Integrate Final Model and Perform Nonlinear Analysis (Steps 9,10)

Step 9:
User integrates the Modelname_New.pre file with BBC .pre files to create a complete input for NONLINEAR module. This can be done automatically by using a UI script as shown in Demos 18 and 19.



Step 10:

Option NON Nonlinear SSI Analysis Batch Run

The new Option NON includes new hysteretic models (2 JEAC 4601 models and 2 Hybrid models) applicable for both the shear and bending wall in-plane deformation for 3DFEM (Shell models). The shear and bending deformation can be combined using the *COMB_Shear_Bend* module as described in Demos 18 and 19.

Step 11:

Post-Processing:

The main results of the nonlinear SSI analysis are the FILE8 and FILE4 (.n4) files for the converged solution that can be post-processed exactly like for a linear analysis to compute structural node displacements and accelerations, and element stresses.

Modeling of Interaction Between Shear and Bending Effects

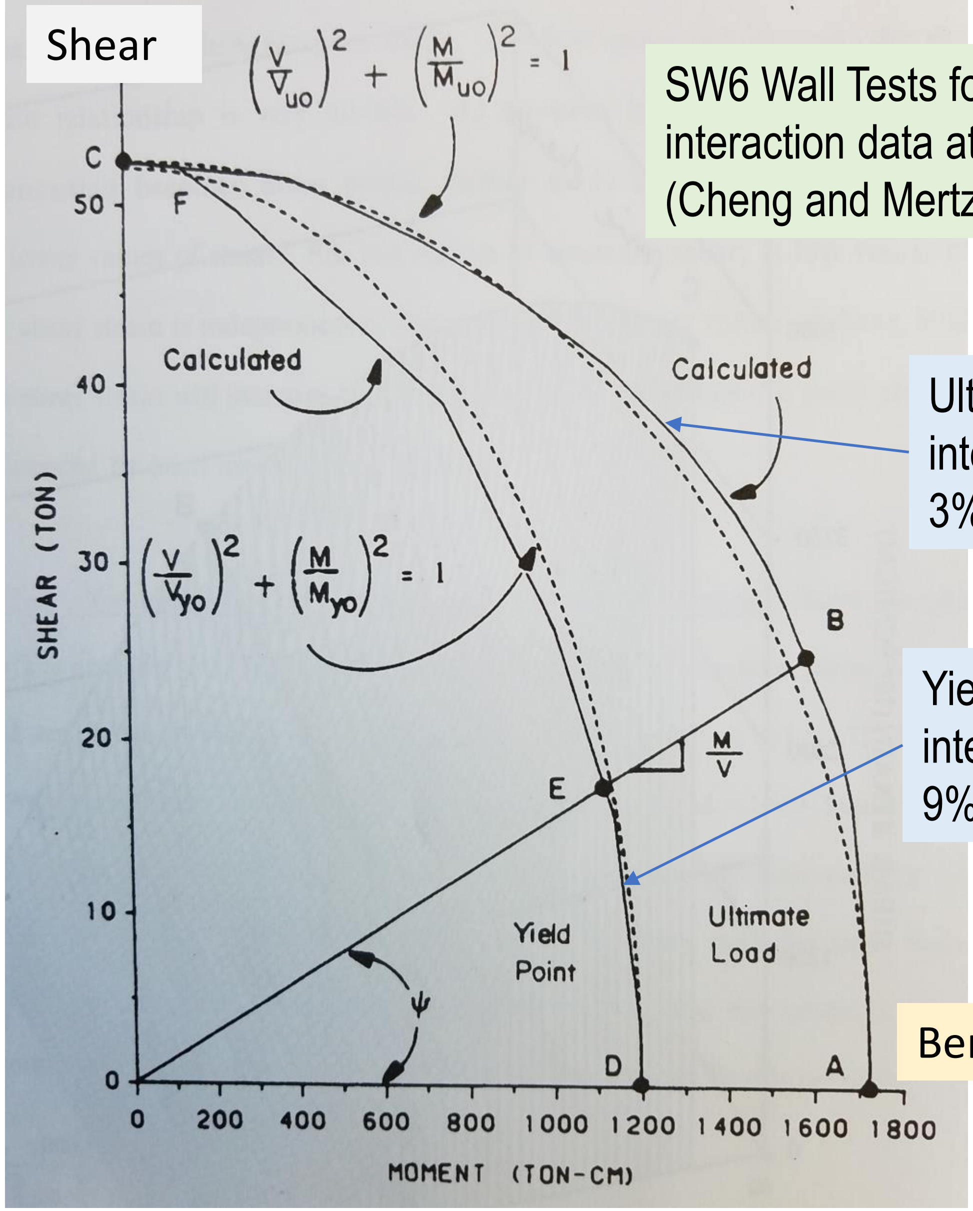
These interaction effects are included at each SSI iteration by the following Option NON options:

1) ***<u>Shear Governing:</u>*** *A*ssuming that the shear stiffness variations are governing the wall stiffness degradation at each SSI iteration *(RC wall material stiffness degradation based on the Shear hysteretic models only, i.e. material Esb=Es, fully coupled)*

2) ***<u>Bending Governing</u>:*** *A*ssuming that the bending stiffness variations are governing the wall stiffness at each SSI iteration *(RC wall material stiffness degradation based on the Bending hysteretic models only, i.e. material Esb=Eb, fully coupled)*

3) ***<u>Shear and Bending</u>:*** The equivalent bending and shear stiffnesses are computed independently at each SSI iteration *(RC wall material stiffness degradation based on both Shear and Bending hysteretic models, i.e. material Esb is different from Es and Eb). An elliptical interaction curve for combining the shear and bending stiffnesses is applied at each SSI iteration.*

Nonlinear Shear-Bending Interaction Effects (Comb_Shear_Bend)

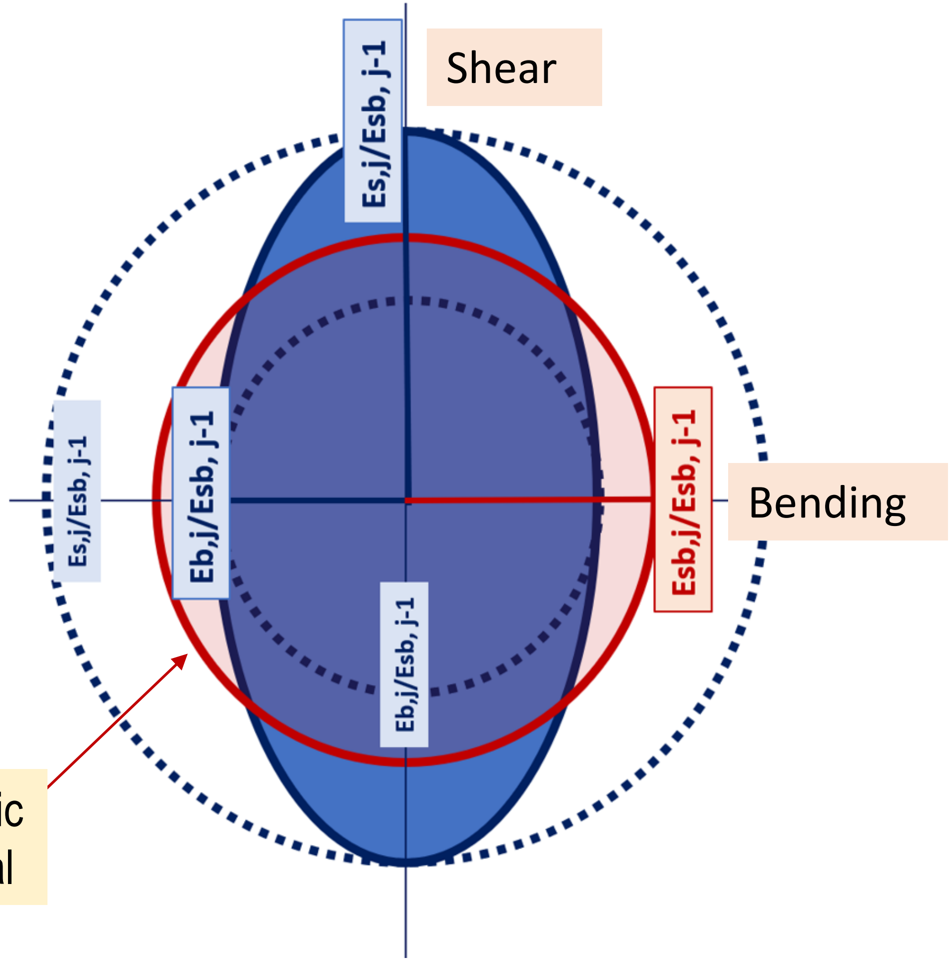


SW6 Wall Tests for the shear-bending interaction data at NCKU laboratories (Cheng and Mertz, 1989)

Ultimate state shear-bending interaction equation has only a 3% error comparing with tests

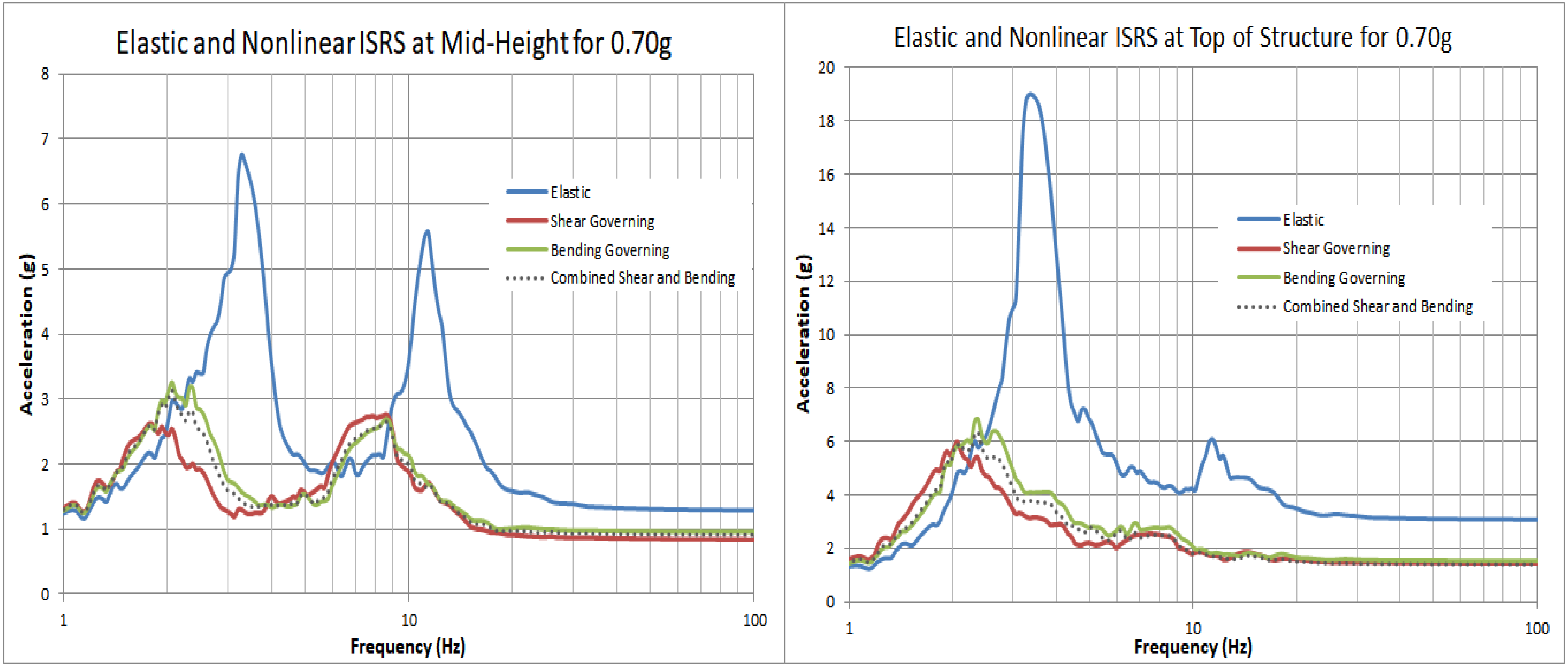
Yielding state shear-bending interaction equation has only a 9% error comparing with tests

In Option NON, shear-bending interaction effects are included *at each SSI iteration* based on an ellipsoidal interaction curve.

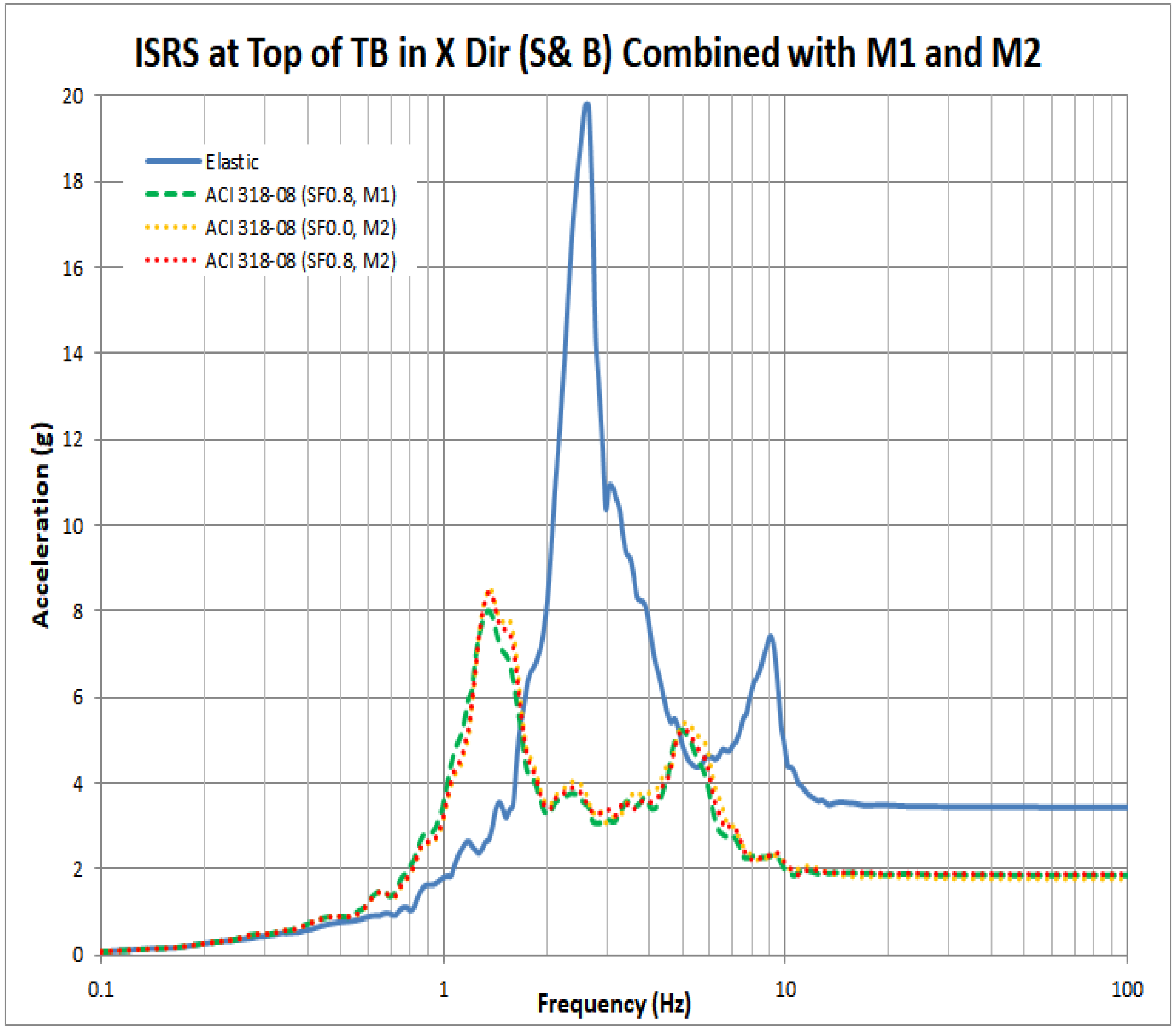


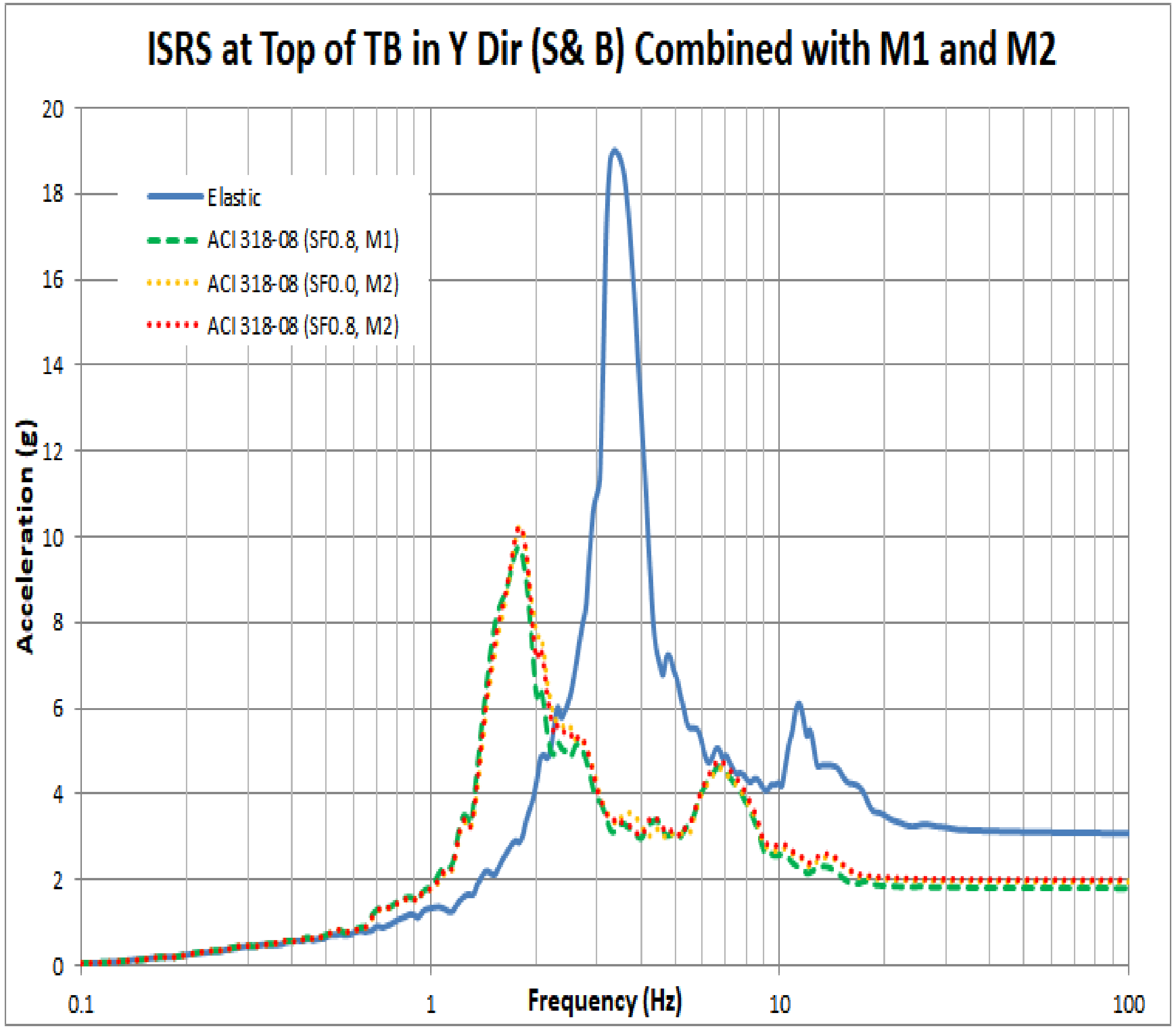
Isotropic material

Computed ISRS for 0.70g: 1) Shear Governing, 2) Bending Governing and 3) Combined Shear and Bending with M1

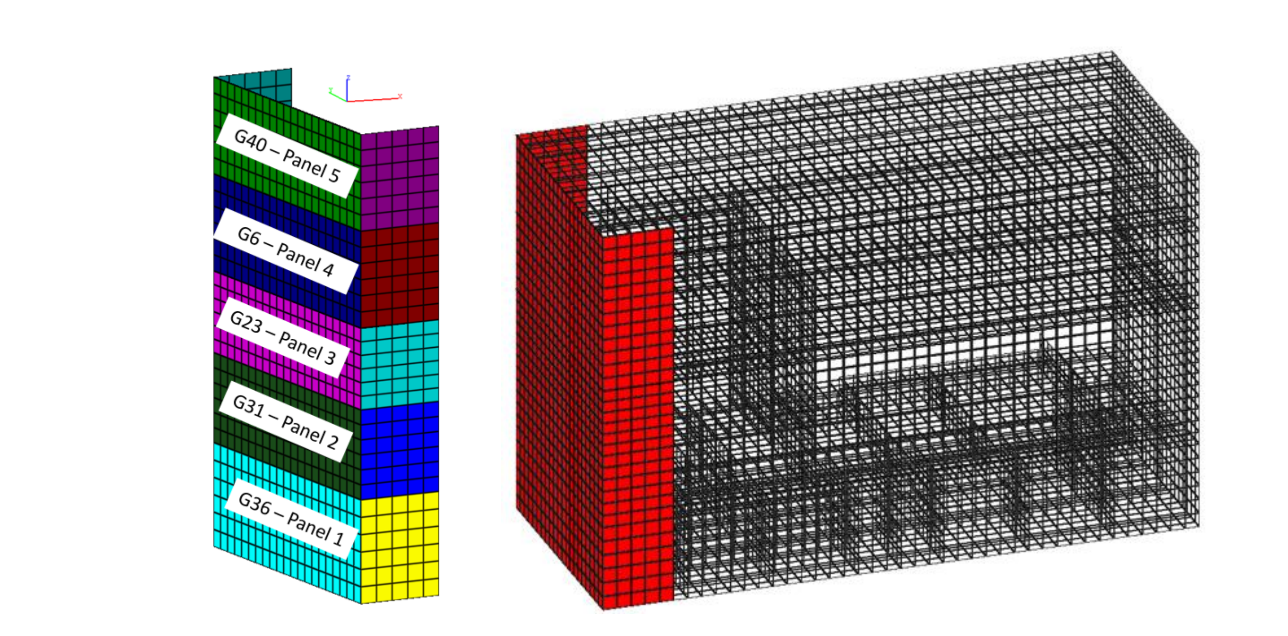


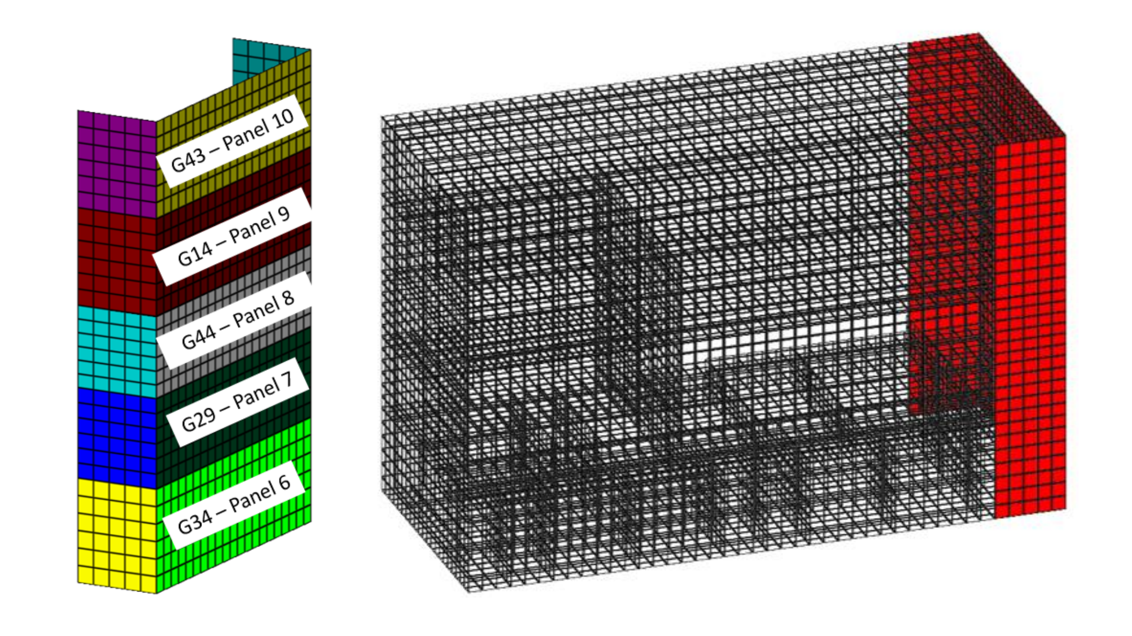
Combining Shear and Bending Interaction Effects Stiffness. Comparing ISRS Results for M1 and M2 Methods

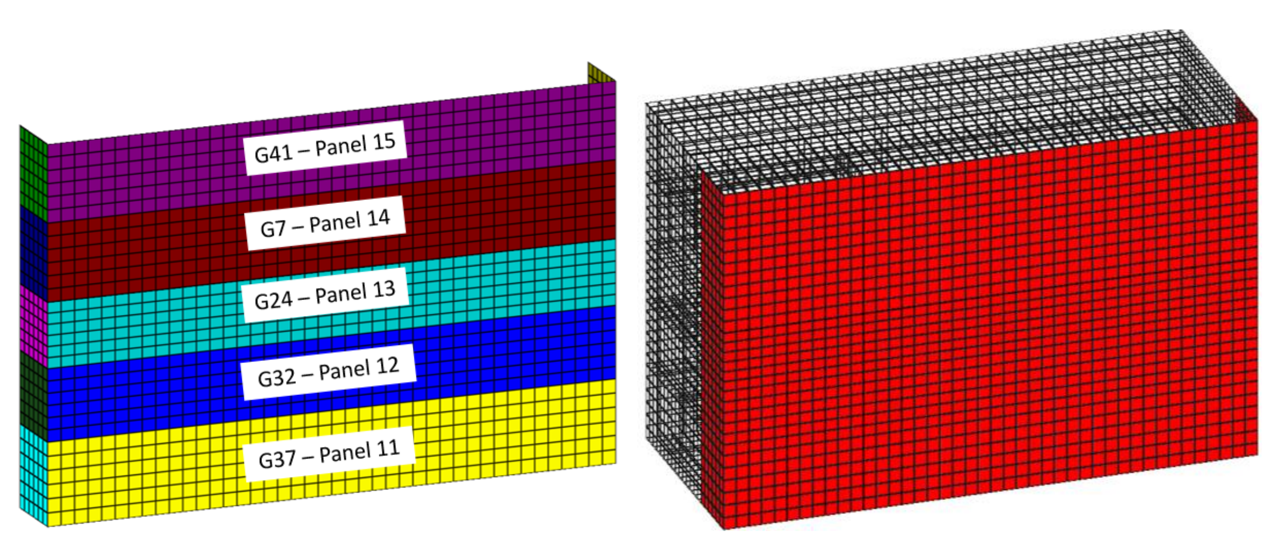


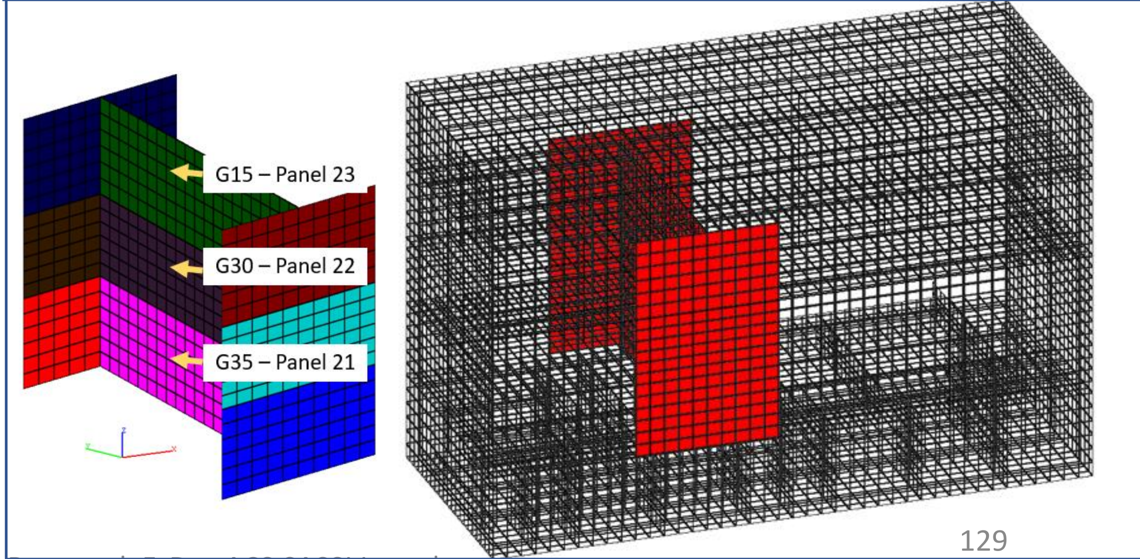


ABSHEAR - Comparative Results for US and Japan Standards

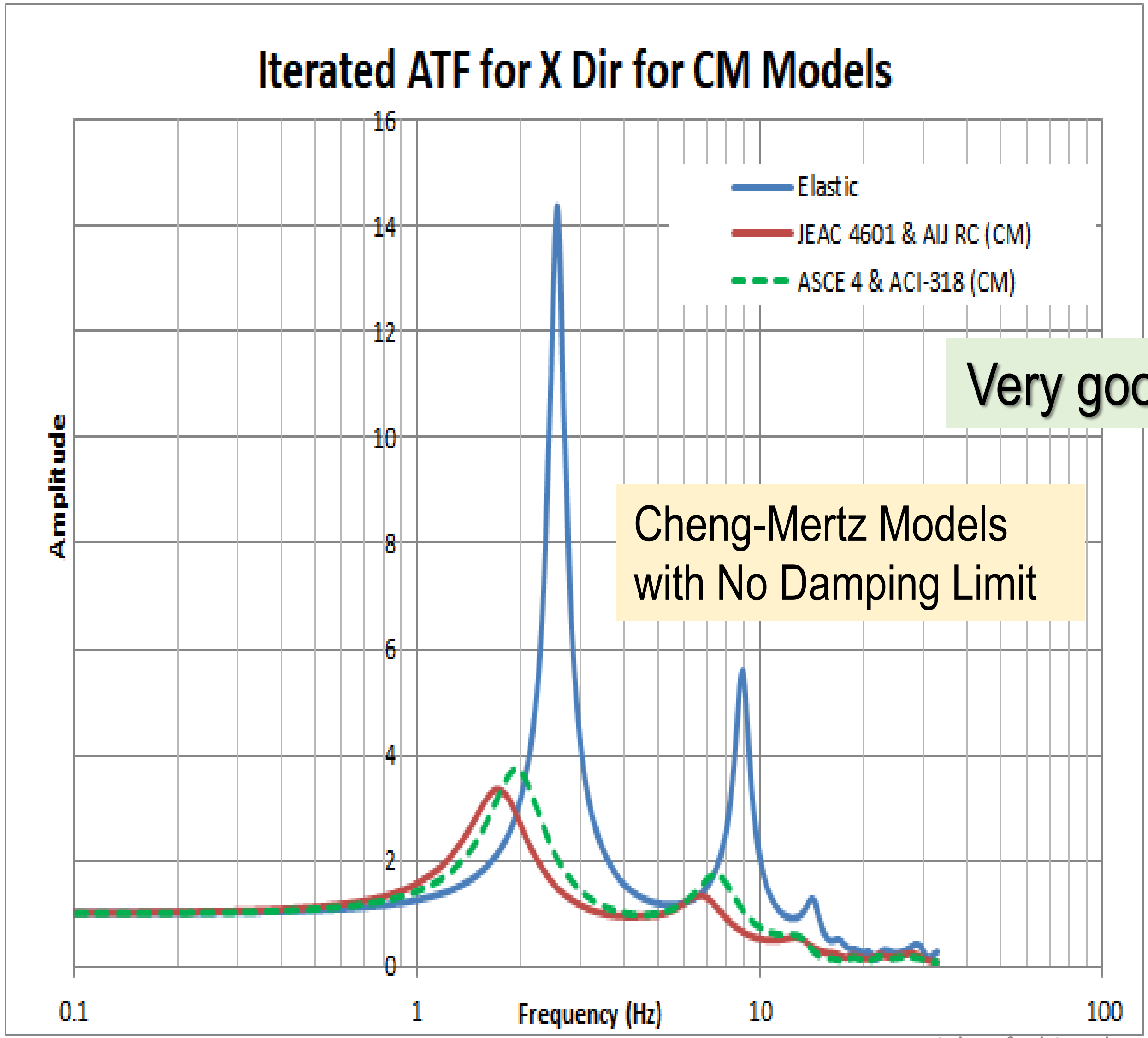




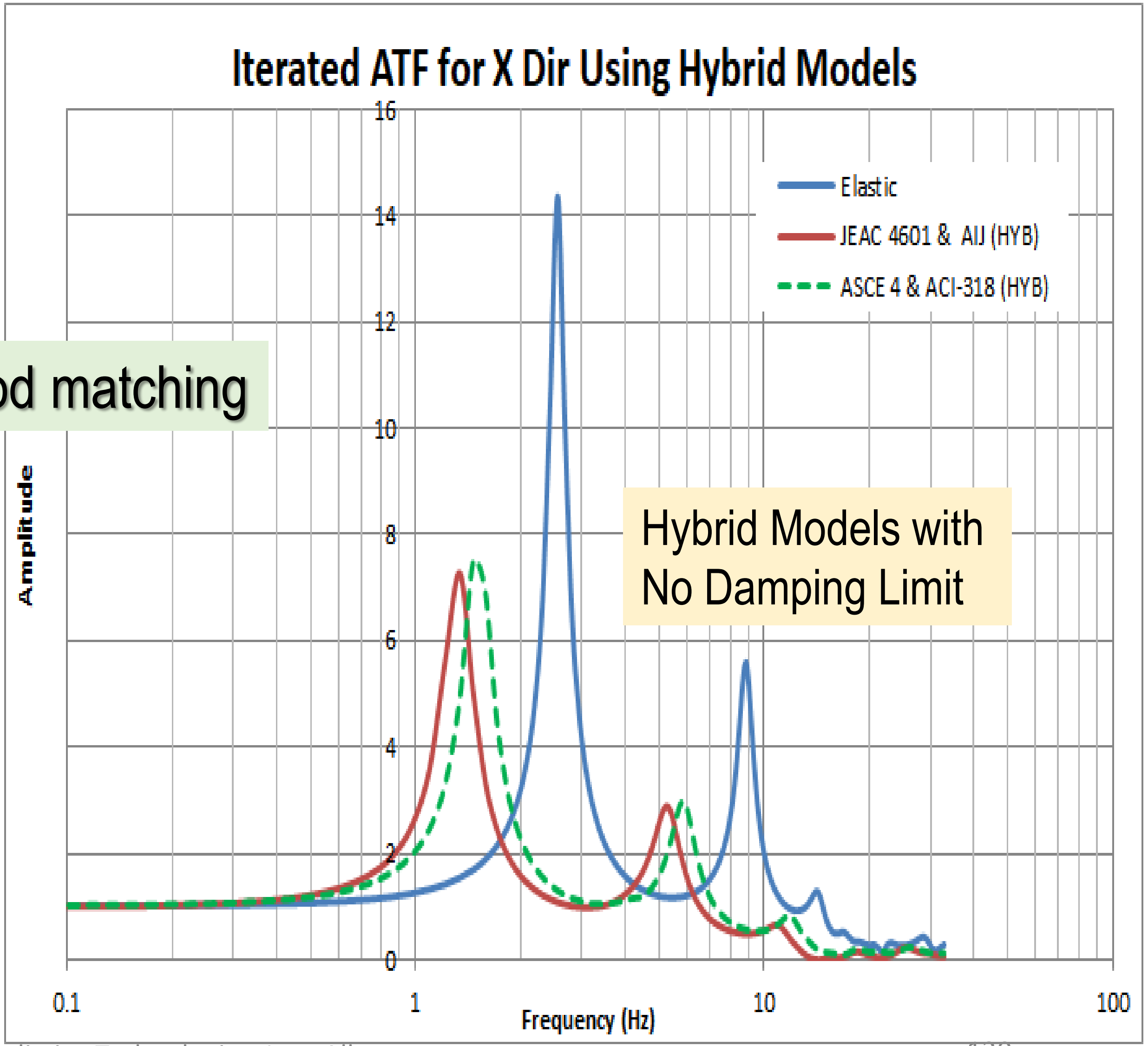




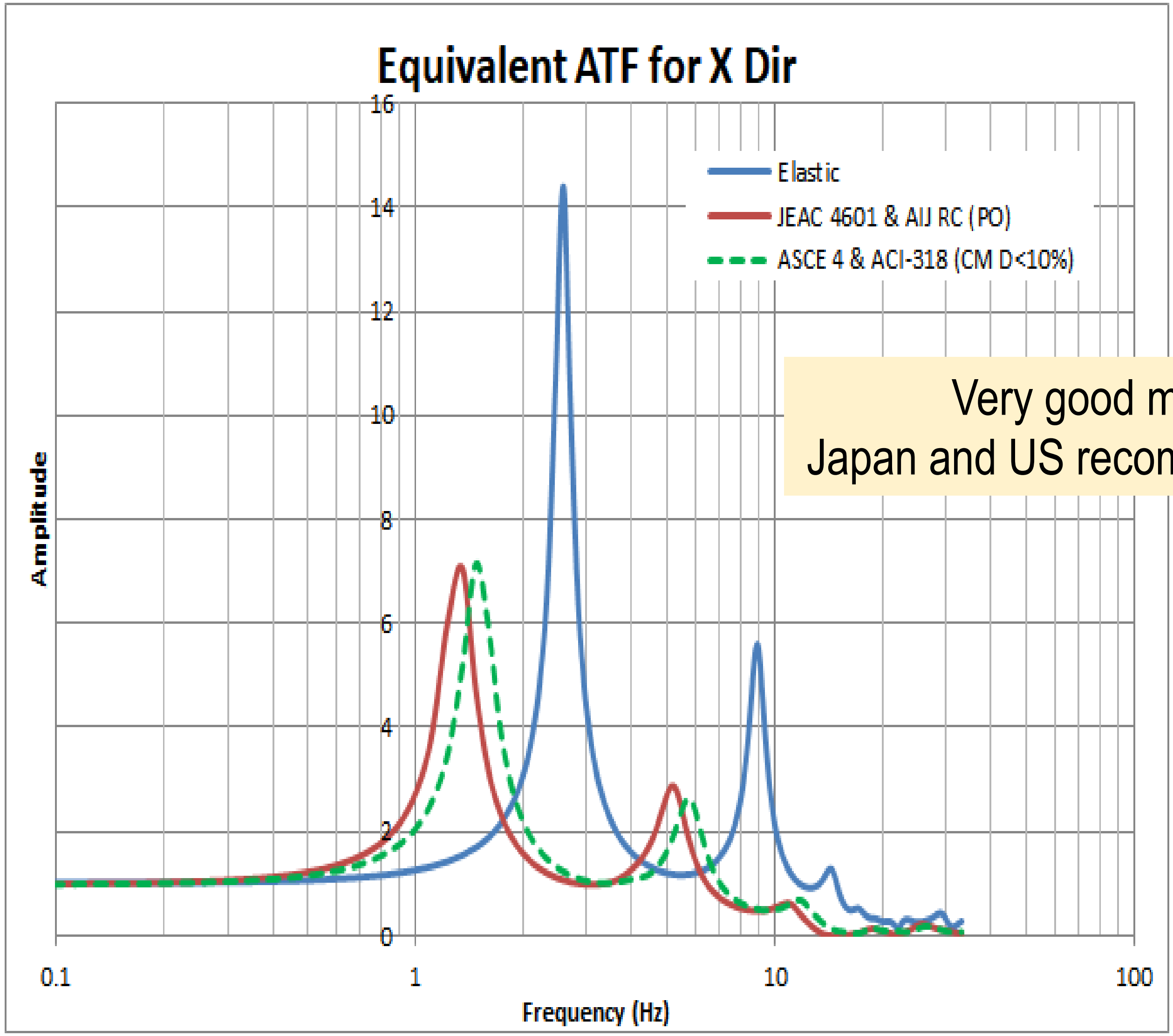
Iterated ATF Response Using Same Hysteretic Models for US and Japan Design Practices

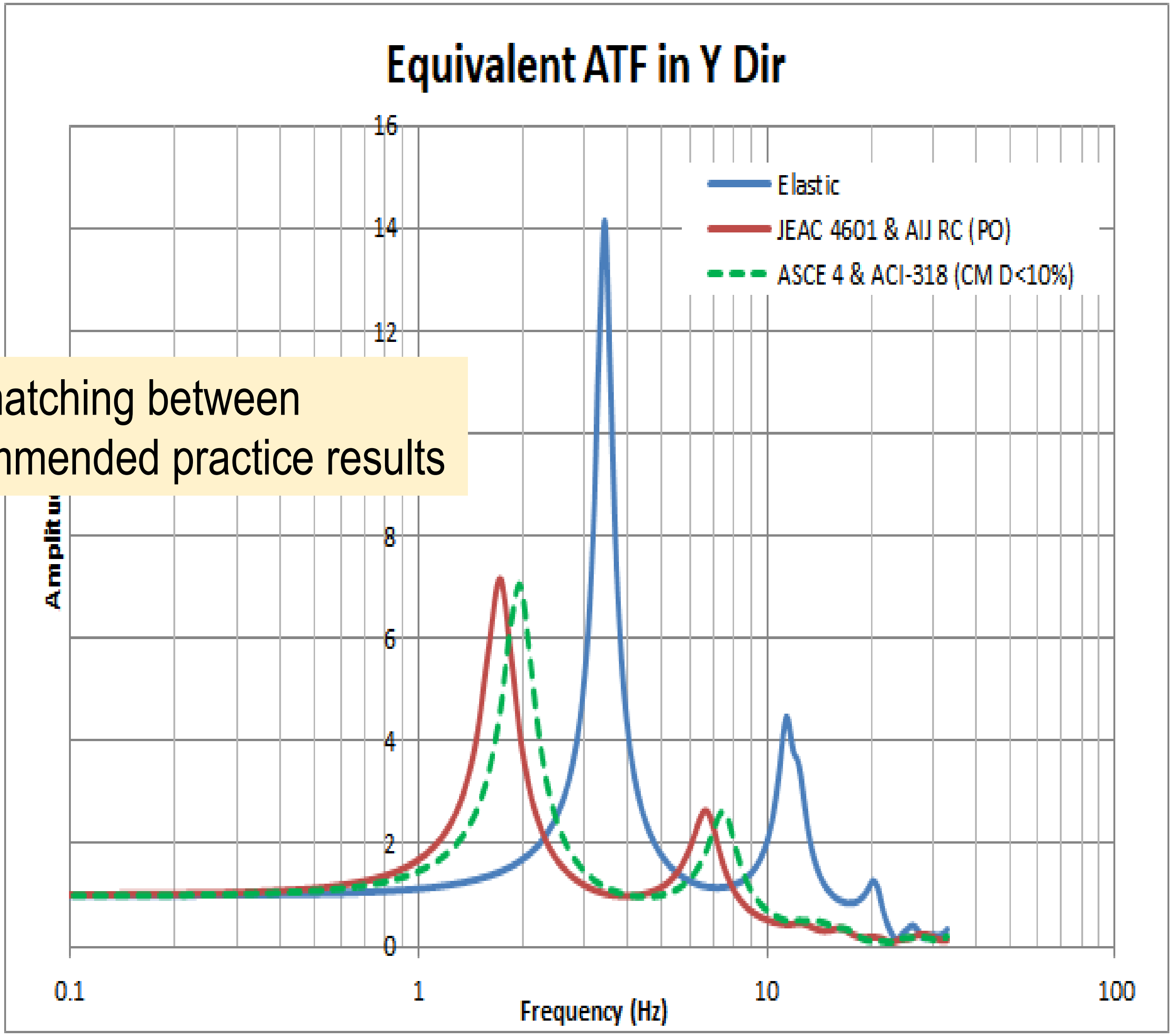


Very good matching



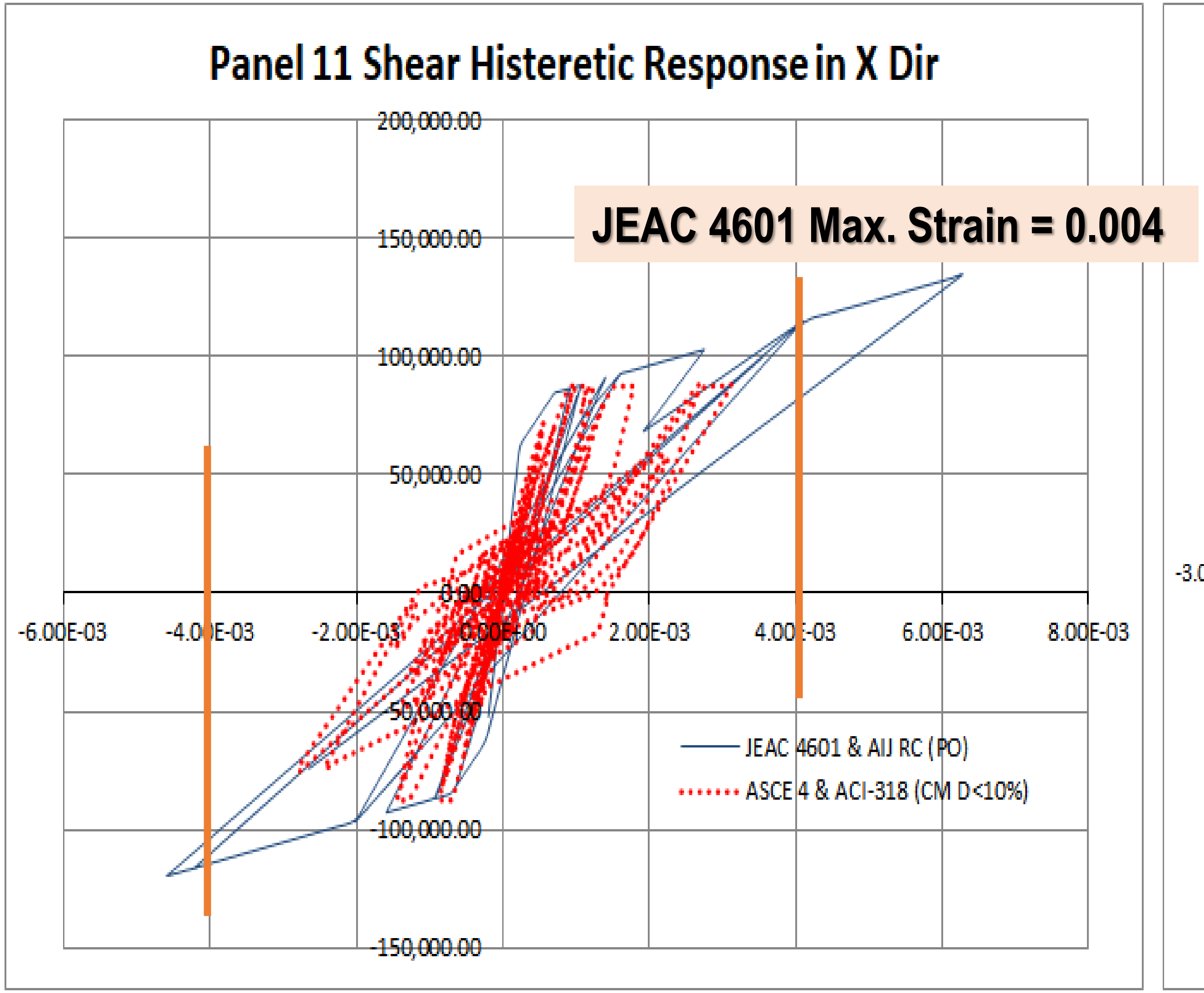
Iterated ATF for JEAC PO Models and CM Models with D<10%

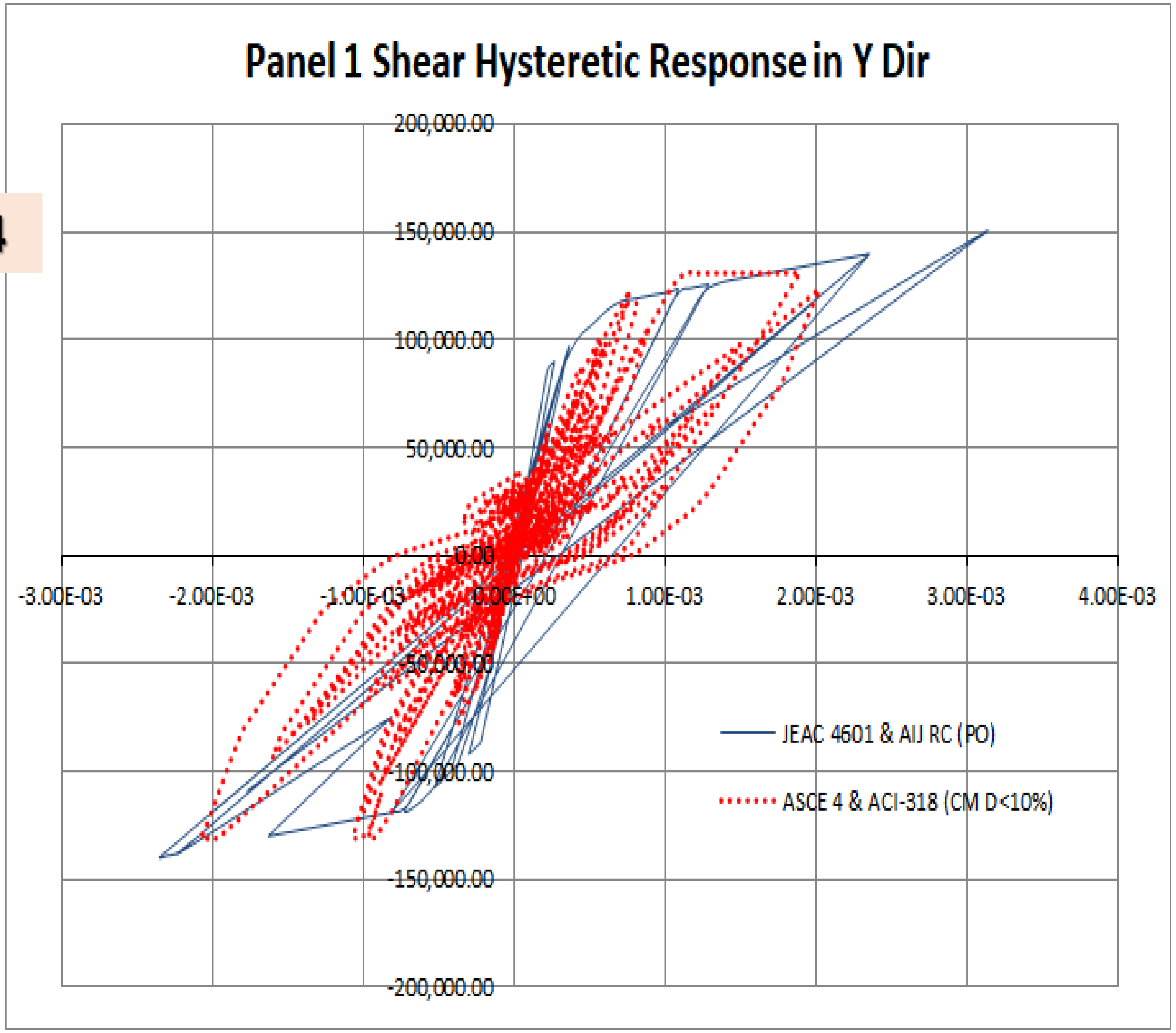




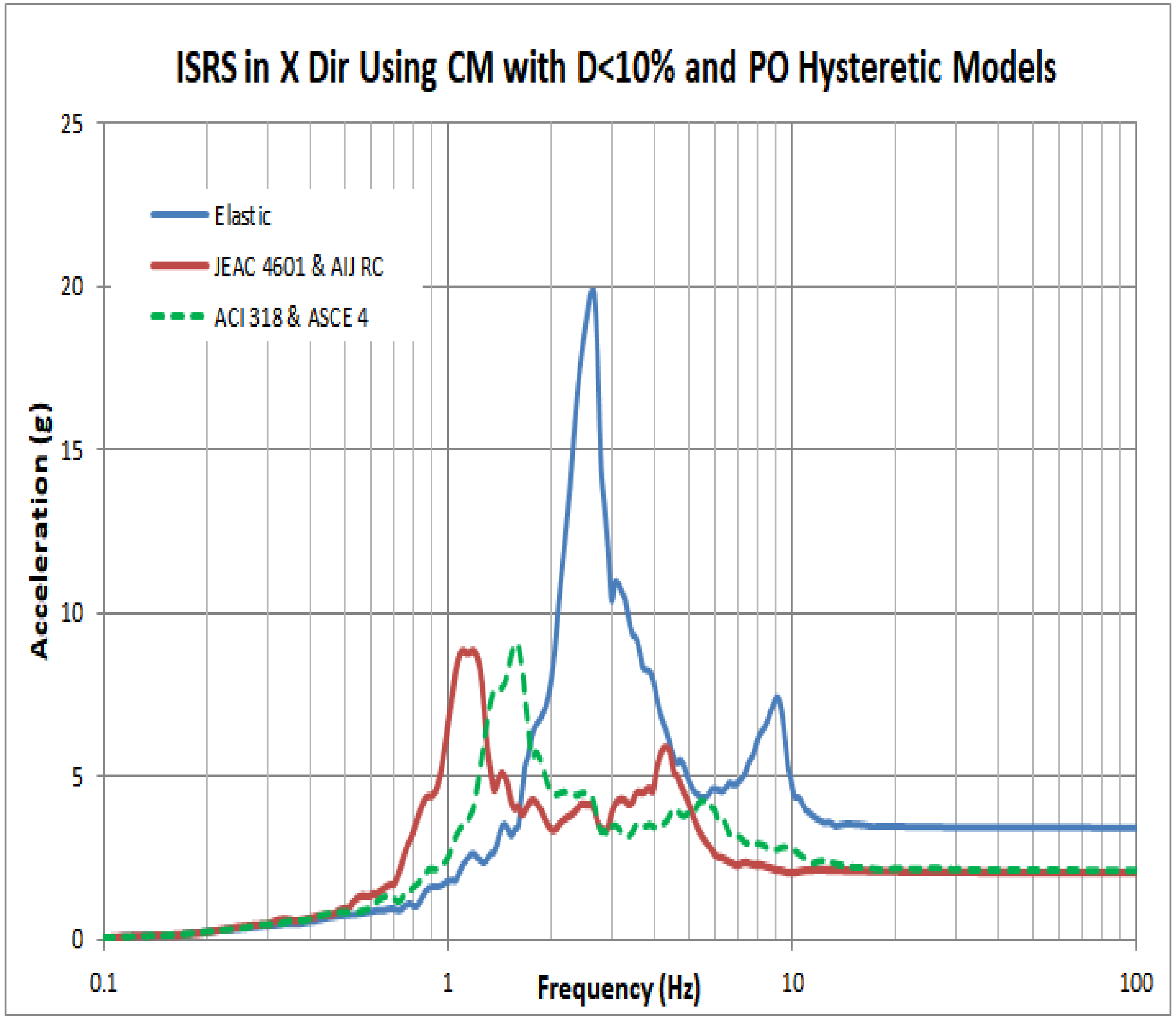
Very good matching between
Japan and US recommended practice results

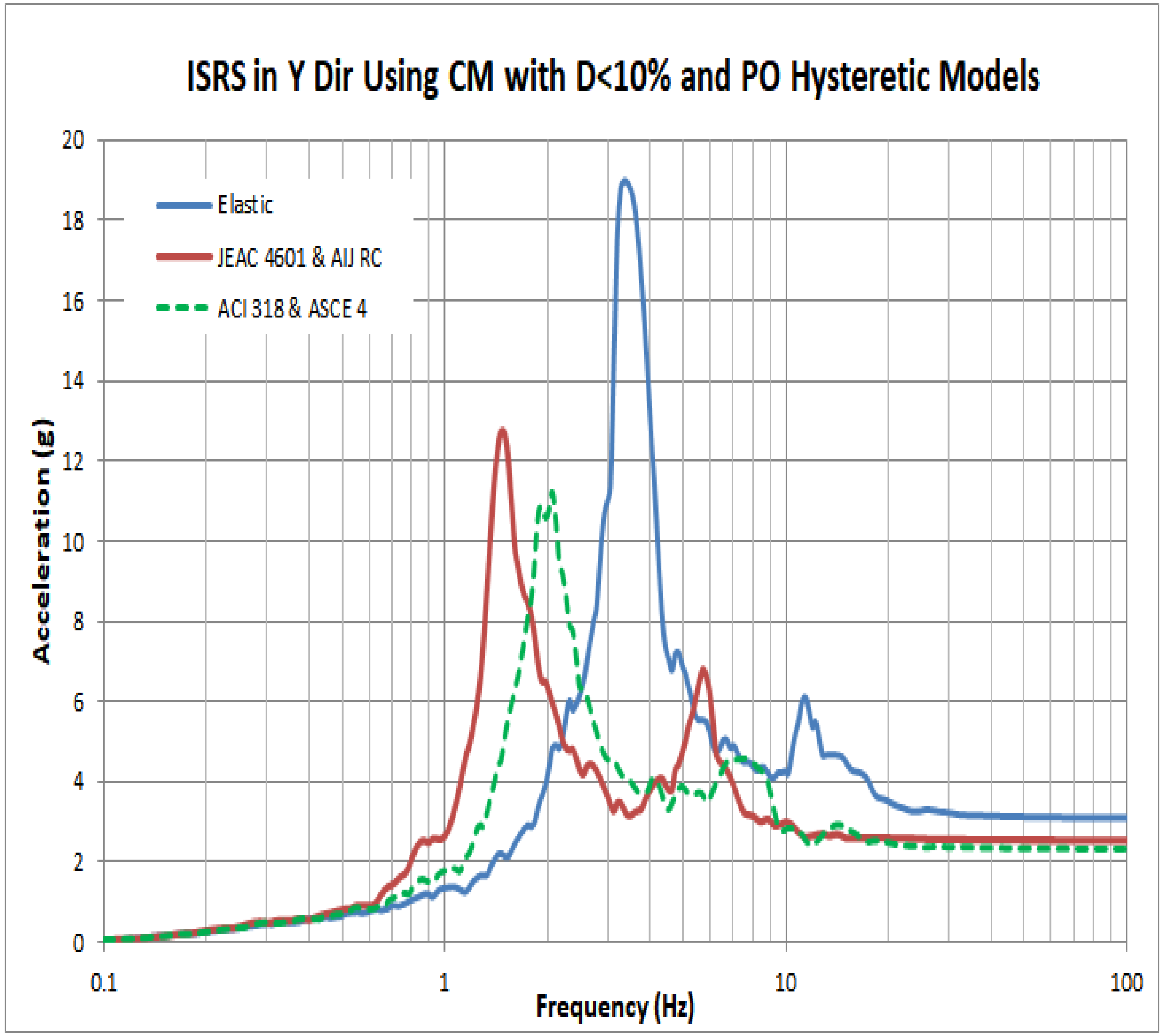
Shear Hysteretic Response for JEAC PO and CM with D<10%





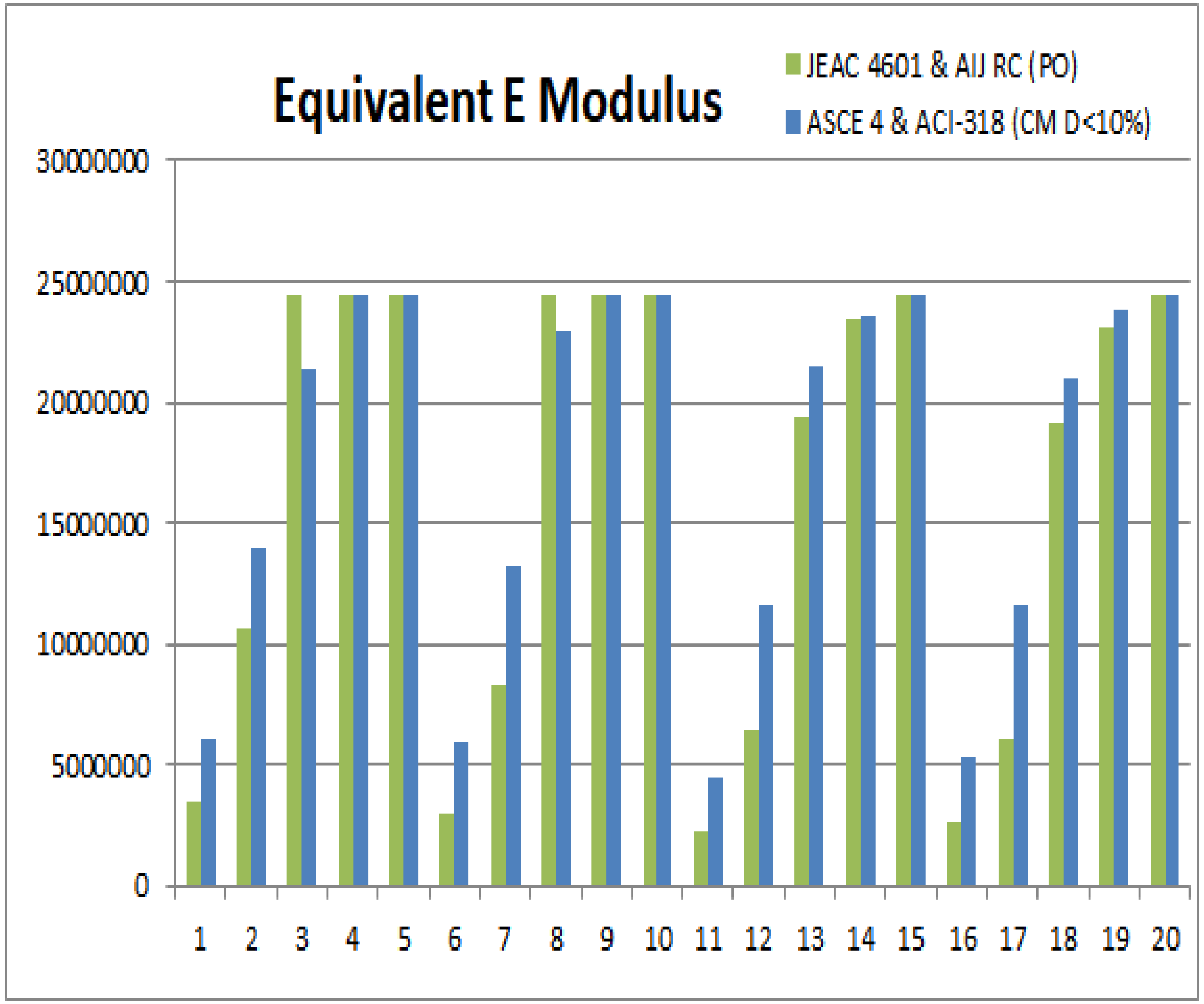
Iterated ISRS for JEAC PO Models and CM Models with D<10%

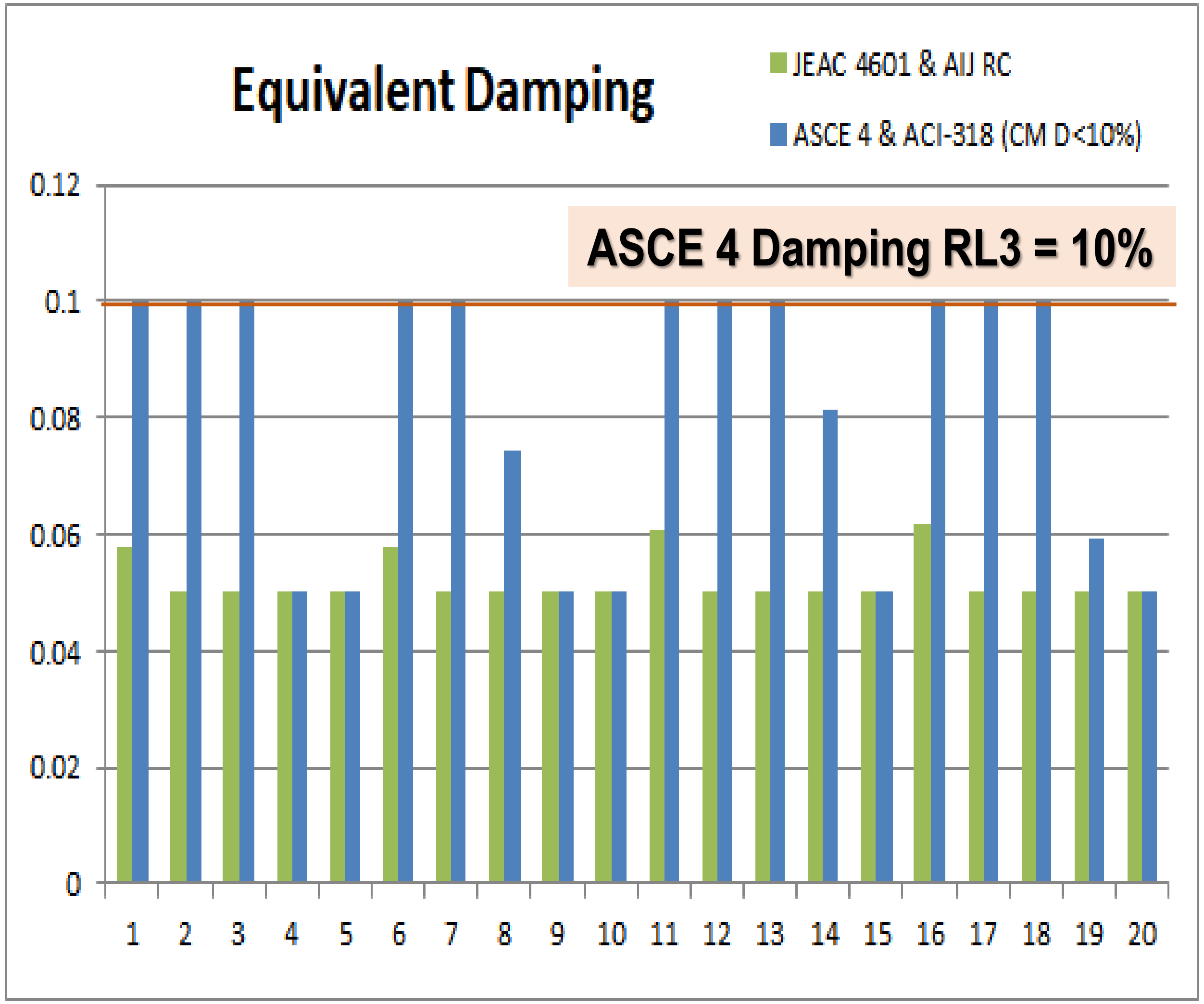




Iterated Walls Stiffness and Damping for 0.70g RG1.60 Input

Using JEAC PO and CM Models with Damping < 10% per ASCE 4 Section 3 Recommendation

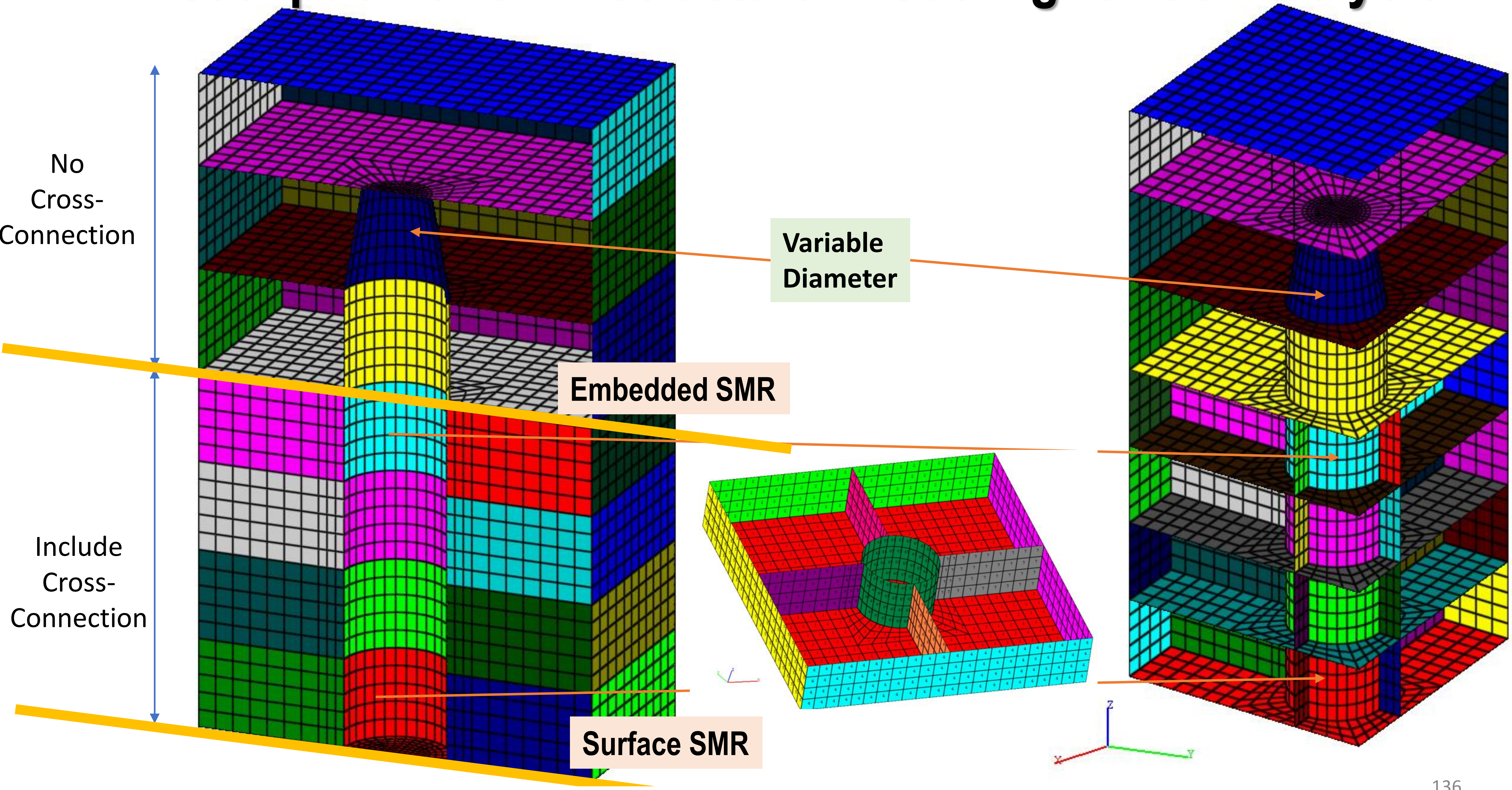




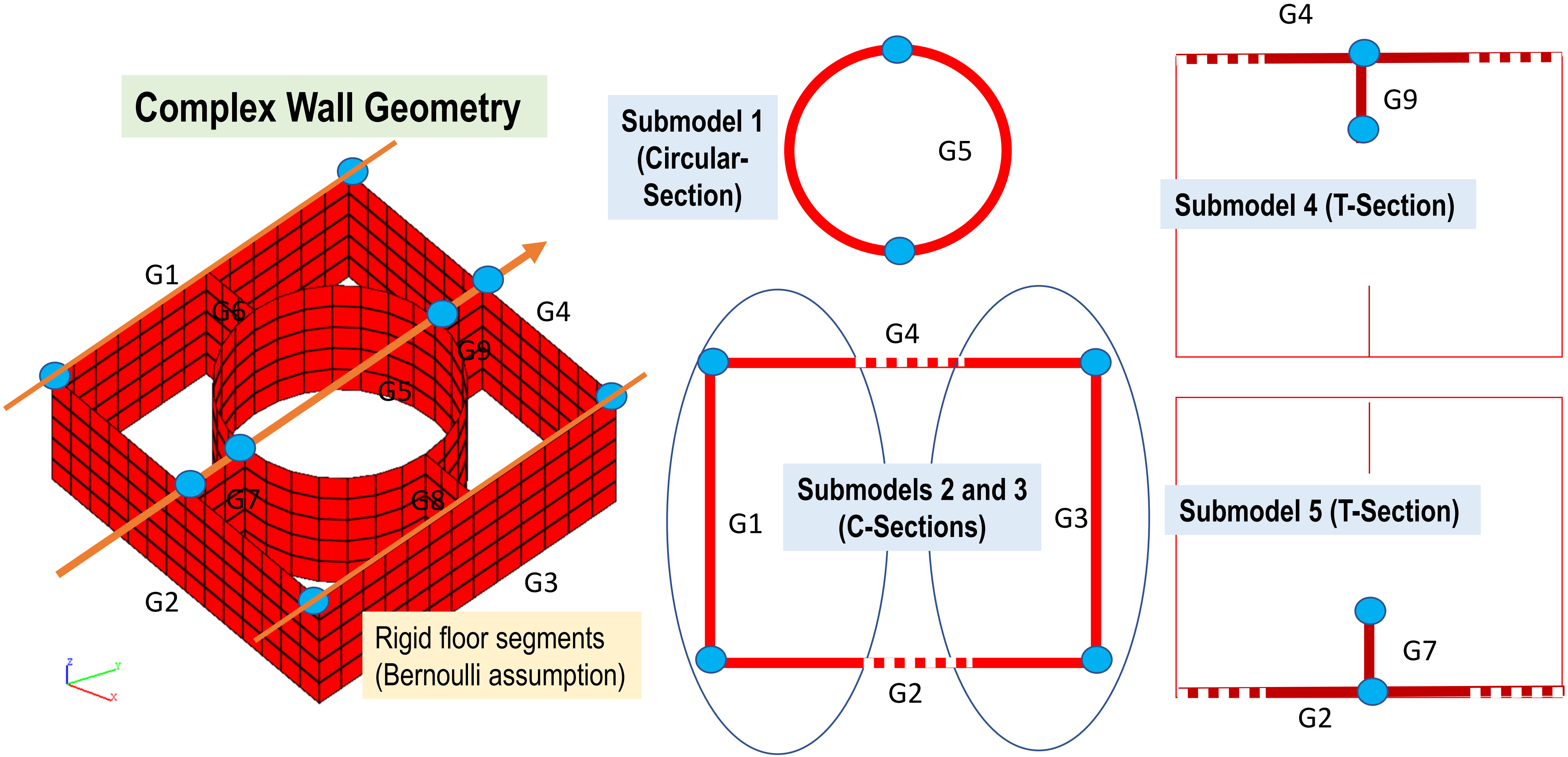
Nonlinear SSI Analysis for Surface and Embedded SMR SSI Models (Demo 19)

(Including 2D Fiber Model)

Description of SMR Structural Modeling for SSI Analysis



Handling Complex Geometries Using Multiple Submodels (.pre)

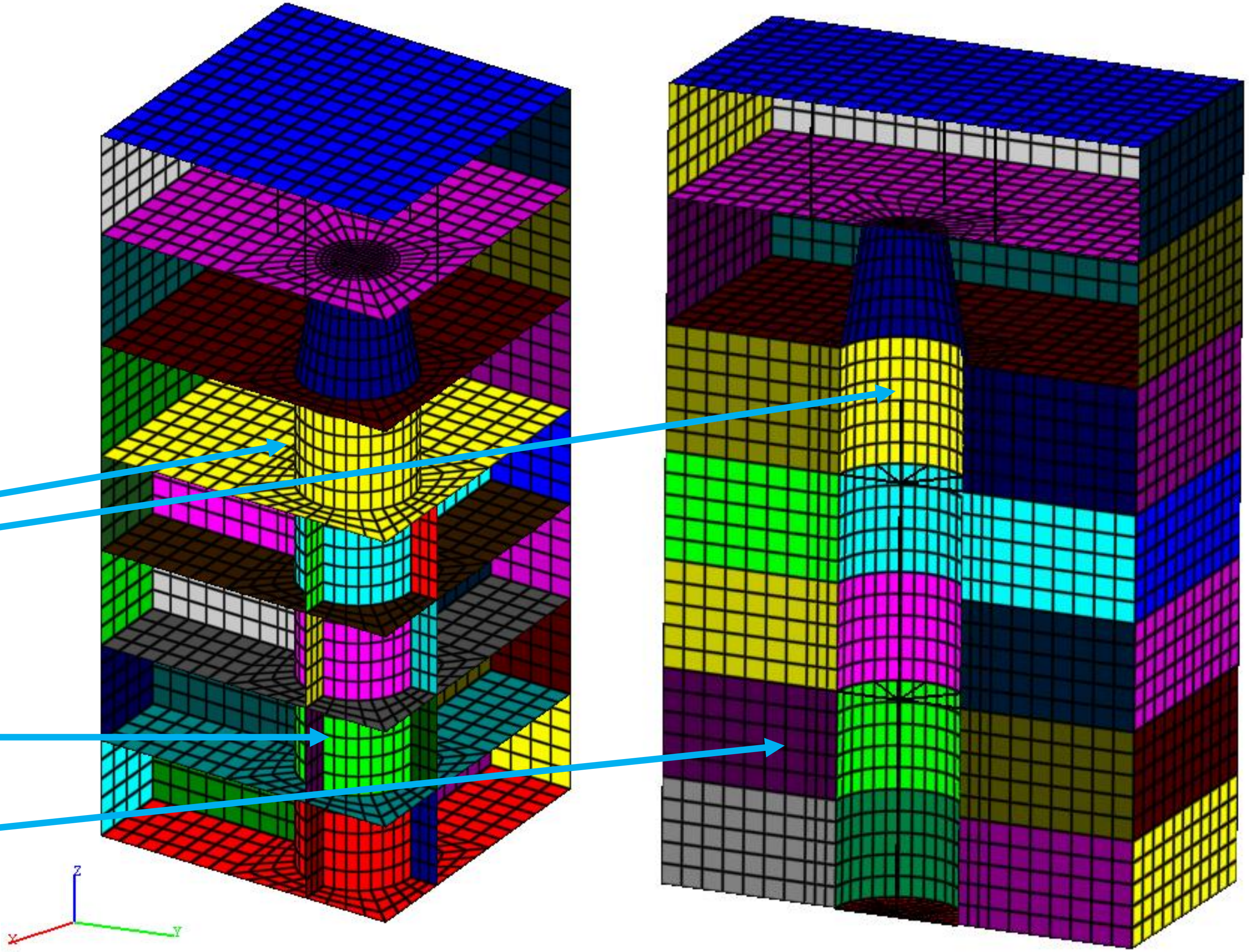


Splitting SMR Model in 9 Wall Submodels

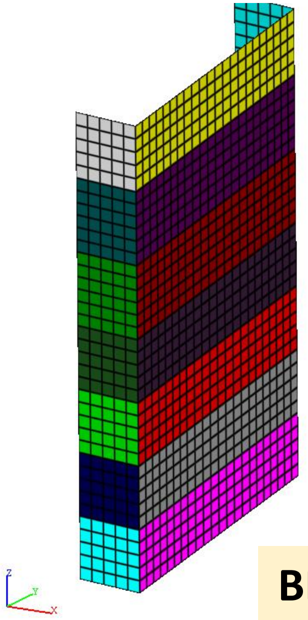
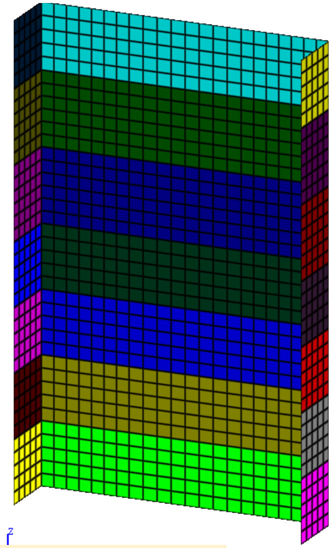
SMR_SURF_cuts - Notepad

File Edit Format View Help

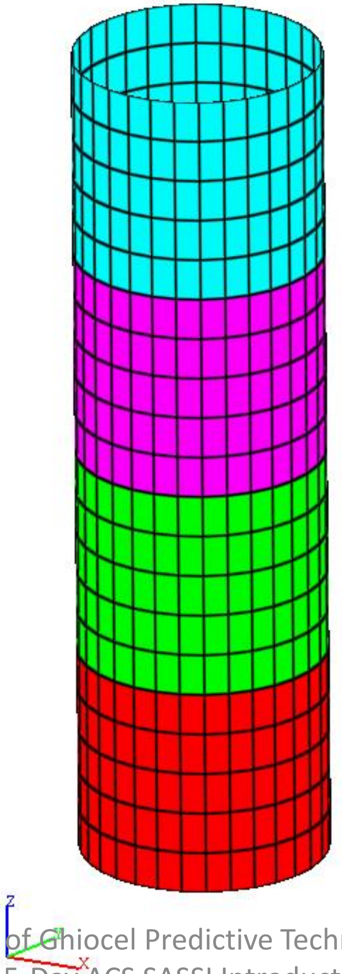
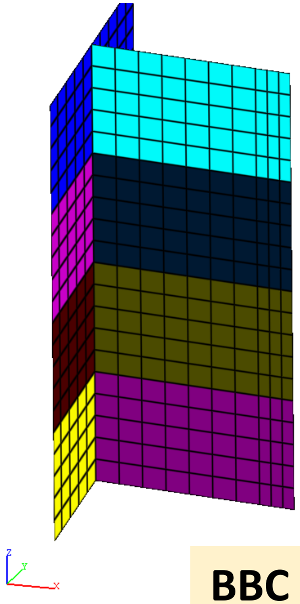
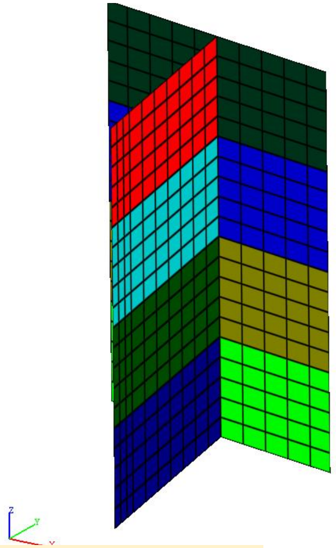
```
var,PATH,C:\SSI\SMR2021\Model-Pre-files\
var,MODELNAME,SMR_SURF_Final
DMODEL,0
DMODEL,1
DMODEL,2
DMODEL,3
DMODEL,4
DMODEL,5
DMODEL,6
DMODEL,7
DMODEL,8
DMODEL,9
CUTCLR,1,9,1
ACTM,0

INP,@PATH[1]\@MODELNAME[1].pre

**************************************************
* CIRCULAR MIDDLE SECTION
CUTADD,1,65,RANGE,1,160,1
CUTADD,1,66,RANGE,1,160,1
CUTADD,1,67,RANGE,1,160,1
CUTADD,1,68,RANGE,1,160,1
CUTADD,1,69,RANGE,1,192,1
CUT2SUB,1,1
ACTM,1
WRITE,@PATH[1]\@MODELNAME[1]_circular.pre
ACTM,0

**************************************************
* INTERIOR MIDDLE WALLS 1
CUTADD,2,6,RANGE,1,50,1
CUTADD,2,15,RANGE,1,50,1
CUTADD,2,24,RANGE,1,50,1
CUTADD,2,33,RANGE,1,50,1
CUTADD,2,42,RANGE,1,60,1
CUTVOL,2,-25,25,50,50,-118,-4.3
CUT2SUB,2,2
ACTM,2
WRITE,@PATH[1]\@MODELNAME[1]_interior1.pre
ACTM,0

**************************************************
```



SMR Wall Submodels (4 Exterior, 1 Circular, 4 Connections)

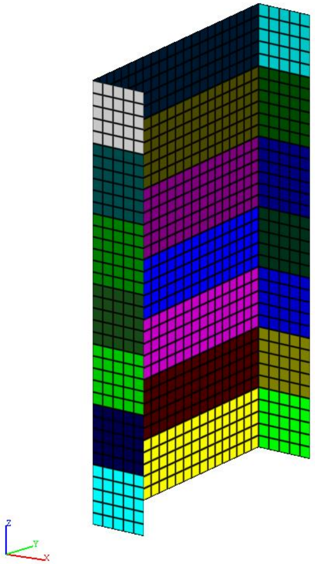
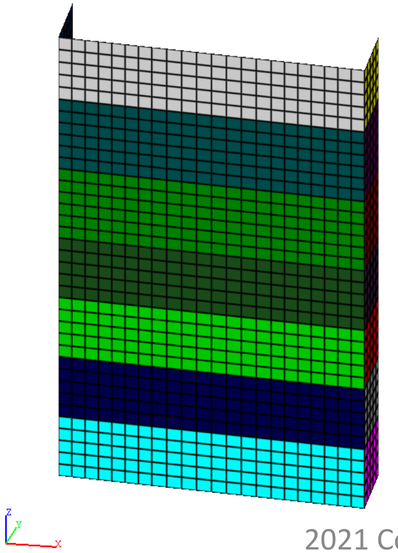
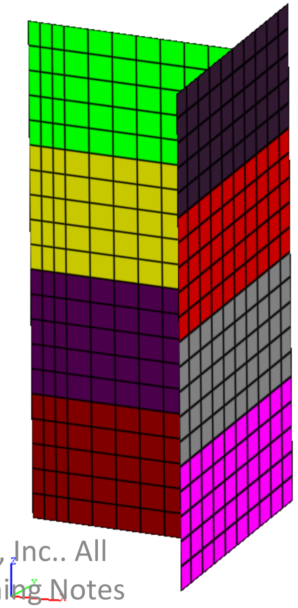
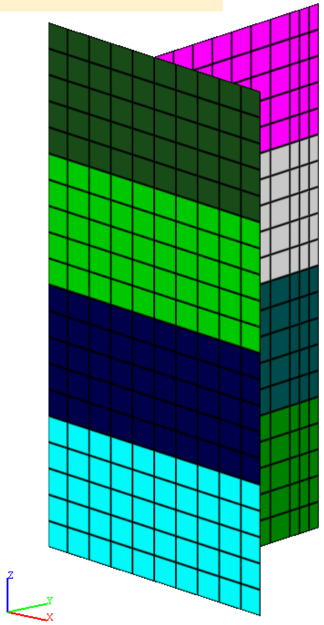



BBC_2DFiber
Dw=0

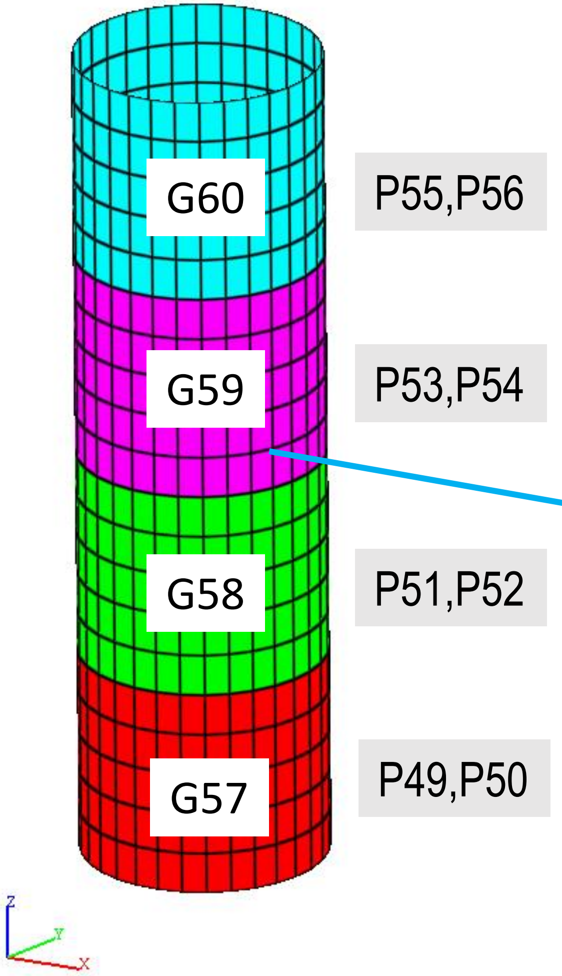




BBC_JEAC_ACI (Dw>0)
BBC_2DFiber (Dw<0)

BBC_JEAC_ACI (Dw>0)
BBC_2DFiber (Dw<0)

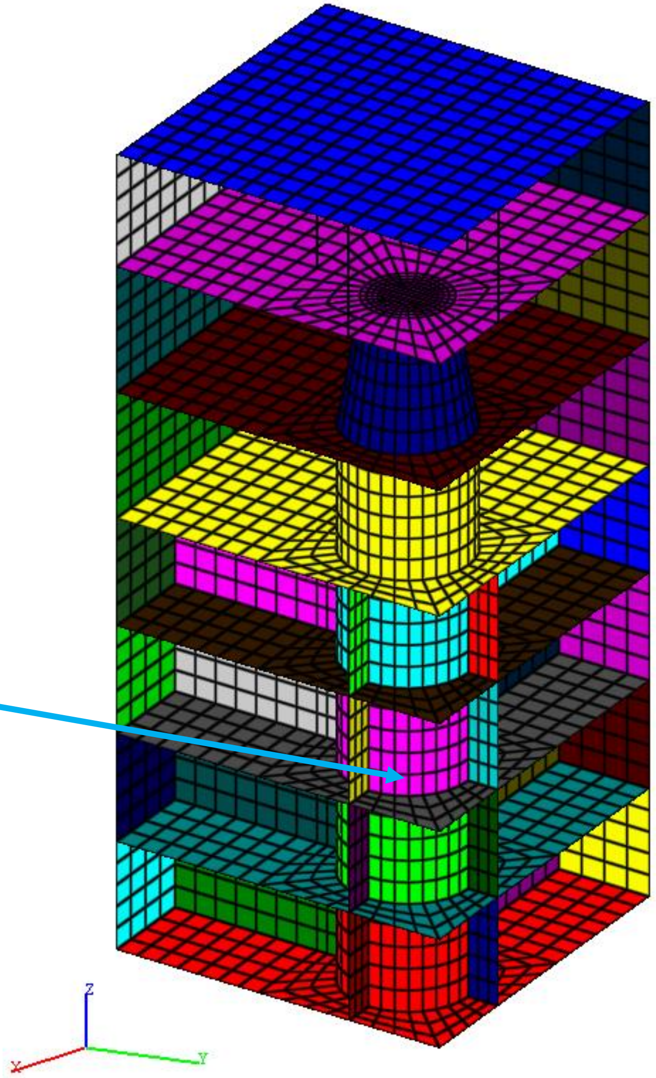





SMR Panel Numbering for Circular Closed Section Wall

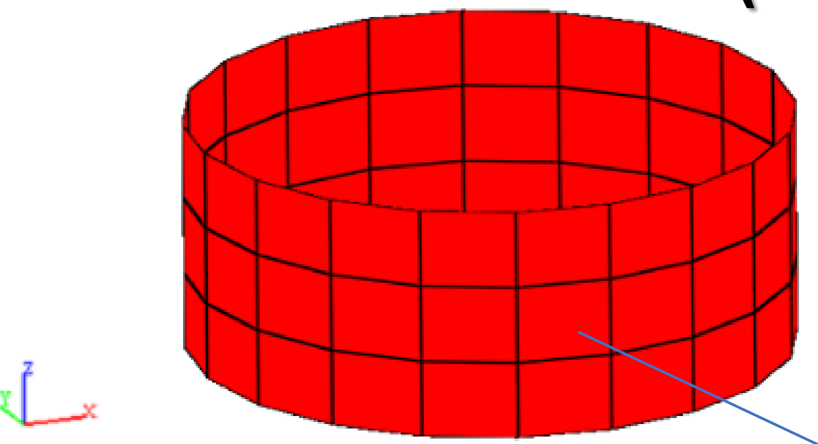


Panel Numbering:

P,49,57,20,1,1 X
P,50,57,20,1,1 Y
P,51,58,20,1,1 X
P,52,58,20,1,1 Y
P,53,59,20,1,1 X
P,54,59,20,1,1 Y
P,55,60,20,1,1 X
P,56,60,20,1,1 Y



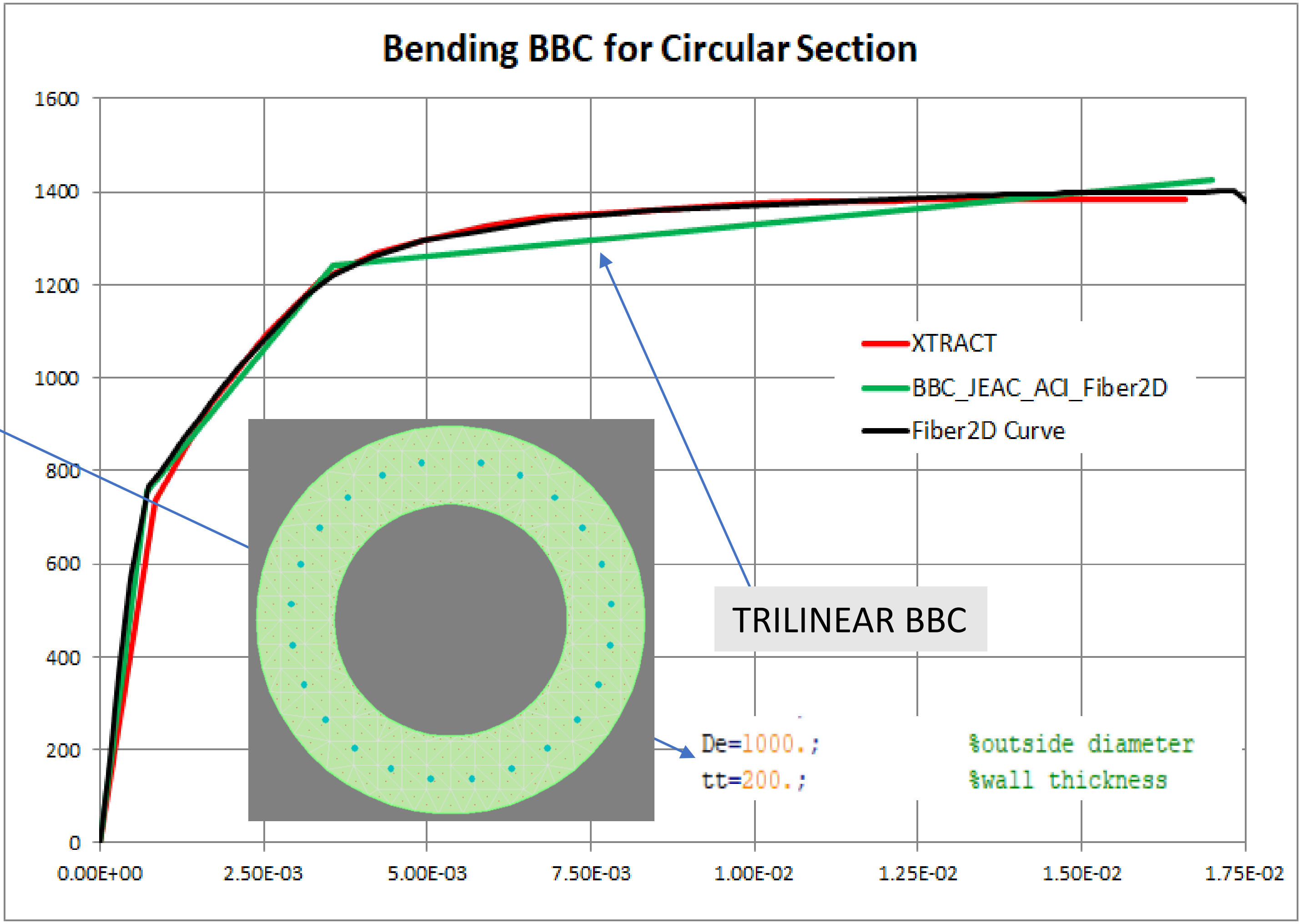
Circular Section Shape Verification of 2DFiber Model vs. XTRACT (BBC_JEAC_ACI_Fiber2D.exe)



Shear Wall Reinforcement:
Vertical Rebar Ratio = 1%
Horizontal Rebar Ratio = 1%

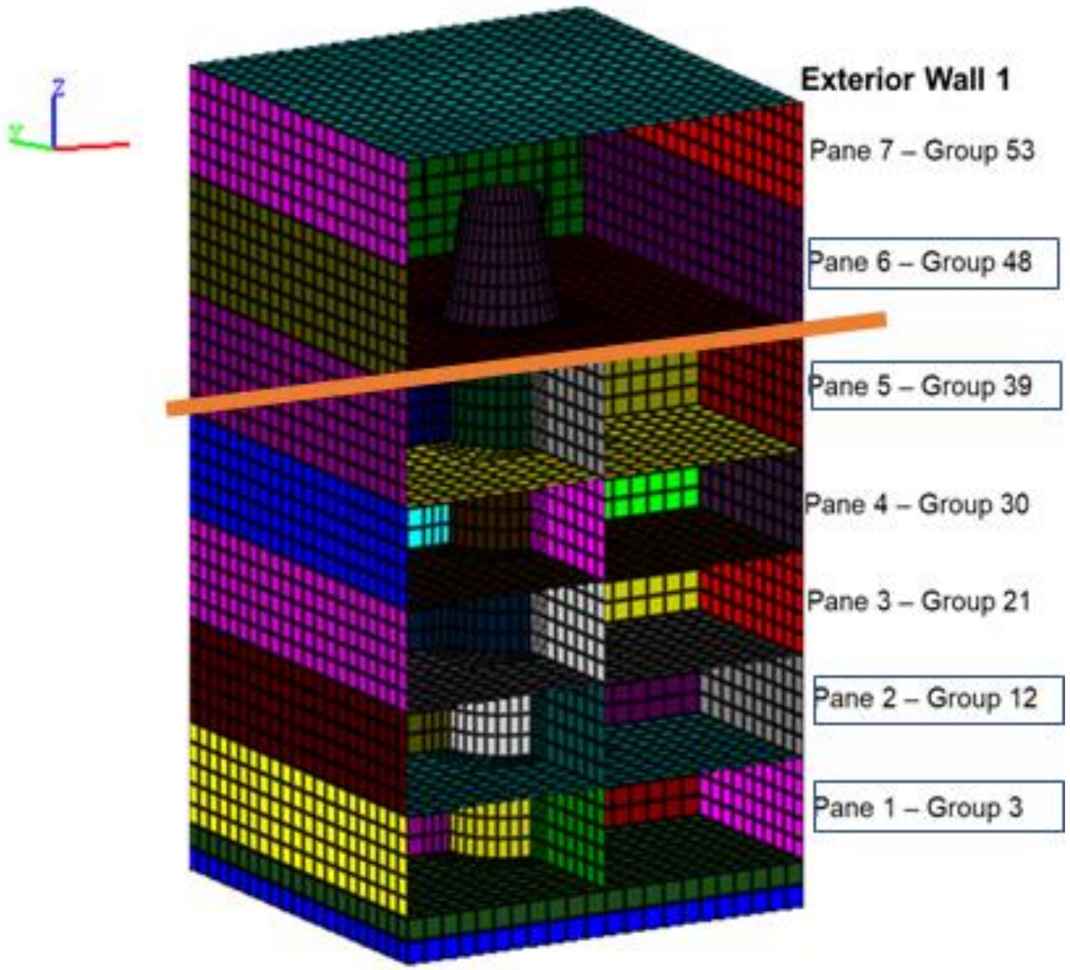
Concrete Material Properties:
Elastic Modulus = 24400000 (kN/m2)
Compression Strength = 30000 (kN/m2)
Yield Strain = 0.002
Ultimate Strain = 0.0035

Rebar Material Properties:
Young modulus = 200,000,000. (kN/m2)
Yield Strength = 345000. (kN/m2)
Yield strain = 0.001725
Ultimate strain = 0.10



Axial force N=2500 KN, with mesh_size=50mm.

SMR Wall Verification of 2D Fiber Model vs. JEAC 4601. SMR Model – V&V Problem 65 for V4.3.2



Shear Wall Reinforcement:

Flange 1 Vertical Rebar Ratio = 1.195%
Flange 2 Vertical Rebar Ratio = 1.598%
Web Vertical Rebar Ratio = 1.246%
Web Horizontal Rebar Ratio = 0.953%

Concrete Material Properties:

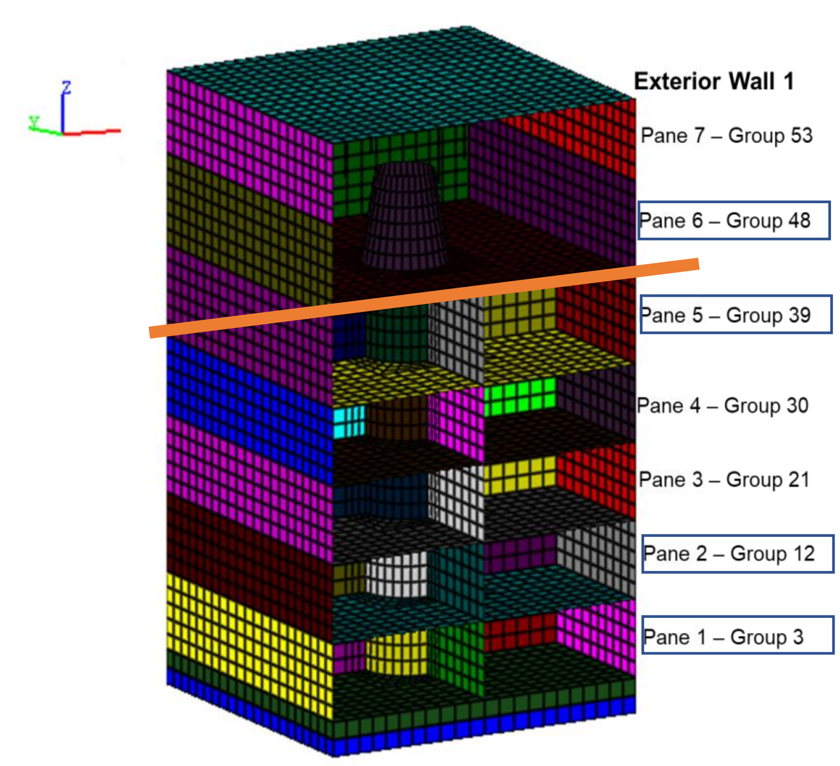
Elastic Modulus = 24854600 (Kn/m2)
Compression Strength = 33000 (Kn/m2)
Yield Strain = 0.002
Ultimate Strain = 0.004

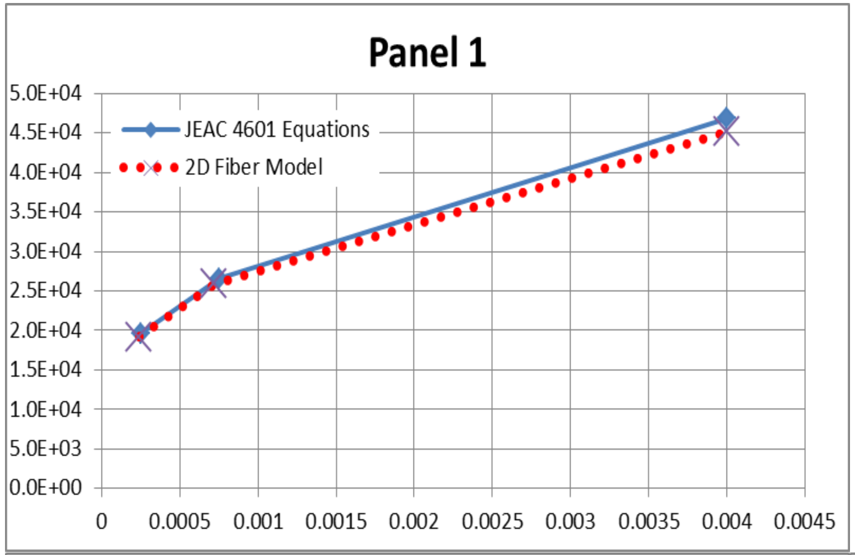
Rebar Material Properties:

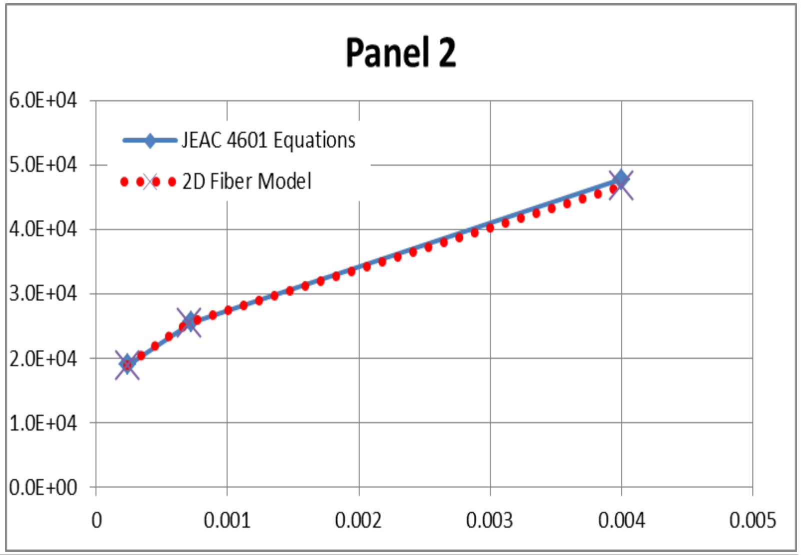
Young modulus = 205000000. (Kn/m2)
Yield Strength = 345000. (Kn/m2)
Yield strain = 0.00185
Ultimate strain = 0.05

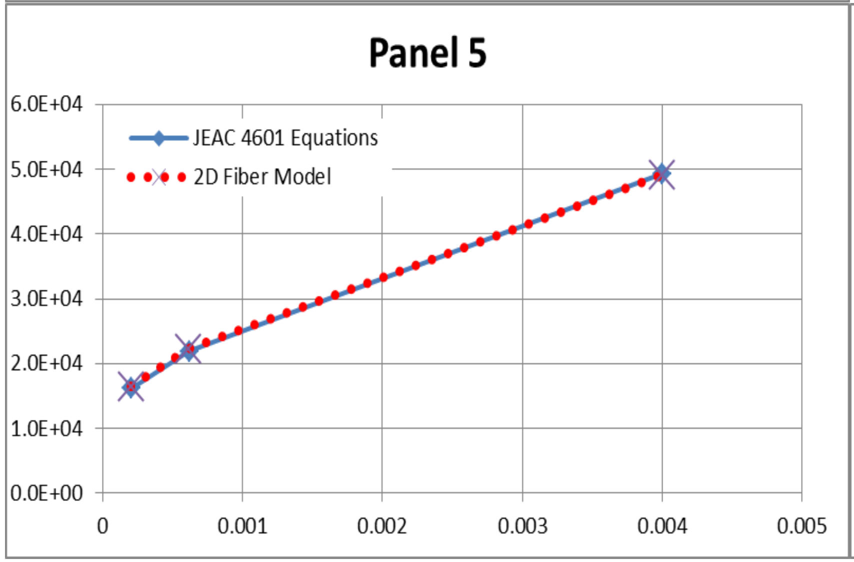
JEAC 4601 vs. 2DFiber Shear BBC (BBC_JEAC_ACI_Fiber2D.exe)

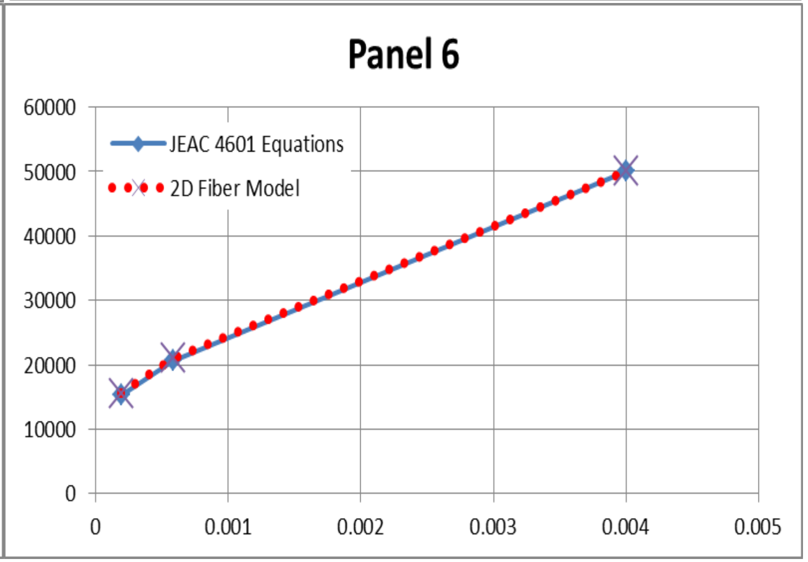
C-Shape Exterior Wall





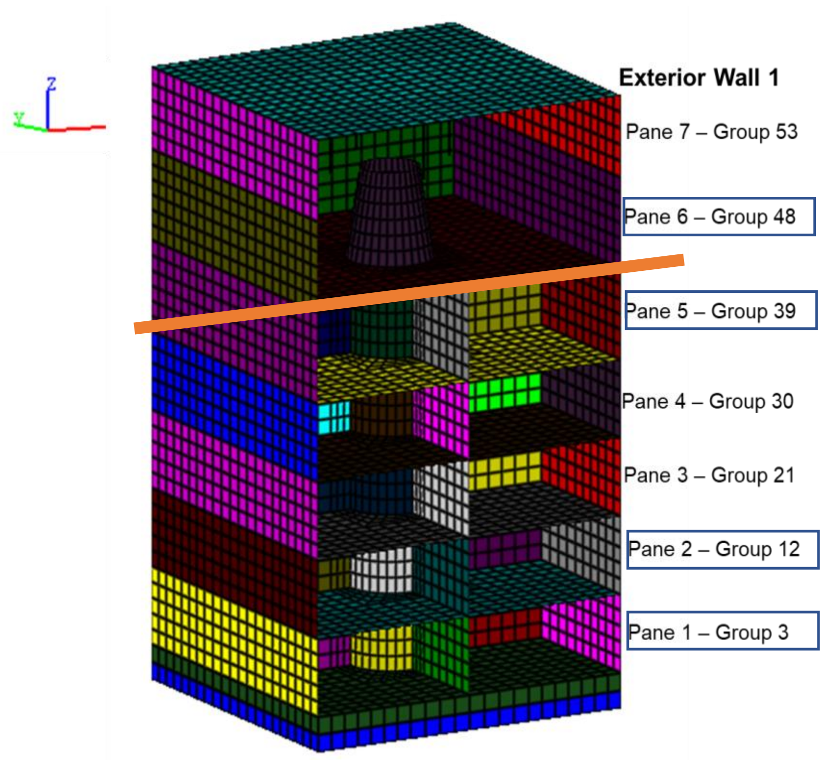


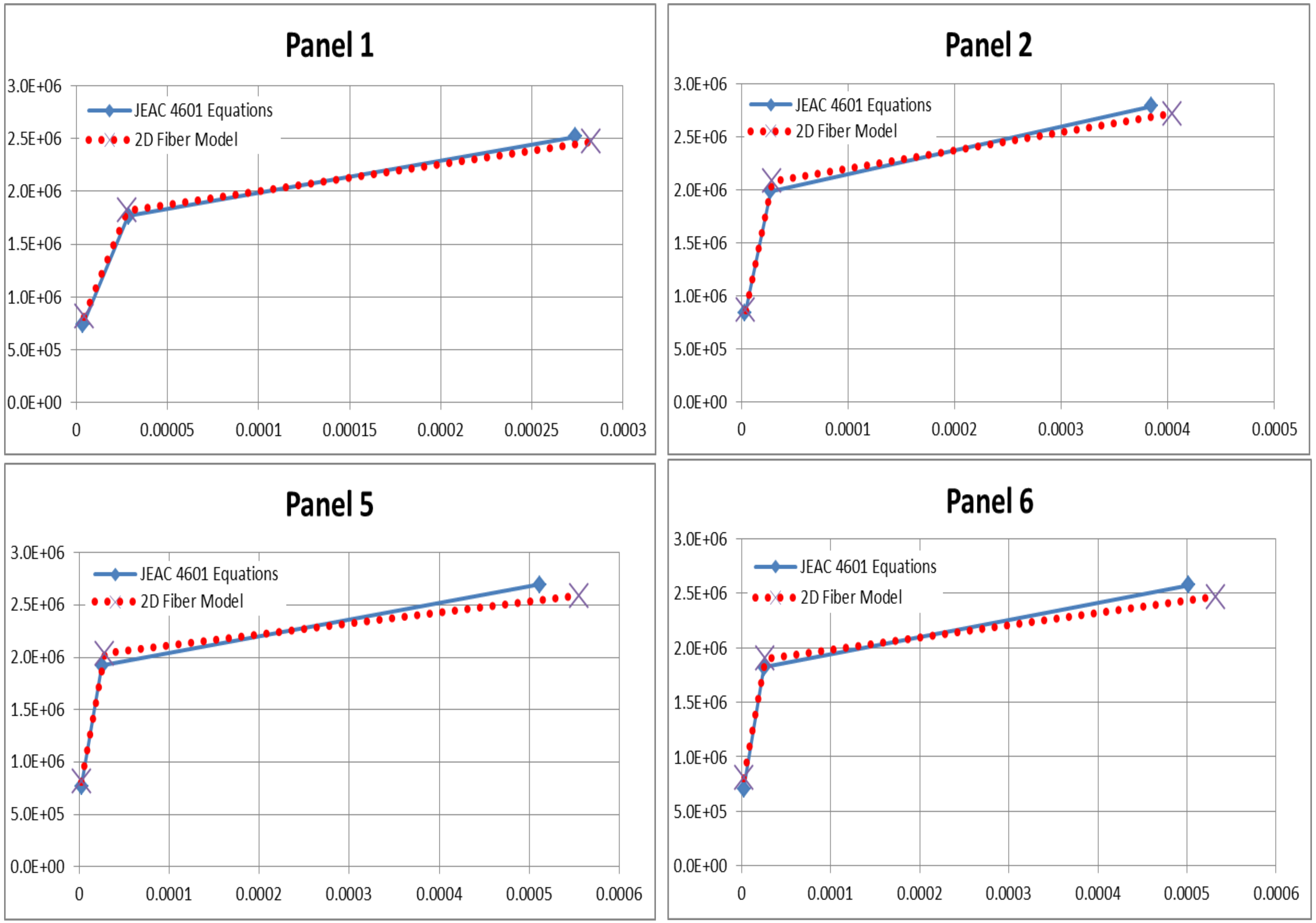




JEAC 4601 vs. 2DFiber Bending BBC (BBC_JEAC_ACI_Fiber2D.exe)

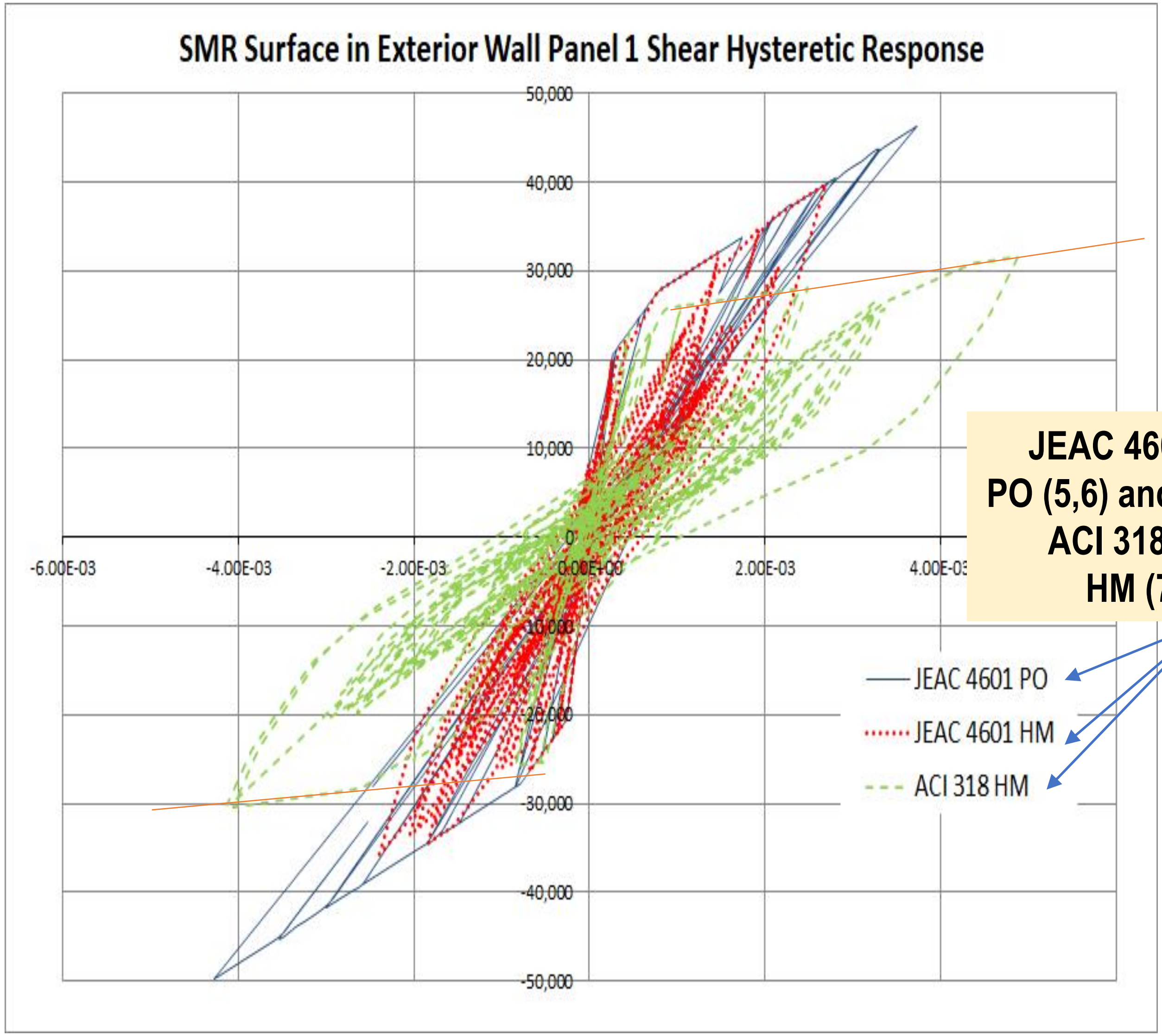
C-Shape Exterior Wall



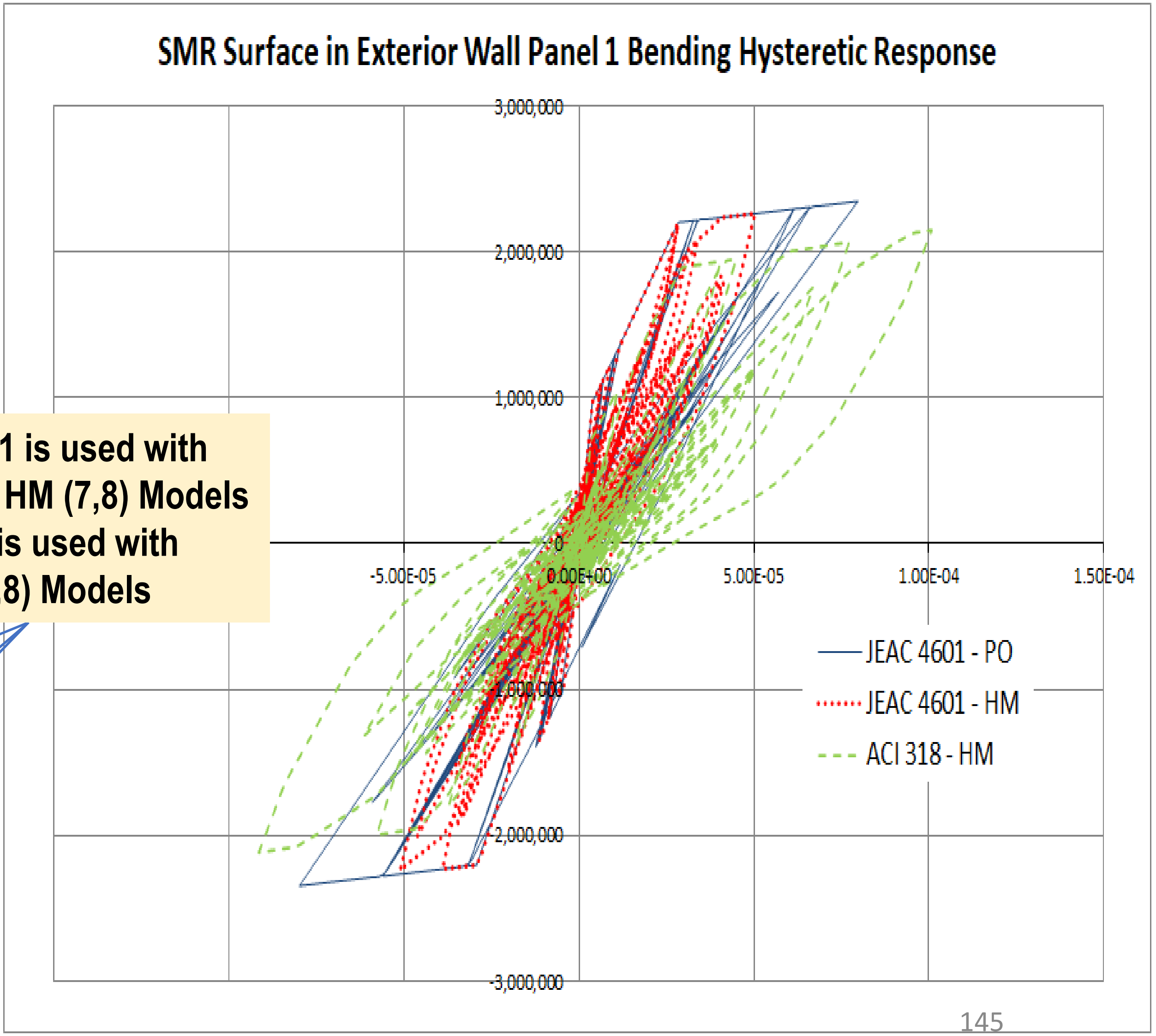


1st Floor Exterior Wall Response Using JEAC 4601 and ACI 318 Option 1

SHEAR



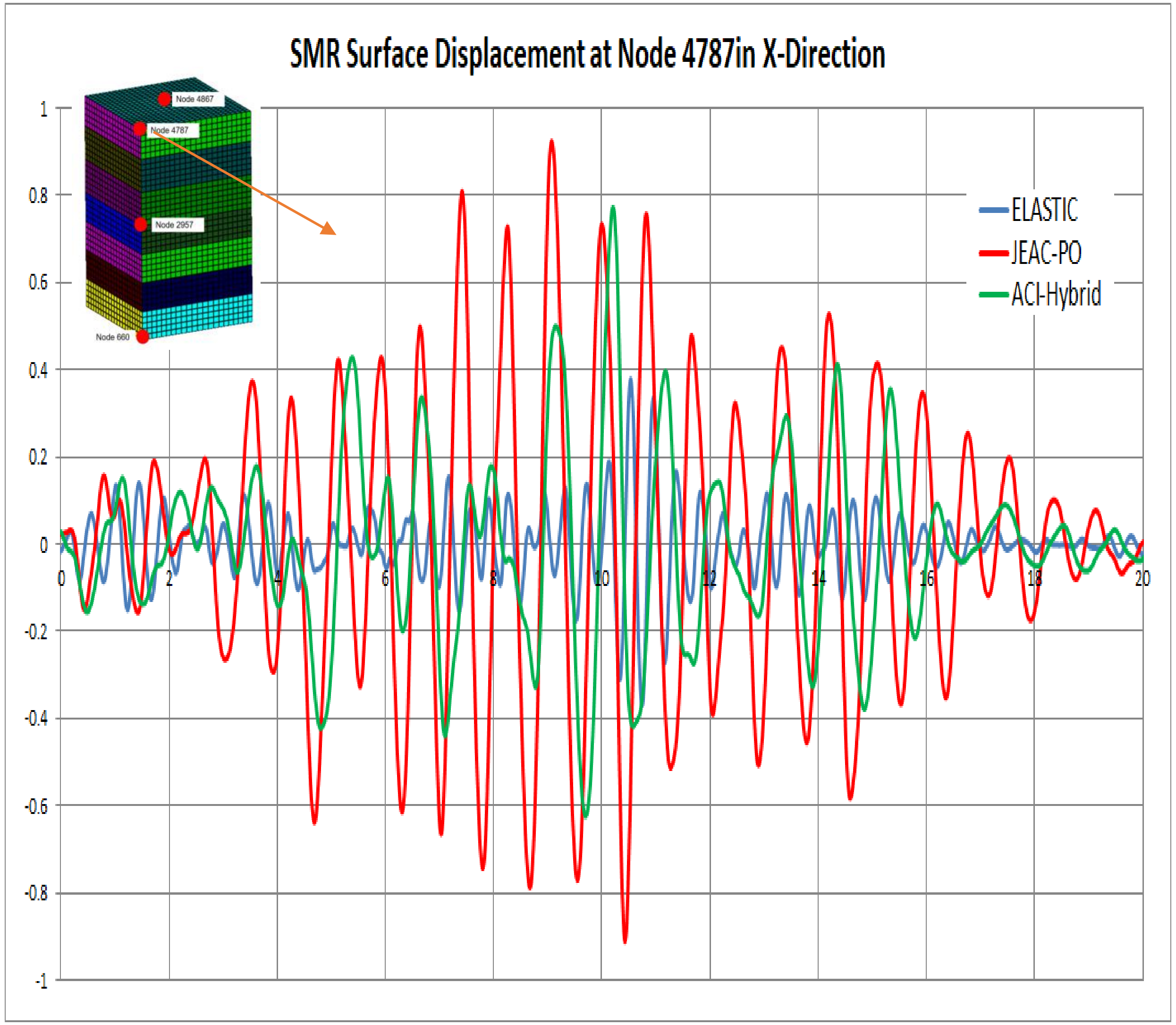
BENDING



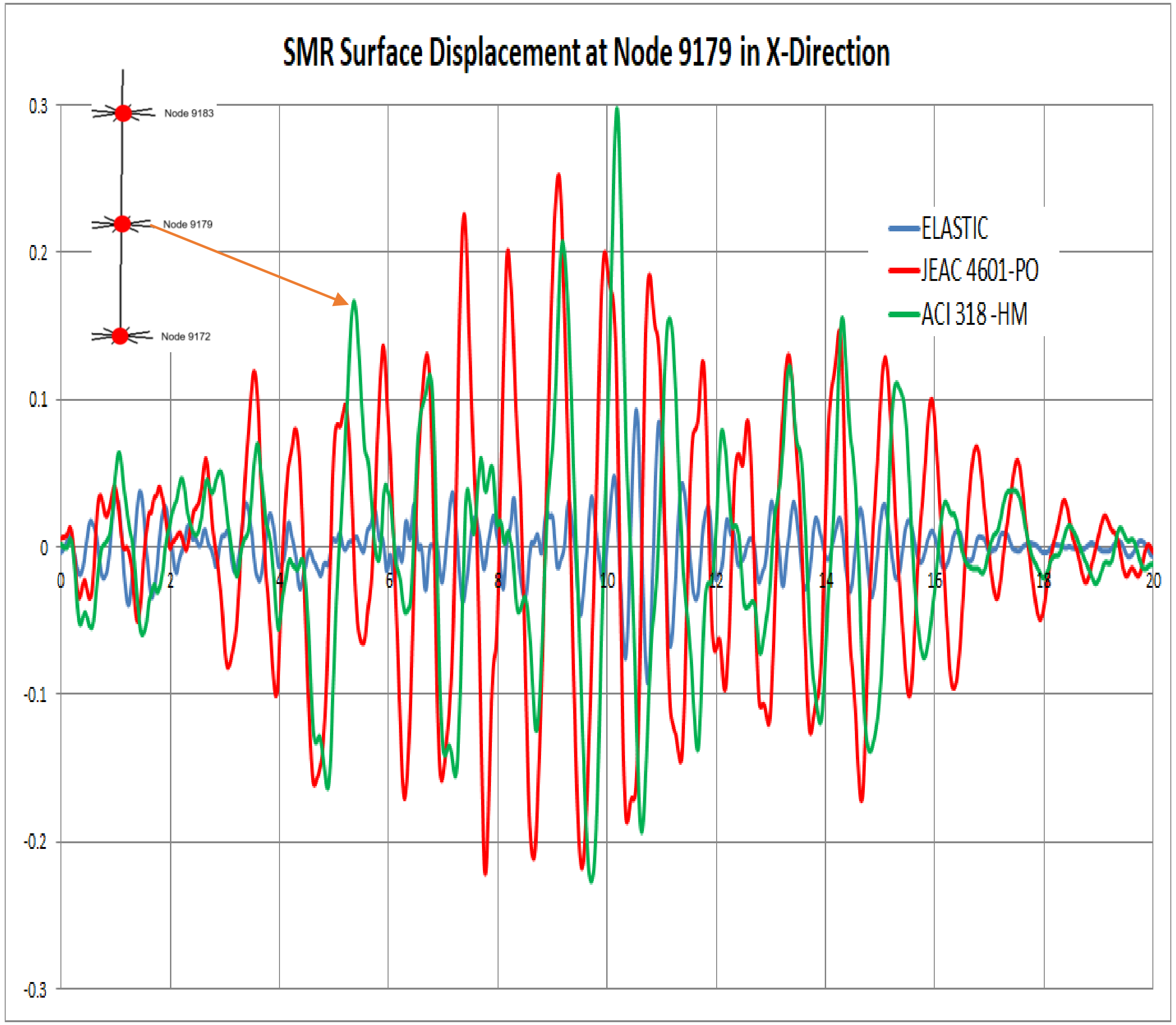
JEAC 4601 is used with PO (5,6) and HM (7,8) Models
ACI 318 is used with HM (7,8) Models

Displacements in SMR Using JEAC 4601 and ACI 318 Option 1

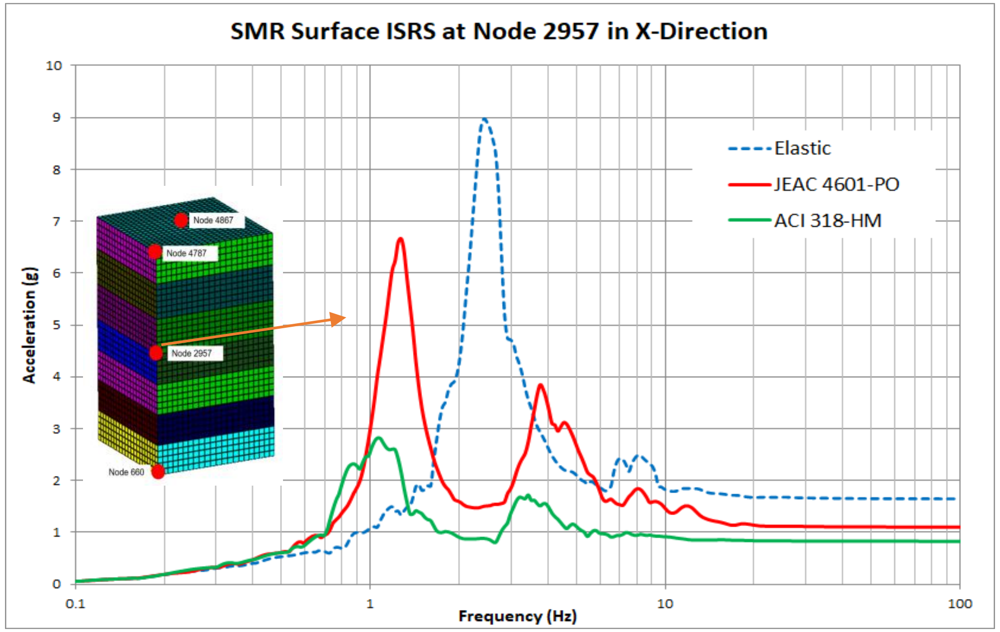
SMR Structure Top Corner

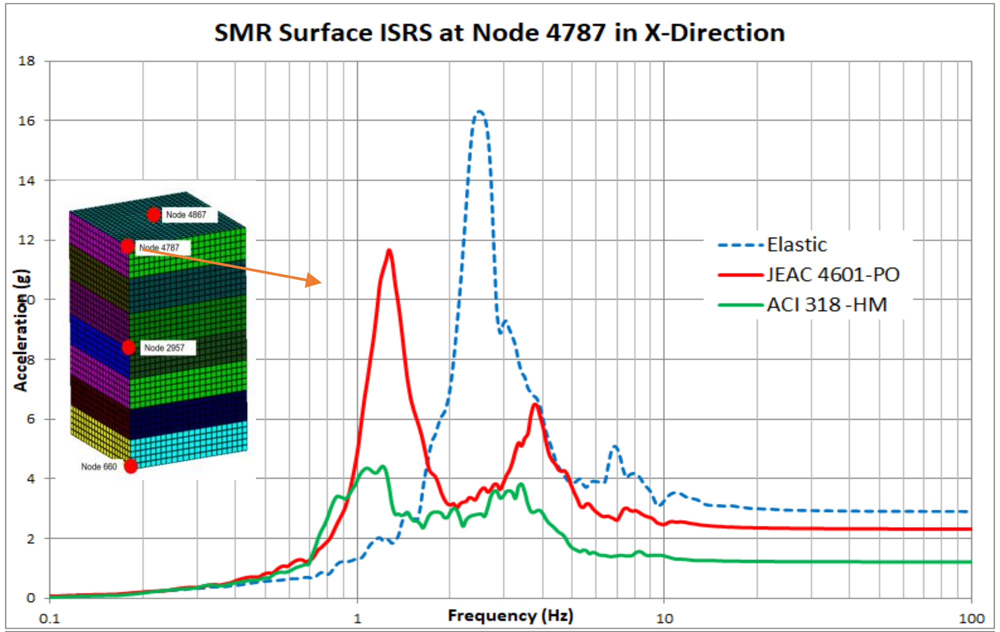


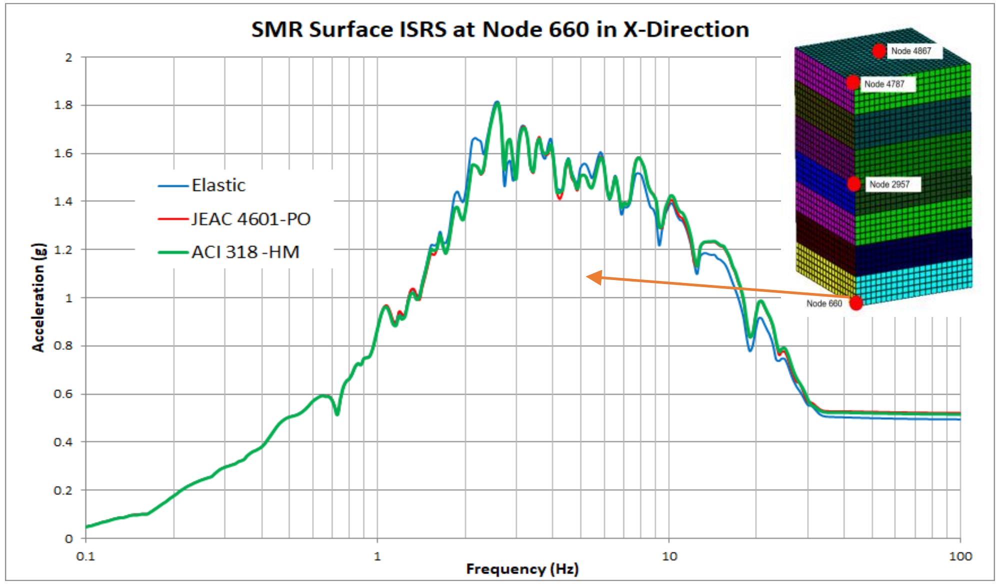
RVC Mid Elevation

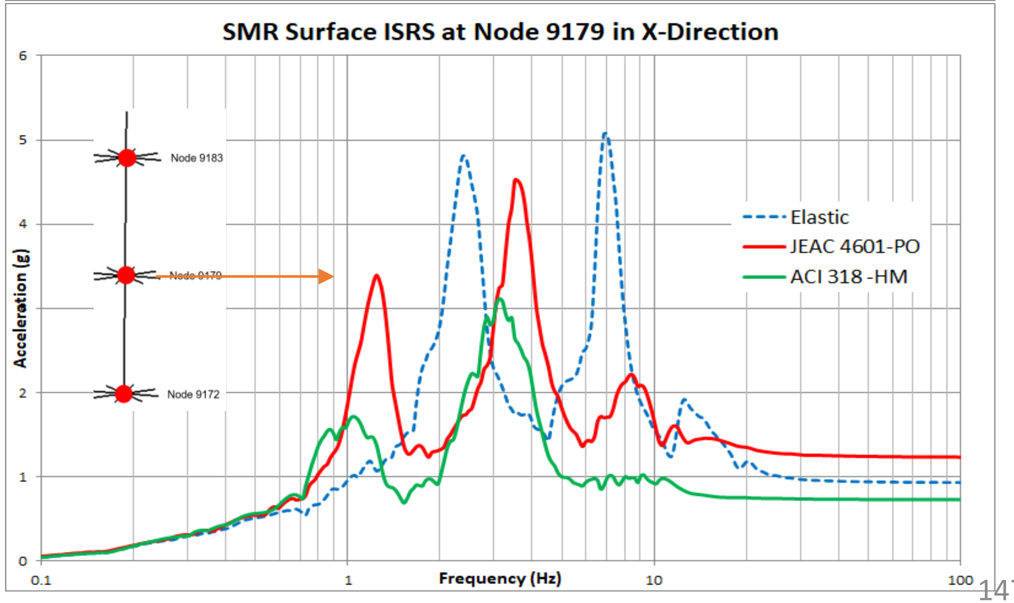


ISRS in Surface SMR Using JEAC 4601 and ACI 318 Option 1

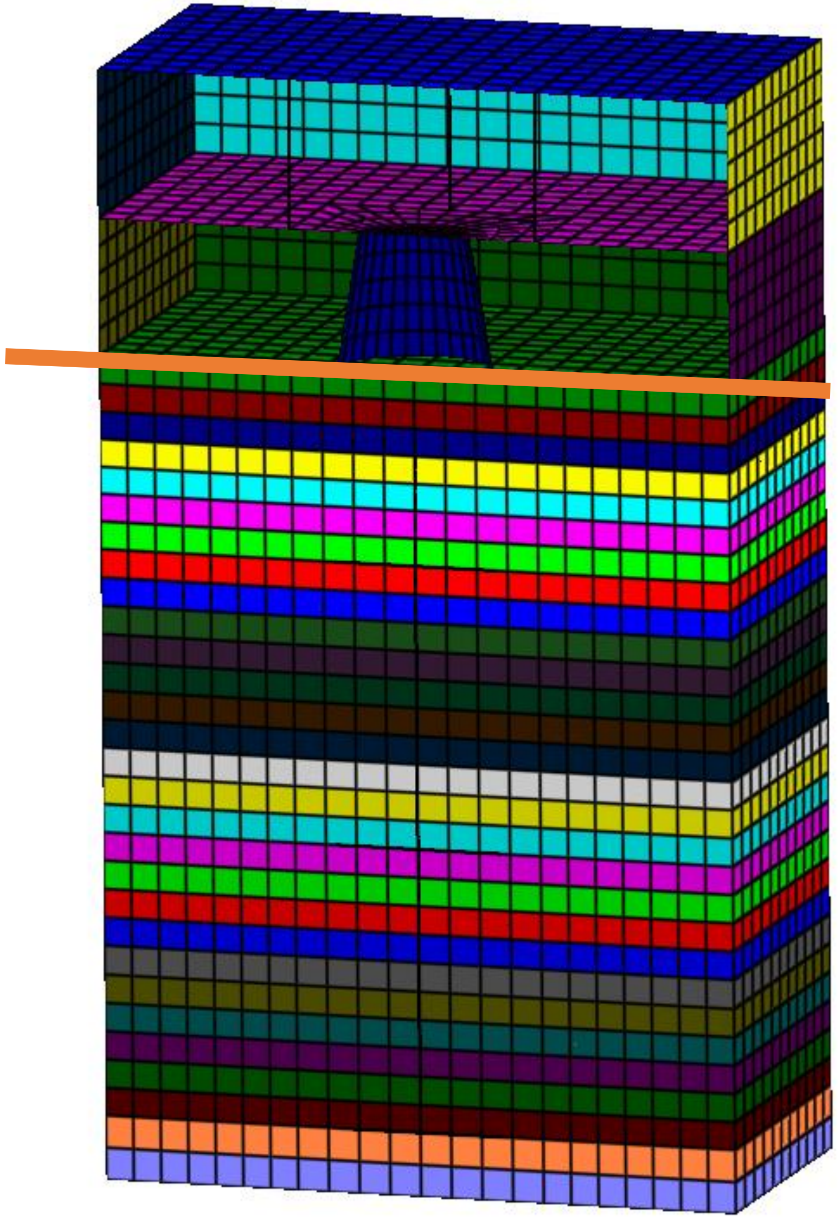


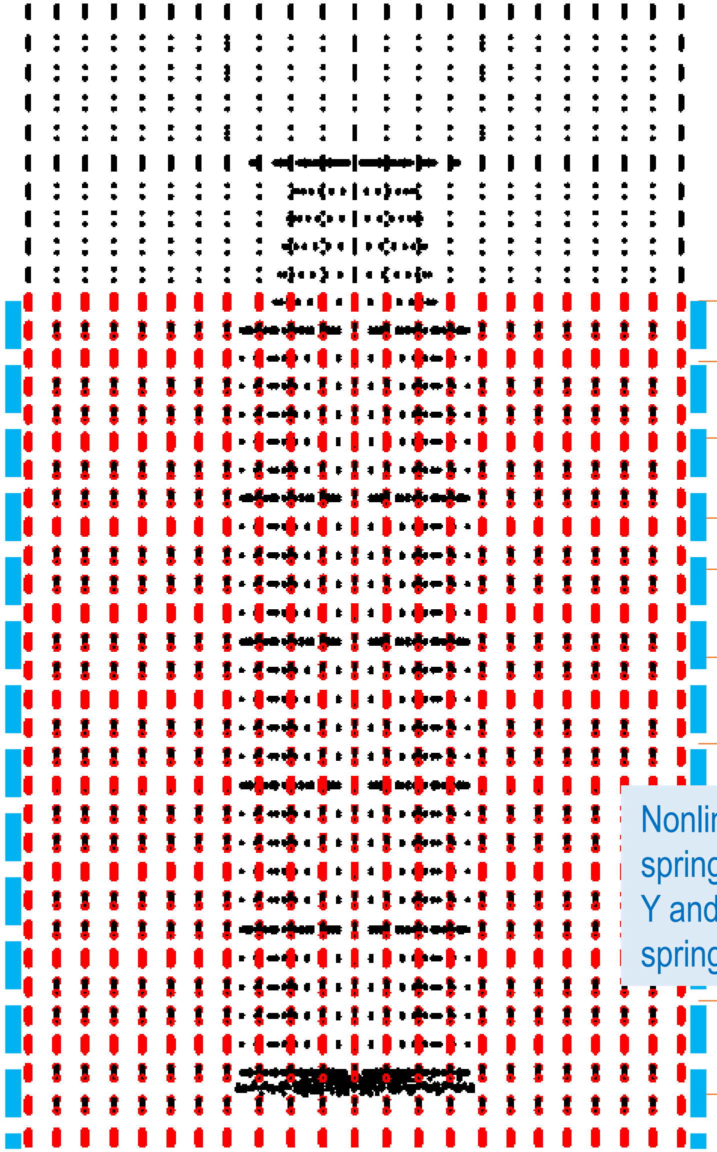






Embedded SMR with Nonlinear Springs at Soil Interface for 0.60g

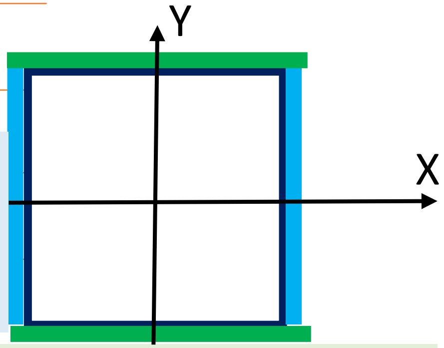




Side-soil interface modeled using nonlinear springs:

- **Nonlinear interface**

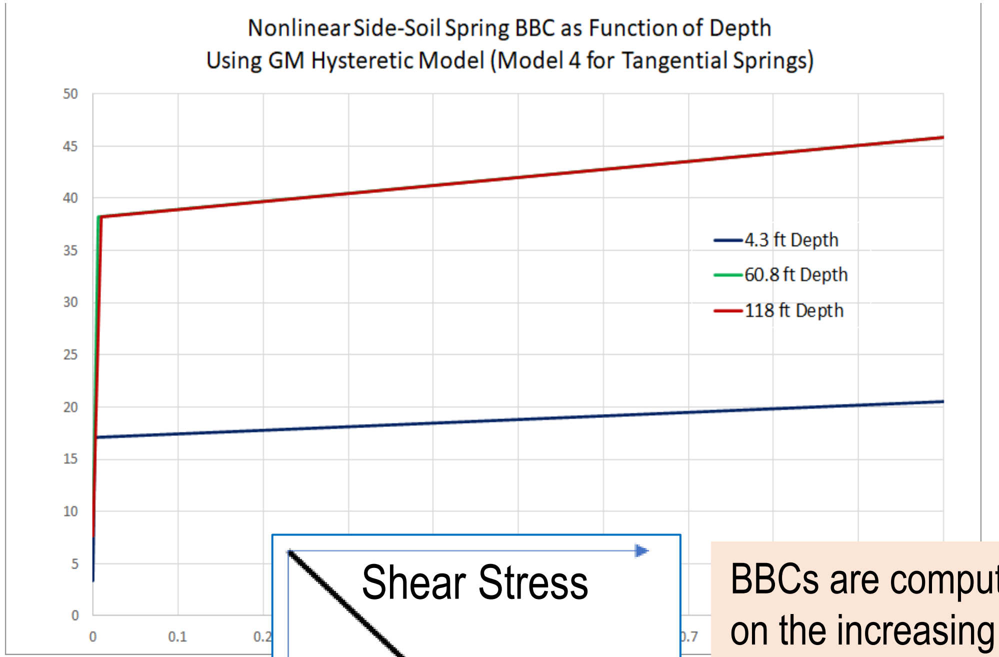
Rigid springs in normal directions X, Y and Z, and nonlinear springs in tangential directions X, Y and Z. Friction forces vary with depth.

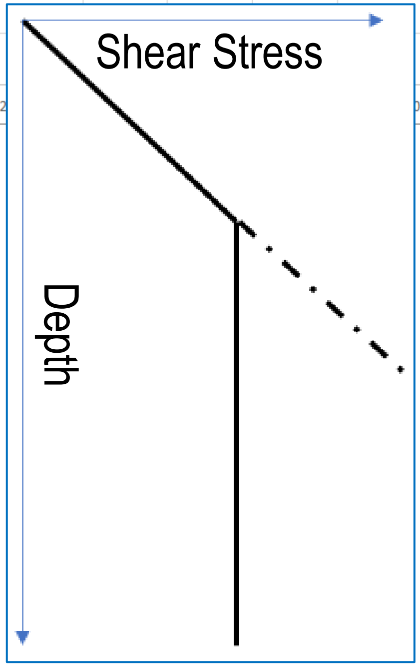


Nonlinear side-soil springs in tangential Y and Z directions springs

Nonlinear side-soil springs in tangential X and Z directions springs

Adjusted Tangential Spring BBCs for Soil Interface as Function of Depth. Applied General Massing Hysteretic Soil Model (Model 4)





BBCs are computed based on the increasing static soil pressure on the lateral wall down to layer 8 for which the shear stress is above 2ksf (API standard). Then, shear spring BBC stays the same for larger depths.

Layer Top	KX, KY and KZ	Crack displ	Crack Force	Yelding displ	Yielding Force	Ultimate Disp	Ultimate
SURFACE	1.00E+06	3.00E-06	3.00E+00	1.50E-03	15	1	18
1	1.00E+06	3.42E-06	3.42E+00	1.71E-03	17.07628	1	20.49153
2	1.00E+06	4.02E-06	4.02E+00	2.01E-03	20.09801	1	24.11761
3	1.00E+06	4.62E-06	4.62E+00	2.31E-03	23.11973	1	27.74368
4	1.00E+06	5.23E-06	5.23E+00	2.61E-03	26.14146	1	31.36976
5	1.00E+06	5.82E-06	5.82E+00	2.91E-03	29.09292	1	34.9115
6	1.00E+06	6.42E-06	6.42E+00	3.21E-03	32.11465	1	38.53758
7	1.00E+06	7.03E-06	7.03E+00	3.51E-03	35.13638	1	42.16365
8	1.00E+06	7.65E-06	7.65E+00	3.82E-03	38.22838	1	45.87405
9	1.00E+06	7.65E-06	7.65E+00	4.13E-03	38.22838	1	45.87405
10	1.00E+06	7.65E-06	7.65E+00	4.44E-03	38.22838	1	45.87405
11	1.00E+06	7.65E-06	7.65E+00	4.75E-03	38.22838	1	45.87405
12	1.00E+06	7.65E-06	7.65E+00	5.06E-03	38.22838	1	45.87405
	E+06	7.65E-06	7.65E+00	5.37E-03	38.22838	1	45.87405
	E+06	7.65E-06	7.65E+00	5.68E-03	38.22838	1	45.87405
	E+06	7.65E-06	7.65E+00	5.99E-03	38.22838	1	45.87405
	E+06	7.65E-06	7.65E+00	6.30E-03	38.22838	1	45.87405
	E+06	7.65E-06	7.65E+00	6.61E-03	38.22838	1	45.87405
	E+06	7.65E-06	7.65E+00	6.91E-03	38.22838	1	45.87405
	E+06	7.65E-06	7.65E+00	7.22E-03	38.22838	1	45.87405
	E+06	7.65E-06	7.65E+00	7.53E-03	38.22838	1	45.87405
	E+06	7.65E-06	7.65E+00	7.84E-03	38.22838	1	45.87405
	E+06	7.65E-06	7.65E+00	8.16E-03	38.22838	1	45.87405
	E+06	7.65E-06	7.65E+00	8.47E-03	38.22838	1	45.87405
	E+06	7.65E-06	7.65E+00	8.78E-03	38.22838	1	45.87405
25	1.00E+06	7.65E-06	7.65E+00	9.09E-03	38.22838	1	45.87405
26	1.00E+06	7.65E-06	7.65E+00	9.40E-03	38.22838	1	45.87405
27	1.00E+06	7.65E-06	7.65E+00	9.70E-03	38.22838	1	45.87405
28	1.00E+06	7.65E-06	7.65E+00	1.00E-02	38.22838	1	45.87405
29	1.00E+06	7.65E-06	7.65E+00	1.04E-02	38.22838	1	45.87405

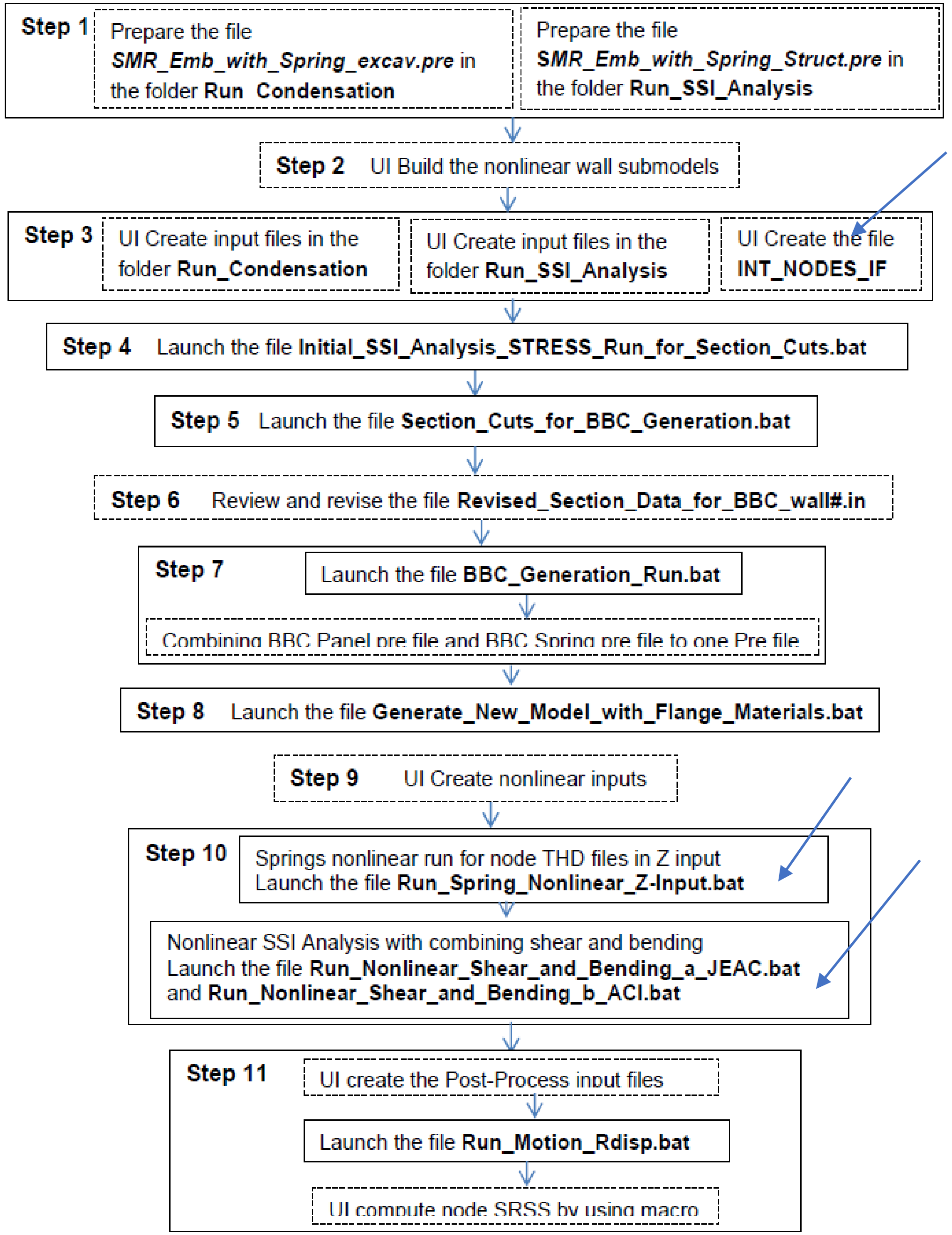
Nonlinear Seismic SMR SSI Analysis Steps Using Option NON Advanced:

- **The Section_Cuts_for_BBC module –** This module performs automatic wall section geometry identification and the section-cuts for the selected RC wall sections.

- **The BBC_JEAC_ACI_Fiber2D module –** This module computes the shear and bending backbone curve (BBC) for all the RC wall sections based on the either US standard or Japan standard recommendations for walls with planar section shapes (as I, C, T, or rectangular shapes), or based a 2D fiber model for walls with non-planar section shapes (as circular or rombic closed-section shapes with double symmetry).

- **The Create_Flange_Materials module –** This creates and assigns new flange materials for each nonlinear wall that are included in a new structure model .pre file named ModelName_NEW.pre file (appends the "_NEW" strings to the initial .pre file name used for elastic analysis).

- **COMB_Shear_Bend module –** This module combines the shear and bending interactive effects in all nonlinear RC walls at each nonlinear SSI iteration.

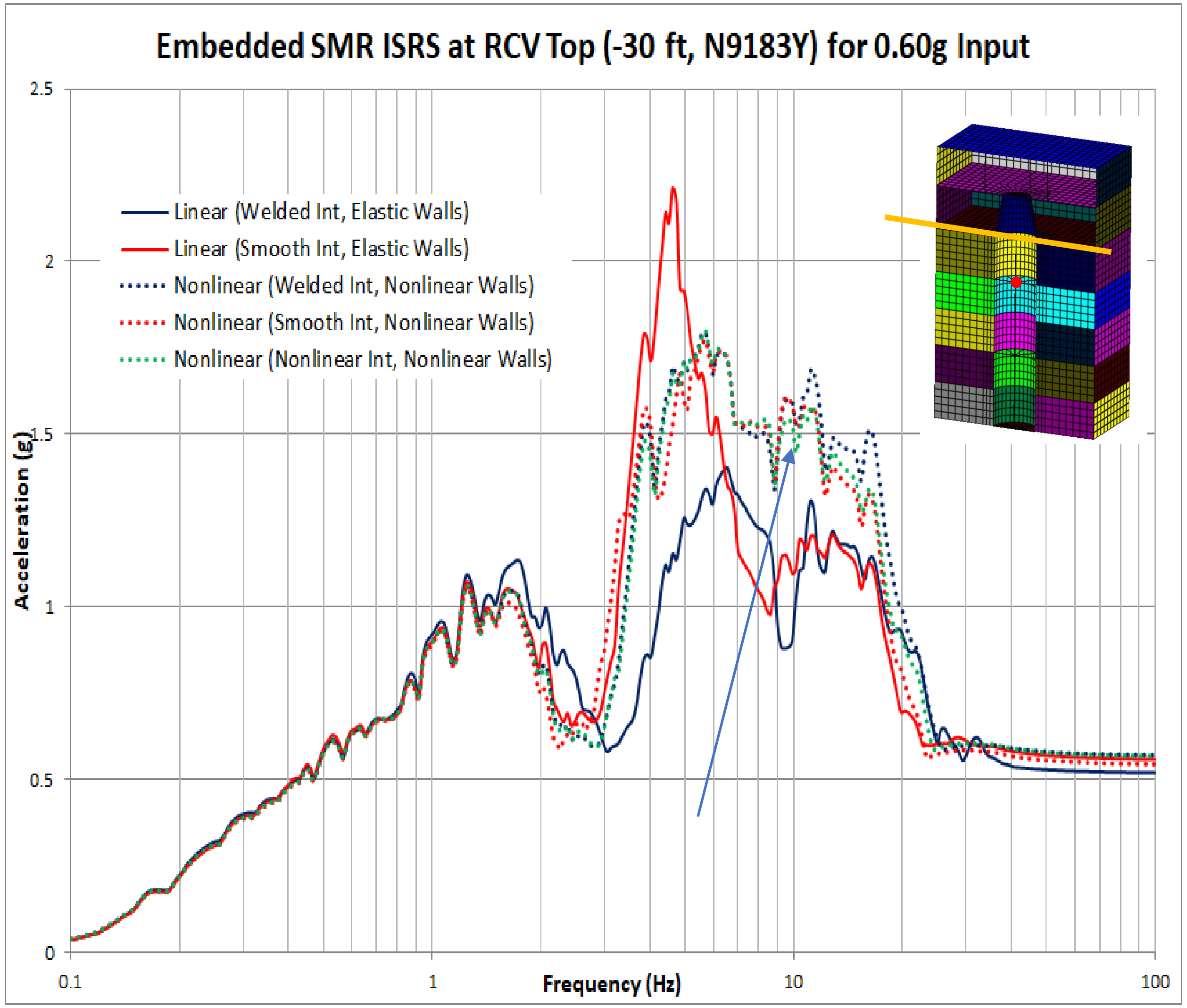


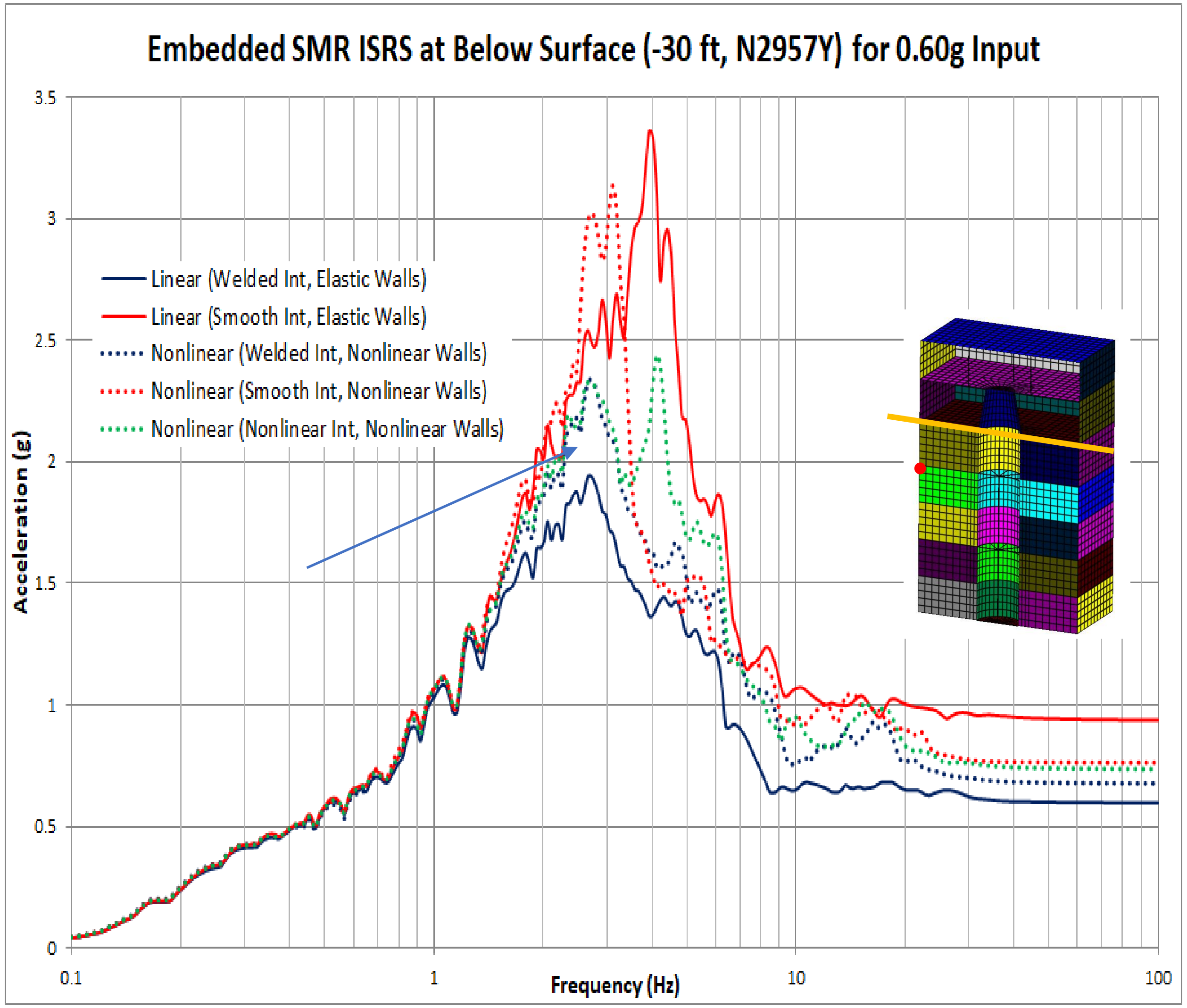
SMR ATF Below Surface – Linear vs. Nonlinear Soil Interface

-30ft Depth

Linear/Nonlinear Tangential Springs
Linear/Nonlinear Walls

-30ft Depth



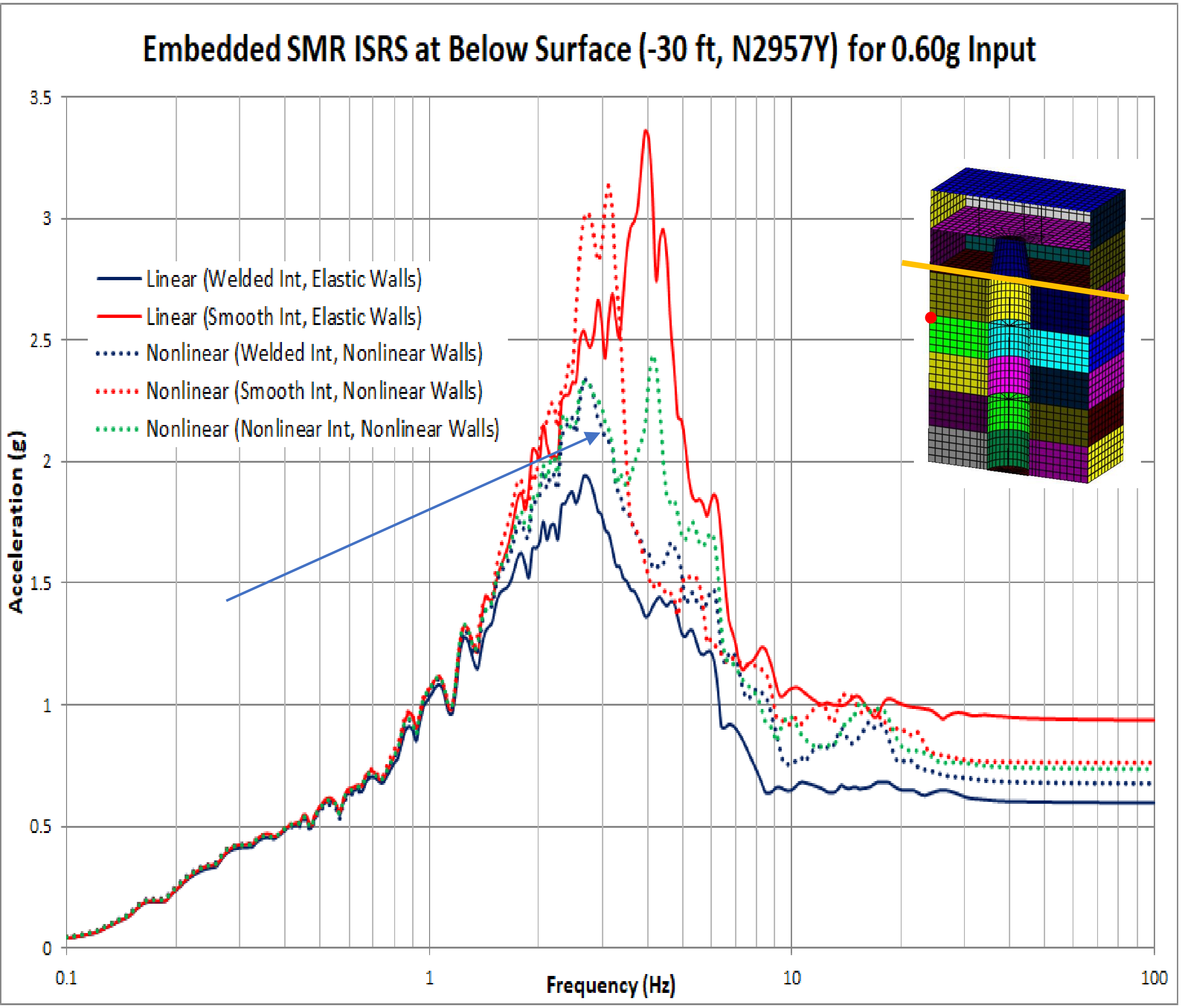


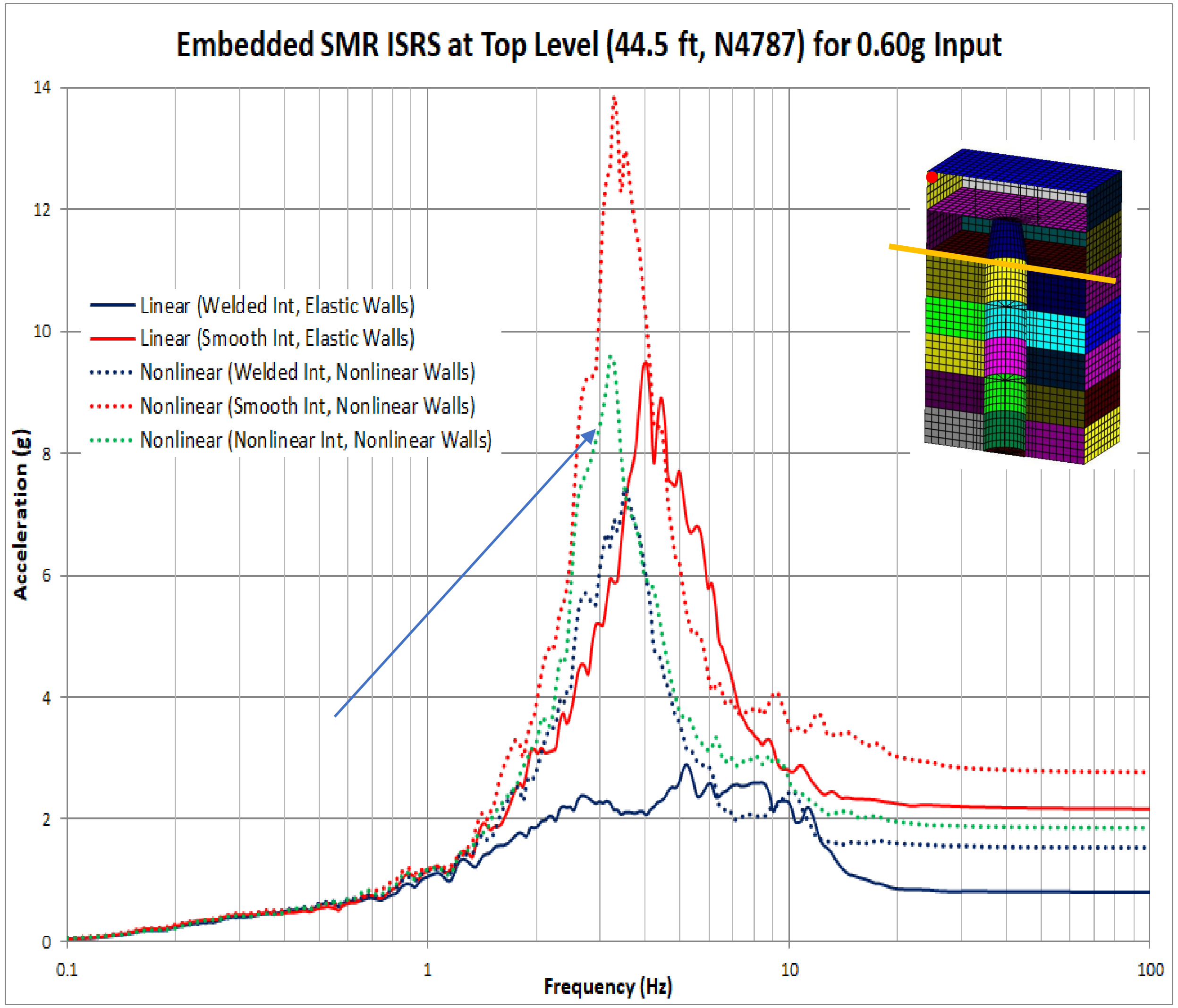
SMR ISRS Below Surface – Linear vs. Nonlinear Interface for 0.60g

-30ft Depth

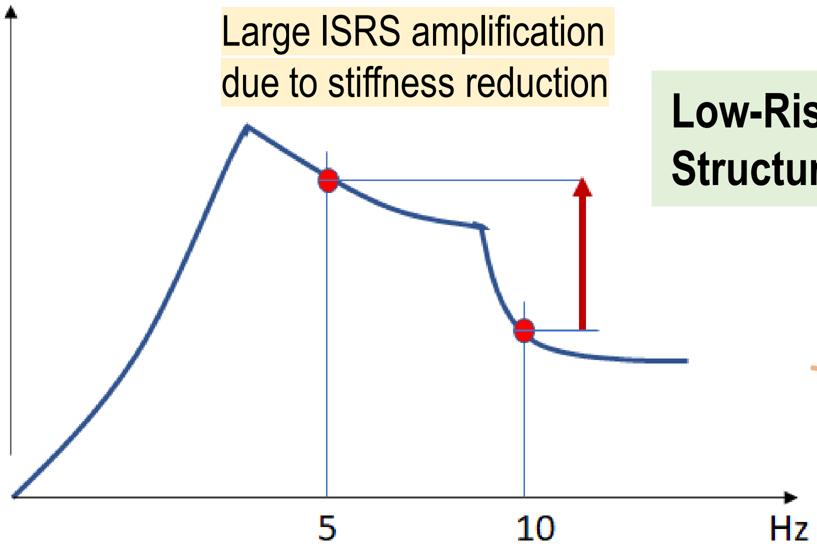
Linear/Nonlinear Tangential Springs
Linear/Nonlinear Walls ACI 318

44.5ft Above Ground

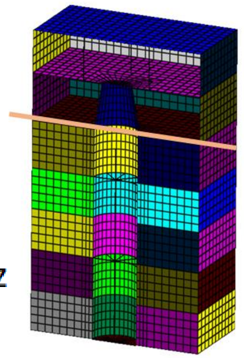


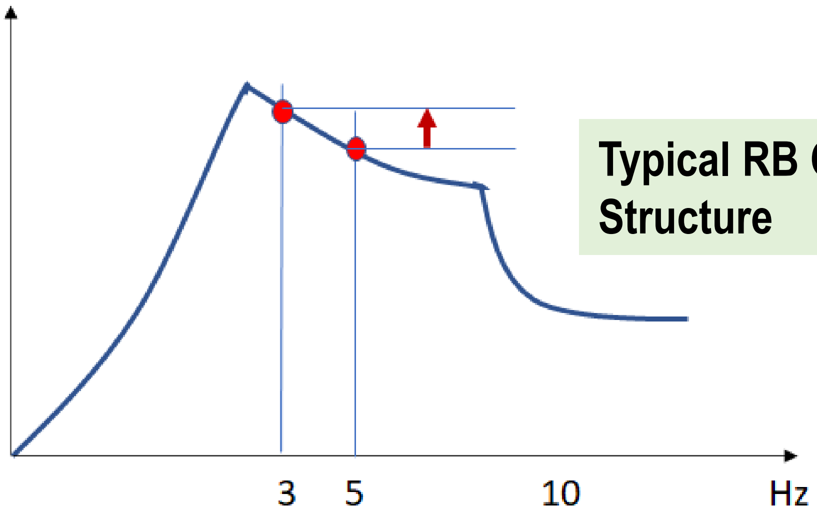


Low-Rise SMR Super Structure Frequency is Shifted to Lower Frequencies due to Nonlinear Wall & Interface Stiffness Reduction

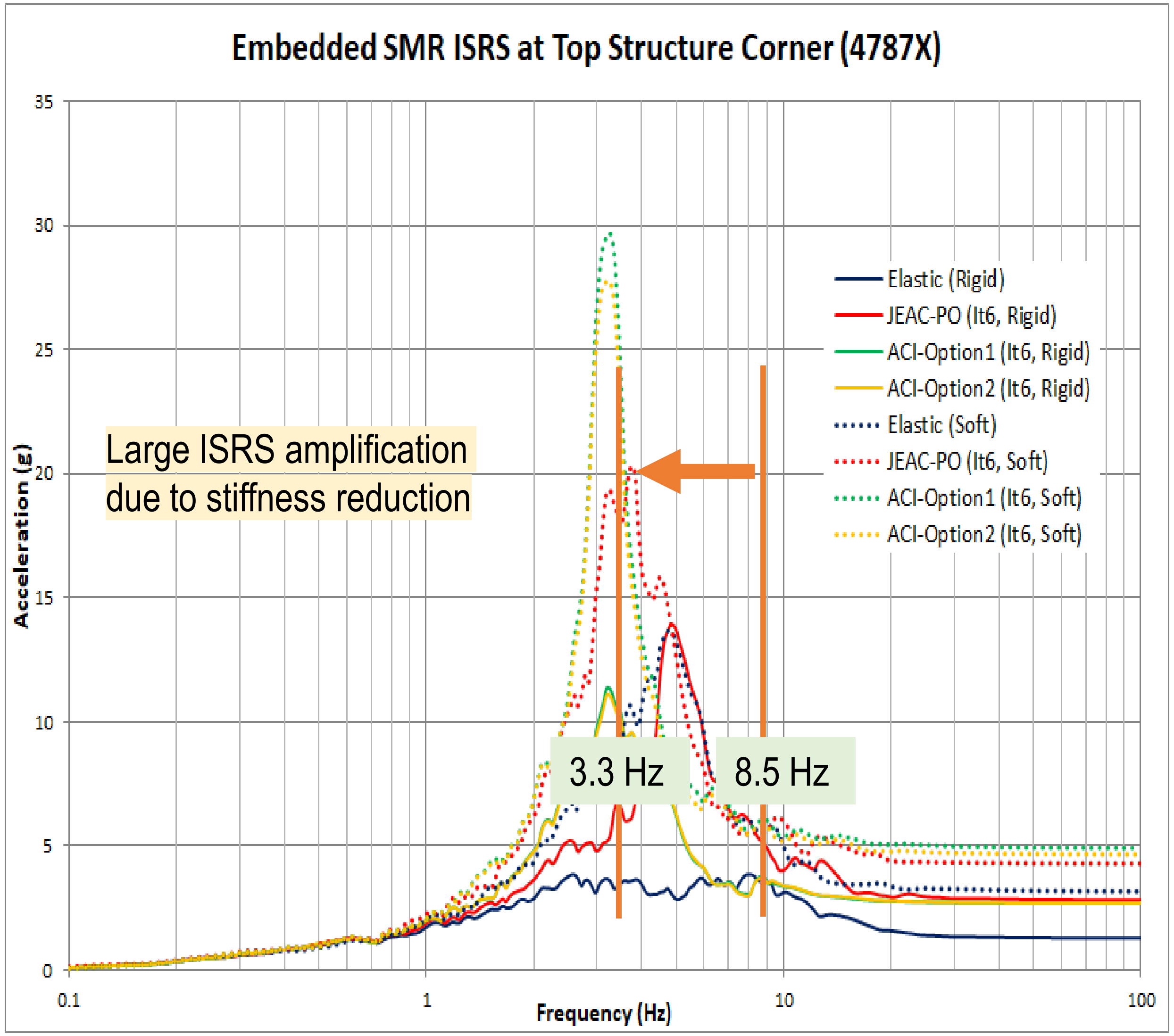


Low-Rise SMR Structure

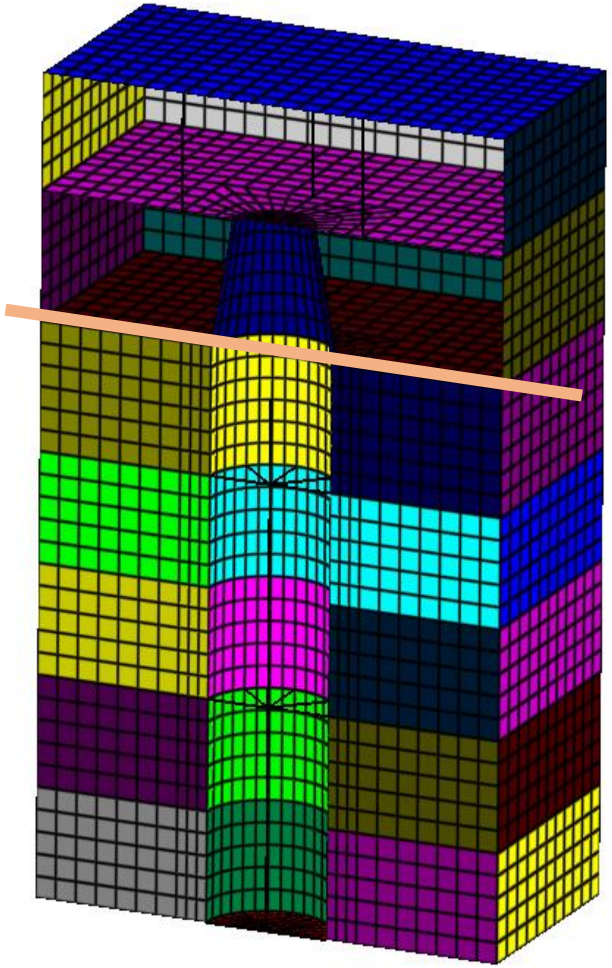




Typical RB Complex Structure



Embedded SMR Linear/Nonlinear SSI Analysis Using FVROM-INT (20 SOIL key frequencies and 200 SSI frequencies) – new test



Using 4 parallel runs on 128 GB RAM workstations

	Running Time		Node Counts		# of Freq.
	Total (h)	Per Freq (h)	Int. Nodes	Cond. Nodes	
Condensation	2.79	0.11	7491	3081	20
Interpolation			-	3081	200
SSI Solution	2.77	0.01	-	3081	200
	5.6 hours				

Speed up ratio = 3.50

	Running Time		Node Counts		# of Freq.
	Total (h)	Per Freq (h)	Int. Nodes	Cond. Nodes	
Direct SSI	19	0.097222	7491	-	200
	19.4 hours				
	Speed ratio =		3.497354		

1 minute/freq per one nonlinear walls-nonlinear interface iteration.
3 iterations are less than 10% difference in elements, 4-5 ideal

End of Part 4 Presentation
Thank You!