Hybrid Simulation 101 Short Course
NHERI Lehigh Experimental Facility
Lehigh University
June 24, 2025

Hybrid Simulation 101: Short course on the Theory, Implementation, and Application of Real-time Hybrid Simulation

James Ricles, Ph.D., P.E.
Bruce G. Johnston Professor
Director, NHERI Lehigh Real-time Multi-Directional Hybrid Simulation Facility
Director, ATLSS Engineering Research Center

Faisal Nissar Malik
PhD Graduate Research Assistant,
NHERI Lehigh Real-time Multi-Directional Hybrid Simulation Facility

Lehigh University

1

Session 3: Implementation and Execution

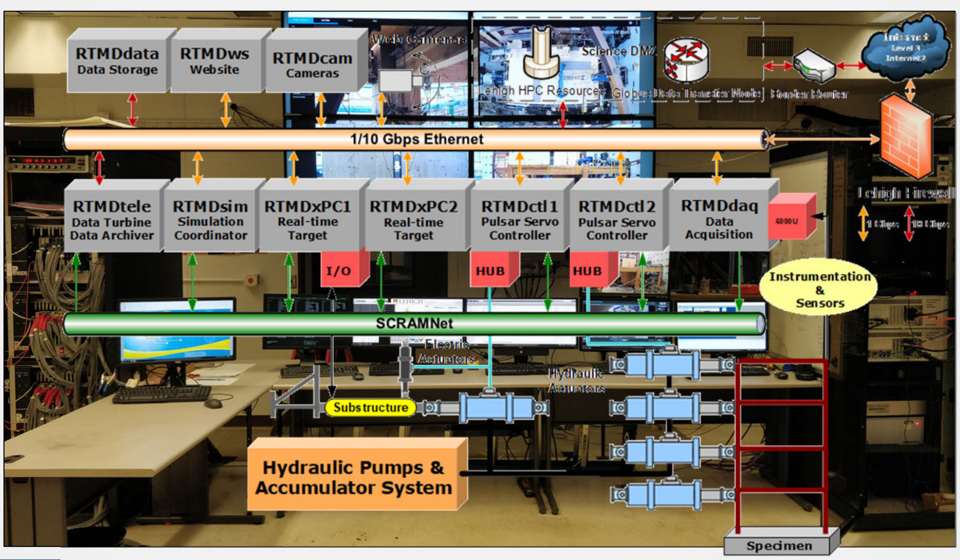
Session 4: Hands-on RTHS Group Assignment

Session 5: Groups Perform RTHS

Session 6: Groups Presentations

Session 3: Implementation and Execution

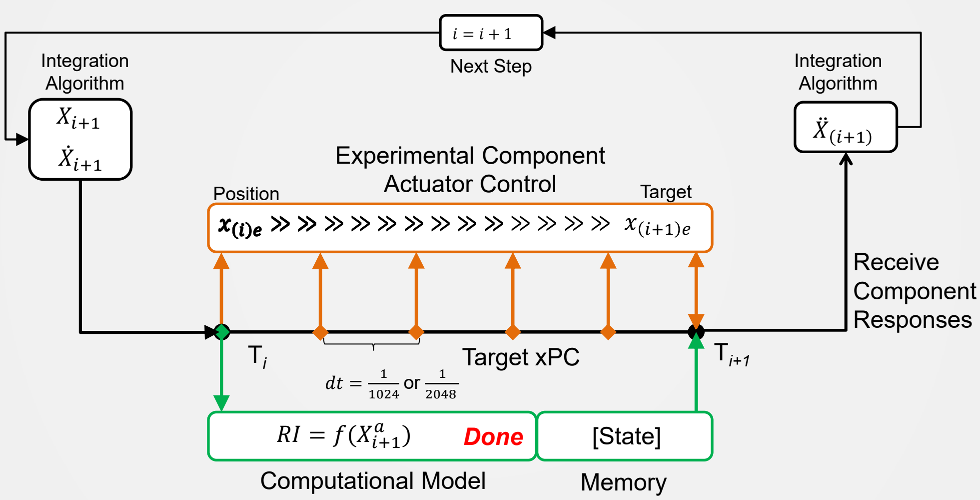
NHERI Lehigh EF Real-time IT Infrastructure



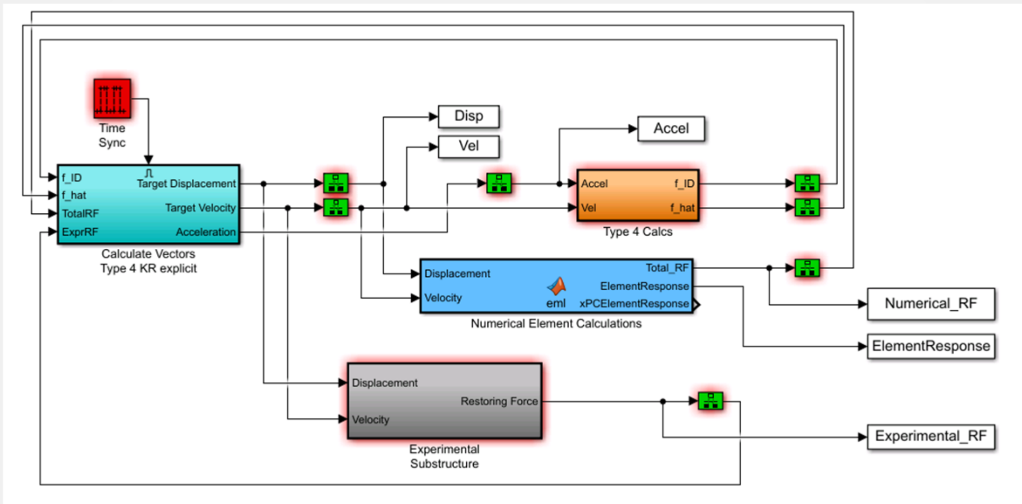
Communication Protocol: SCRAMNet

- Target xPC, DAQ, and Controller equipped with SCRAMNet GT 200 Cards
- SCRAMNet: Replicated shared memory
  - Fiber-optic based
    High-Speed, Low-Latency Industrial Protocol for
    Real-time data control
- Communication delay of less than 100 nano seconds

RTHS: Model Flow through Simulink Simulation

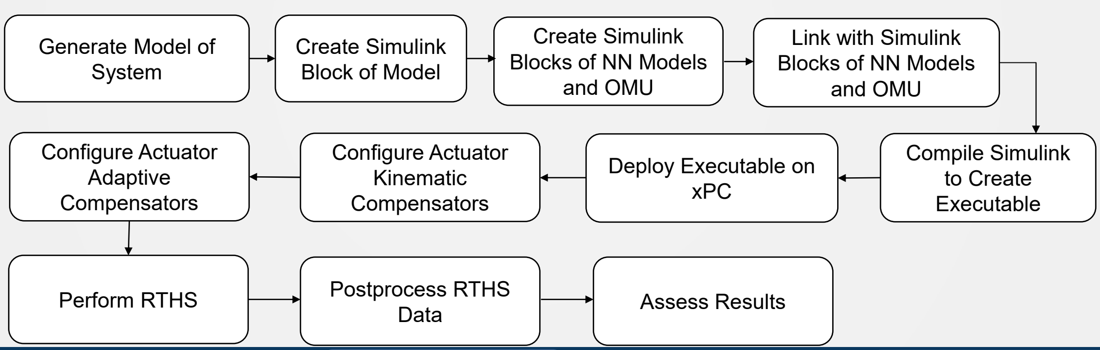


RTHS: Simulink Block Diagram



Steps for Conducting a RTHS

1) Generate model of system using HyCom-3D
2) Create Simulink block of system model
3) Create Simulink blocks of neural network models and on-line model updating models
4) Link Simulink blocks
5) Compile Simulink linked blocks to create executable
6) Deploy executable on xPC with a real-time operating system
7) Configure actuator kinematic compensators
8) Configure actuator adaptive compensators
9) Conduct RTHS
10) Postprocess RTHS Data
11) Assess results



HyCOM-3D: 3-D Real-time Computational Modeling

- MATLAB and Simulink based 3D computational modeling and simulation coordinator software for dynamic time history analysis of inelastic-framed structures and performing real-time hybrid simulation
- Simulink architecture facilitates real-time testing through multi-rate processing
- Run Modes
  - ❑ MATLAB script for numerical simulation
  - ❑ Simulink modeling for Real-Time Hybrid simulation with experimental elements via Real-Time Targets, and hydraulics-off for training and validation of user algorithms.
- User's Manual for training

HyCom-3D: 3-D Real-time Computational Modeling

Configuration Options:

- Three-dimensional analysis
- Coordinate system of nodes
- Boundary, constraint and restraint conditions
- **3-D Explicit-formulated Elements**
  - Elastic beam-column
  - Elastic spring
  - Inelastic beam-column stress resultant element
  - Non-linear spring
  - NL Displacement-based beam-column fiber element
  - NL Force-based beam column fiber element
  - Zero-length
  - NL planar panel zone
  - Elastic beam-column element with geometric stiffness
  - User-defined Reduced Order Modeling elements
  - **Co-Rotational elastic, NL force and displacement-based fiber elements**
  - Gap elements
  - Super elements
  - Multi-axis experimental elements
- Geometric imperfections and Geometric nonlinearities
- Steel wide flange and HSS sections (link to AISC Database), General sections and reinforced concrete sections
- Structural mass: lumped and consistent element mass formulations
- Inherent damping: proportional, non-proportional, modal
- Adaptable dissipative, explicit-based integration methods
- Real-time online model updating
- Neural Network Modeling
- Nonlinear static analysis (load or displacement control; Riks Method)
- Transient multi-directional multi-natural hazard dynamic analysis: Implicit Generalized α, HHT-α, Newmark's CAA Method, Explicit KR- α, Explicit MKR- α
- Restart feature for sequential analysis of hazards
- **Materials**
  - Elastic
  - Bilinear elasto-plastic
  - Hysteretic
  - Bouc-Wen
  - Trilinear
  - Stiffness degrading
  - Concrete
  - Steel
  - Fracture
  - Initial stress
  - Tension-only
  - Compression-only
  - SMA
- **Preprocessing**
  - Model Visualization
- **Postprocessing**
  - Data archival
  - Animation

HyCom-3D Users Manual

Revised 1-7-2024

HyCoM-3D: A Program for 3D Nonlinear Analysis and Real-Time Hybrid Simulation of 3-D Civil Infrastructure Systems

3D Hybrid Computational Modeling Program
Version 4.1.1 User's Manual

by

James Ricles
Bruce G. Johnston Professor of Structural Engineering
Department of Civil and Environmental Eng., Lehigh University

Chinmoy Kolay
Assistant Professor, Indian Institute of Technology Kanpur, India
(Former Research Scientist, ATLSS Center)

Thomas Marullo
Research Scientist, ATLSS Center

Faisal Nissar Malik
Research Assistant, CEE Department

ATLSS Report No. 20-02

Updated: January 2024

ATLSS is a National Center for Engineering Research
on Advanced Technology for Large Structural Systems

117 ATLSS Drive
Bethlehem, PA 18015-4729

Phone: (610)758-3525
Fax: (610)758-5902
www.atlss.lehigh.edu
Email: inatl@lehigh.edu

TABLE OF CONTENTS

HyCom-3D Users Manual

© Ricles and Malik, 2025

4.4.17 General Uniaxial Material Model for Cyclic Nonlinear Analysis of Steel-Based Elements -Steel AZ (TYPE 17) 34
4.5. Section data block 38
4.5.1. Wide flange section – 2D elements (TYPE 1) 39
4.5.2. Rectangular RC section– 2D elements (TYPE 2) 40
4.5.3. General Section– 2D elements (TYPE 3) 41
4.5.4. Wide flange section – 3D elements (TYPE 4) 42
4.5.5. Rectangular RC section – 3D elements (TYPE 5) 43
4.5.6. General Section– 3D elements (TYPE 6) 44
4.5.7 HSS Tube Section – 3D elements types 14, 15, 20, or 21) (TYPE 44
4.6. Element data block 45
4.6.1. Elastic beam-column element (TYPE 1) 43
4.6.2. Elastic spring element (TYPE 2) 44
4.6.3. Plastic hinge inelastic beam-column element (parallel component theory) (TYPE 3) 45
4.6.4. Dummy column element (TYPE 4) 47
4.6.5. Rotational spring element with stiffness and strength deterioration capabilities (TYPE 5) 48
4.6.6. Displacement based fiber beam-column element (TYPE 6) 51
4.6.7. Force based fiber beam-column element (TYPE 7) 52
4.6.8. Zerolength element (TYPE 8) 54
4.6.9. 2D Planar panel zone element (TYPE 9) 55
4.6.10 2D Truss element (TYPE 10) 56
4.6.11. 3D Elastic beam-column element (TYPE 11) 57
4.6.12. 3D Dummy column element (TYPE 12) 60
4.6.13. 3D Truss element (TYPE 13) 61
4.6.14. 3D Displacement based fiber beam-column element (TYPE 14) 62
4.6.15. 3D Force based fiber beam-column element (TYPE 15) 65
4.6.16. 3D zerolength element (TYPE 16) 69
4.6.17. 3D Elastic Super element (TYPE 17) 72
4.6.18. 3D Consolidated Elastic Super element (TYPE 18) 73
4.6.19 3D Elastic beam-column Co-rotational element (TYPE 19) 75
4.6.20 3D Force-based Co-rotational fiber element (TYPE 20) 78
4.6.20 3D Displacement-based Co-rotational fiber element (TYPE 21) 82
4.6.22 3D Uniaxial Experimental Element (TYPE 22) 85
4.6.23 3D Multi-axis Experimental Element (TYPE 23) 86

v

HyCom-3D Users Manual

4.6.15 3D Force-based Fiber Beam-Column Element (TYPE 15)

```
element 3DforceBeamColumn [Units] [ELID] [NODEI] [NODEJ] [DAMPK] [DAMPM] [G]
[A] [Asy] [Asz] [J] [Wt/L] [Theta] [eI] [eJ] integration [intType] [intData] iter [MaxIter]
[TOL] [CO] elementDamping [elmDamp]
```

G	:	Shear modulus	F
A		Cross-sectional (average) of member for purpose of computing J/A quantity in element mass matrix	F
Asy	:	Effective cross-sectional shear area with respect to loading parallel to the local y-axis (local 2-2 axis)	F
Asz	:	Effective cross-sectional shear area with respect to loading parallel to the local z-axis (local 3-3 axis)	F
J	:	Polar moment of inertia	F
Wt/L	:	Element gravity load per unit length (used to generate element consistent mass matrix)	F
Theta	:	Rotational angle (in degrees) of local axis about local x-x axis	F
eI	:	Element rigid end offset at node I; length of rigid end zones expressed as fraction of total element length	F
eJ	:	Element rigid end offset at node J; length of rigid end zones expressed as fraction of total element length	F
intType	:	String based argument for element integration type. Available options are `Lobatto`, `Legendre`, `HingeRadau`, `HingeRadauTwo`, and `RegularizedHingeLobatto`	S
intData	:	It depends on the chosen `intType` as noted below.	

intType		intData
`Lobatto` or `Legendre`	:	`[NIP] [SECIDs]`
`HingeRadau` or `HingeRadauTwo` [6]	:	`[secIDI] [LpI] [secIDJ] [LpJ] [secIDInt]`
`RegularizedHingeLobatto` [7]	:	`[NIP] [secIDI] [LpI] [zetaI] [secIDJ] [LpJ] [zetaJ] [secIDInt]`

NIP	:	Number of integration points along the length of element	F
SECIDs	:	Section ID number(s) for Lobatto or Legendre integration which can be one ID for all sections or multiple IDs for each integration points	I
secIDI	:	Section ID of the integration point(s) at the end-I for HingeRadau, HingeRadauTwo, and RegularizedHingeLobatto integration schemes	I
LpI	:	Length of plastic hinge at end-I for HingeRadau, HingeRadauTwo, and RegularizedHingeLobatto integration schemes	F
zetaI	:	Distance of the additional integration point inside the element end-I for RegularizedHingeLobatto integration scheme	F
secIDJ	:	Section ID of the integration point(s) at the end-J for HingeRadau, HingeRadauTwo, and RegularizedHingeLobatto integration schemes	I
LpJ	:	Length of plastic hinge at end-J for HingeRadau, HingeRadauTwo, and RegularizedHingeLobatto integration schemes	F
zetaJ	:	Distance of the additional integration point inside the element end-J for RegularizedHingeLobatto integration scheme	F

70

secIDInt	:	Section ID of the interior integration points for the HingeRadau, HingeRadauTwo, or RegularizedHingeLobatto integration schemes.	
MaxIter	:	Maximum number of iterations to satisfy element compatibility	F
TOL	:	Tolerance for iterations (typical value of 10^{-16} is used)	F
CO	:	=1 for carrying over unbalanced section forces; =0 for not carrying over	I
elmDamp	:	String based argument for formulating the stiffness proportional part of the element damping matrix. Available options are `CommittedStiffness`, `CurrentStiffness`, and `InitialStiffness`, which uses the element stiffness matrix at the previous time step, current time step, and beginning of the simulation ($t = 0$), respectively.	S
Units	:	Units used in simulation, either: (1) m/sec2; (2) mm/ sec2; (3) ft/ sec2; or (4) in/ sec2. Note that units for element consistent mass matrix must be consistent with respect to units for simulation model. Chose units for accelerations to convert element gravity load per unit length [Wt/L] to mass per unit length.	S

Example(s):

```
# Force based element with HingeRadu integration, 10 iterations, Tol=1e-16, carry over, and current tangent stiffness matrix for damping formulation
element 3DforceBeamColumn in/sec2 1 1 2 1 1 11500 3.75 0.0 0.0 15.0 0.150/12 0.0 0.0 0.0 integration HingeRadau 1 0.5303 1 0.5303 2 iter 10 1.00E-16 1 elementDamping CurrentStiffness

# Force based element with Legendre integration with 5 points (assign SectionID 1 to 1st and 5th integration points, assign SectionID 2 to 2nd through 4th integration points), 5 iterations, Tol=1e-16, carry over, and initial stiffness matrix for damping formulation
element 3DforceBeamColumn in/sec2 1 1 2 1 1 11500 3.75 0.0 0.0 15.0 0.150/12 0.0 0.0 0.0 integration Legendre 5 1 2 2 2 1 iter 5 1.00E-16 1 elementDamping InitialStiffness

# Force based element with Legendre integration with 5 points (assign SectionID 2 all integration points), 5 iterations, Tol=1e-16, carry over, and initial stiffness matrix for damping formulation
element 3DforceBeamColumn in/sec2 1 1 2 1 1 11500 3.75 0.0 0.0 15.0 0.150/12 0.0 0.0 0.0 integration Legendre 5 2 iter 5 1.00E-16 1 elementDamping InitialStiffness
```

Notes

1. The theoretical development for this element can be found in Ref. [8]
2. See Ref. [9] for further details on the element integration schemes.
3. Assign the type of units used to define the value of Load/unit length, (Load). The element gravity load per unit length is converted by the program to element mass per unit length according to: Mass= Load /g, where g is the acceleration of gravity. The program assigns the value for g based on the selected Units (i.e., if Units = m/sec2, then g=9.81 m/sec^2, or Units = in/sec2, then g= 386.4 in/sec^2)
4. Mass units need to be consistent with respect to model and simulation data.

71

HyCom-3D Users Manual

5. Define eI and eJ as fraction of element length, e.g., eI = 0.25 implies that the rigid zone has a length of 0.25L at node I). Rigid offsets are assumed to in between nodes I and J, i.e., the flexible element length is L = L-eI*L – eJ*L. Note that the rigid end offsets affects flexural, axial, and torsional stiffness.
6. For the InitialStiffness element damping type the damping matrix of the element is calculated as [C] = DampM*A_o*[M] + DampK*A_1*[K], where [M] is the element initial stiffness matrix, [K] is the element mass matrix, A_o and A_1 are the Rayleigh proportional damping factors, and DampM and DampK are the user defined element modifier for assigning nonproportional damping to the element.
7. The factor A_1 is applied to the damping forces based on CommittedStiffness, CurrentStiffness, and DampK is not applicable. For these types of element damping the damping matrix of the element associated with mass is calculated as [C] = DampM*A_o*[M], where [M] is the element initial stiffness matrix. This element damping matrix is assembled into then structural damping matrix. The element mass-proportional damping forces are supplemented with any stiffness-proportional damping forces. The element stiffness-based damping forces are obtained during the state determination process and included in the element restoring forces, where the damping component is [$R_{I,elm\_damping}$] = A_1* [$K_{T,elm}$]* [V_c].
8. A_o and A_1 are both assigned values of zero if no Rayleigh Proportional is specified.
9. Positive sign convention for section forces and deformations is associated with local y- and z-axes and right-hand rule.
10. The local coordinate system for the element is shown below in Fig. 18. A rotation of Theta is taken with respect to the element's local axis 1-1 (local x-x axis), with counter-clockwise being positive (i.e., using the right-hand rule), see Fig. 19. The default positioning of the local coordinate system in the global coordinate system also appears below in Fig. 19. The position of the element's local 3-3 axis is determined from the vector cross product of 1-1 x Y-Y = 3-3, where vector Y-Y is a unit vector parallel to the global Y-axis. An exception to how this is calculated occurs for elements that are parallel to the global Y-axis, where is this case X-X x 1-1 = 3-3, where vector X-X is a unit vector parallel to the global X-axis; see Fig. 19.

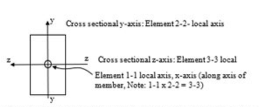


Fig. 18. Element cross-sectional local axes, and definition of local z-z and y-y axes (positive section moment and curvature is associated with local y- and z-axes and right hand rule).

72

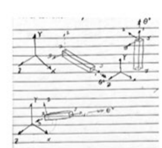


Fig. 19. Positioning of element local axes with respect to global coordinate system.

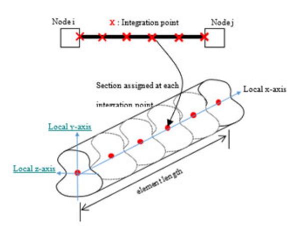
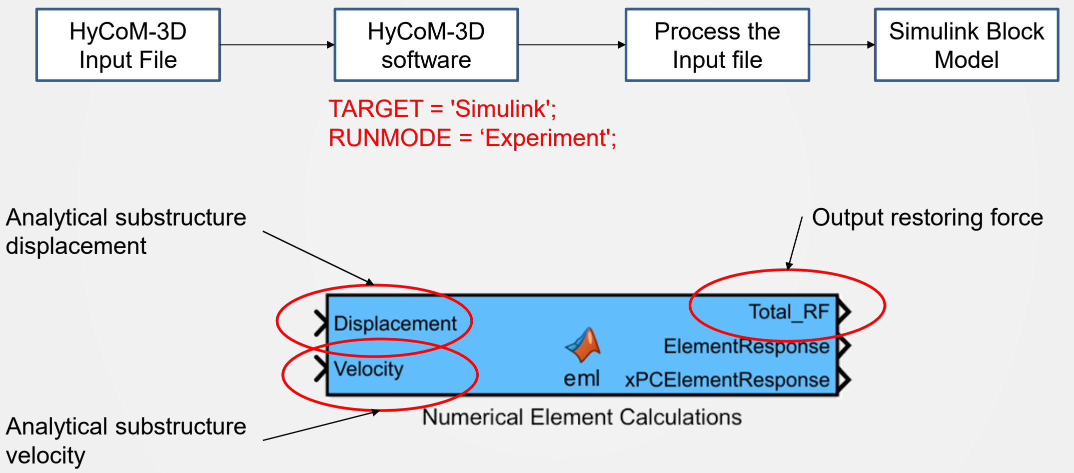


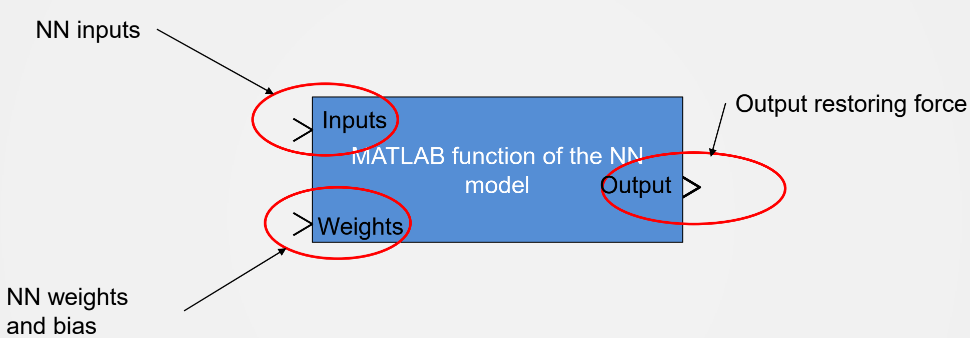
Fig. 20. Schematic view of beam-column element Type 14 and Type 15 [1]

73

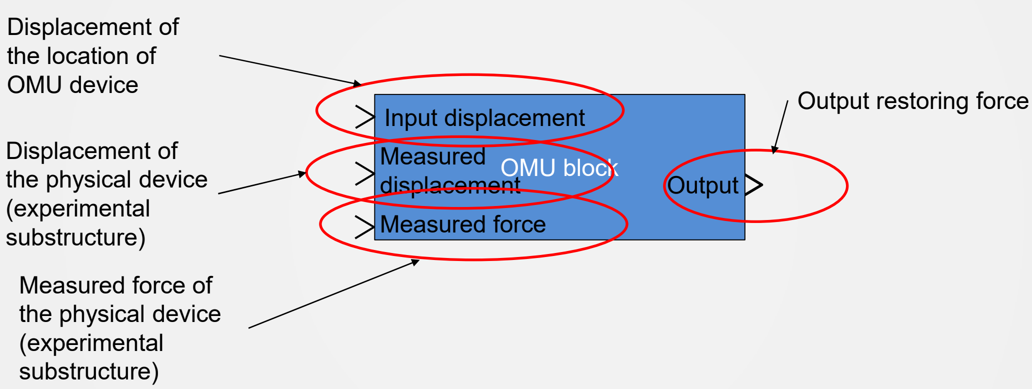
Simulink Block of System Model



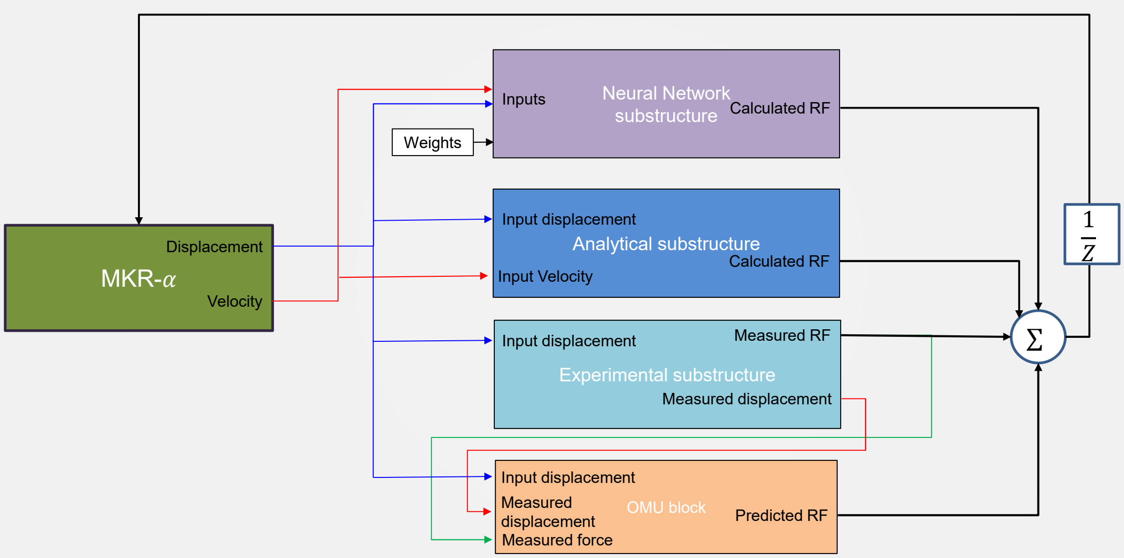
Simulink Block of Neural Network Model



Simulink Block of Online Model Updating using CUKF



Combined Simulink Model of the RTHS



Configuring Actuator Control

- Adaptive timeseries compensation scheme used for Actuator control

$$x^{compensated} = a_0 x^{command} + a_1 \dot{x}^{command} + a_2 \ddot{x}^{command}$$

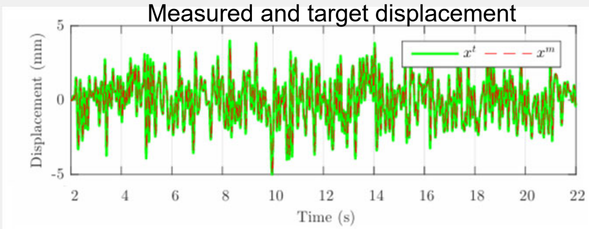
- The initial values of the ATS coefficients obtained from BLWN tests
  - Steps:
    - Perform BLWN tests (usually 0-20 Hz) without ATS
    - Solve the following equation between the target and measured responses

$$\boldsymbol{A} = (x_m^T x_m)^{-1} x_m^T x_t$$

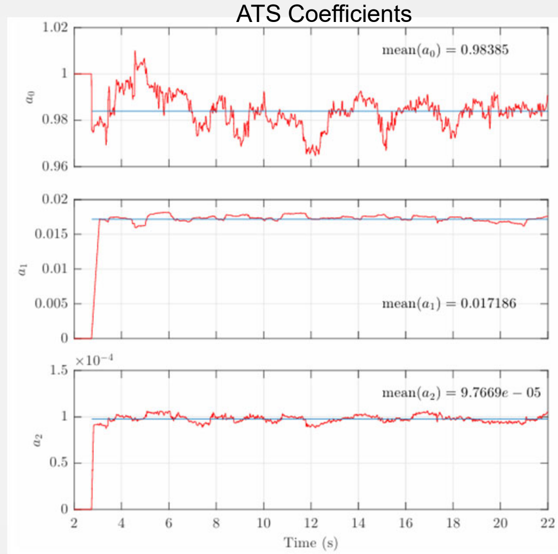
where, x_m is measured and x_t is the target displacement

Configuring Actuator Control

Measured and target displacement

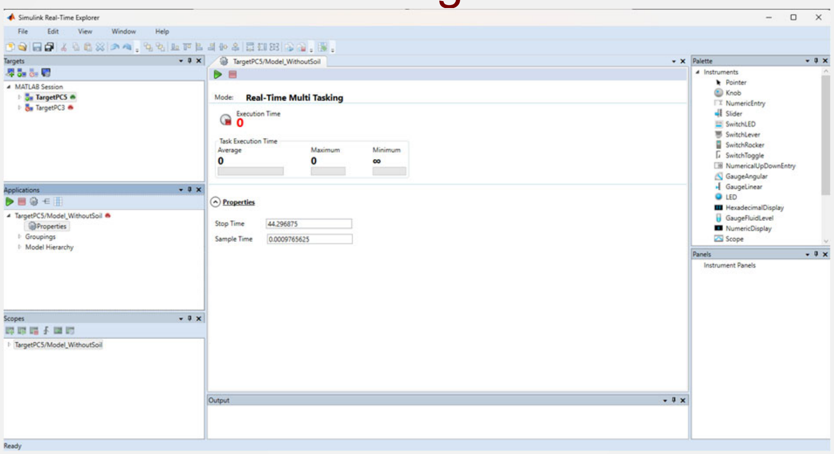


ATS Coefficients



- Initial values taken as the mean values
- Upper and lower limits taken based on the recommendations by Chae et. al (2013) in order to avoid spurious magnification of signal

Performing a RTHS



Further details to be provided in Session 5

LEHIGH UNIVERSITY ATLSS RTMD NSF 75 YEARS OF DISCOVERY & INNOVATION NSF NHERI CYBER-PHYSICAL SIMULATION 21

Assessing RTHS results

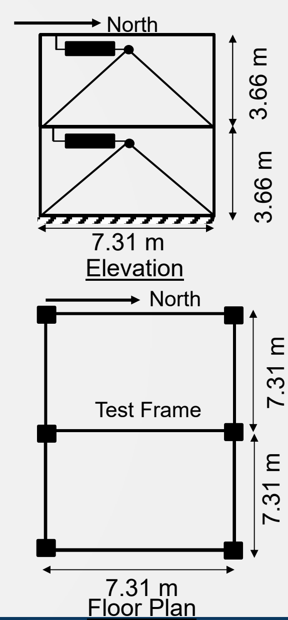
Details to be provided in Session 4

LEHIGH UNIVERSITY ATLSS RTMD NSF 75 YEARS OF DISCOVERY & INNOVATION NSF NHERI CYBER-PHYSICAL SIMULATION 22

Session 4: Hands-on RTHS Group Assignment

Short Course Assignment

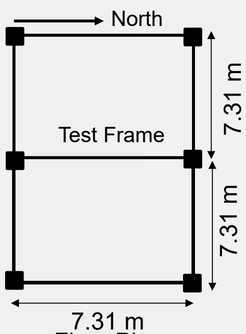
- W12X45 beams and W14X43 columns
- 3D force-based fiber beam column elements
  - BelPlastic Material as uniaxial material in the fibers
  - 5-point Lobatto integration scheme
- P-Delta effects accounted for by using lean on P-Delta columns
- Gravity loading
  - 413.34 kN on the first floor
  - 411.28 kN on the second floor
- Building located in Pomona California on a stiff soil (Type D)
- Two levels of hazard considered
  - DBE (474 years return period)
  - MCE (2474 years return period)
- Friction dampers in the first and second story



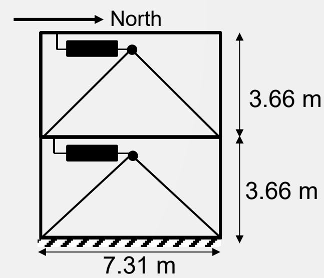
Short Course Assignment

Evaluate the seismic performance of the Lateral Force Resisting System (LFRS) for the structure shown below

- What is the efficacy of the dampers?
- What are the effects of the soil-foundation-structure interaction?
- What is the performance under two prescribed hazard levels –
  - Design Basis Earthquake (DBE – 474-year return period)
  - Maximum Considered Earthquake (MCE – 2474-year return period)



Floor Plan

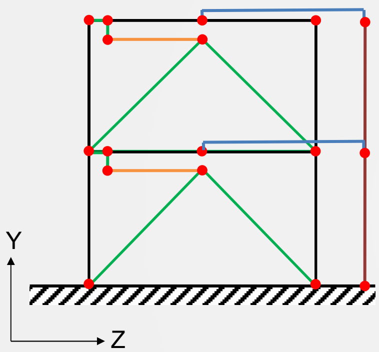


Elevation

Short Course Assignment

- A HyCoM-3D input file and manual are provided on Workstations in Life Cycle Computational Lab
- The file needs to be completed by each individual group
  - Compare the natural periods obtained from the eigenvalue analysis
- A total of 6 cases need to be run
  - Without dampers (Will be done in Life Cycle Computational Lab)
    - DBE and MCE
  - With dampers on a fixed foundation (RTHS – Will be done in the control room)
    - DBE and MCE
  - With dampers considering SFSI (RTHS – Will be done in the control room)
    - DBE and MCE
- Each group will be assigned a workstation and will have to complete the HyCoM-3D input file and run the cases without dampers for the two specified hazard levels
  - Compare the results obtained from your analysis to the provided results

HyCoM-3D model and input file



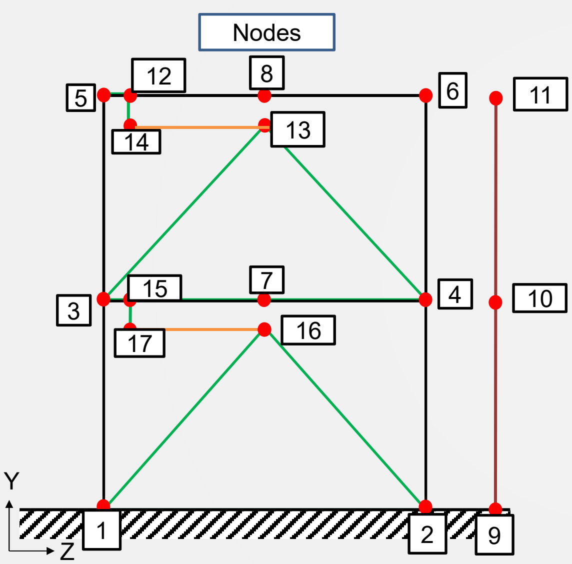
- 3DforceBeamColumnElement
- 3DelasticBeamColumnElement
- 3Ddummy element
- 3DzeroLengthElement (Numerical)/3Duni_exprElement (RTHS)
- Nodes
- Rigid Floor Diaphragms

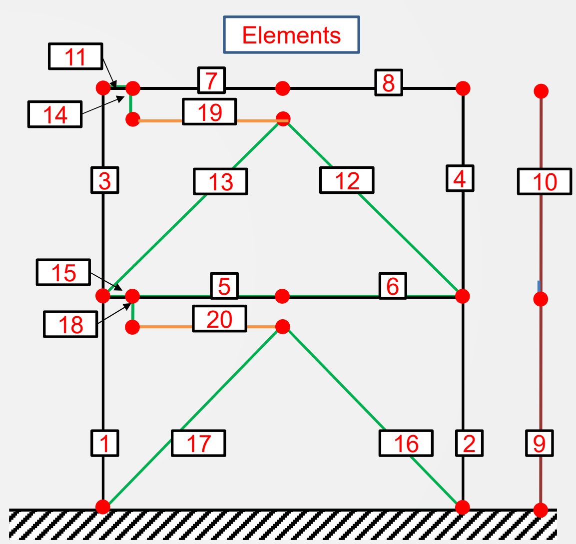
The building is modeled in the YZ plane (Y – Vertical DOF and Z is Translational DOF)

27

© Ricles and Malik, 2025

Node and Element Numbering





HyCoM-3D model and input file (Nodes)

- For each node, the quantities to be defined are
  - Node ID
  - Coordinates (X, Y, and Z)
  - Mass (X, Y, Z, $\theta_X, \theta_Y, \theta_Z$)
- The nodes are already defined in your input file

```
# Define nodes
# node [nodeID] [XCoord] [YCoord] [ZCoord] mass [X] [Y] [Z] [ThetaX] [ThetaY] [ThetaZ]
node    1   7.32    0.00    0.00    mass    0.0001  0.0001  0.0001  0.0001  0.0001  0.0001
node    2   7.32    0.00    7.32    mass    0.0001  0.0001  0.0001  0.0001  0.0001  0.0001
node    3   7.32    3.66    0.00    mass    0.0001  0.0001  0.0001  0.0001  0.0001  0.0001
node    4   7.32    3.66    7.32    mass    0.0001  0.0001  0.0001  0.0001  0.0001  0.0001
node    5   7.32    7.32    0.00    mass    0.0001  0.0001  0.0001  0.0001  0.0001  0.0001
node    6   7.32    7.32    7.32    mass    0.0001  0.0001  0.0001  0.0001  0.0001  0.0001
node    7   7.32    3.66    3.66    mass    0.0001  0.0001  0.0001  0.0001  0.0001  0.0001
node    8   7.32    7.32    3.66    mass    0.0001  0.0001  0.0001  0.0001  0.0001  0.0001
node    9   7.32    0.00    8.84    mass    0.0001  0.0001  0.0001  0.0001  0.0001  0.0001
node    10  7.32    3.66    8.84    mass    0.0001  0.0001  42.136  0.0001  0.0001  0.0001
node    11  7.32    7.32    8.84    mass    0.0001  0.0001  41.918  0.0001  0.0001  0.0001

node    12  7.32    7.32    0.20    mass    0.0001  0.0001  0.0001  0.0001  0.0001  0.0001
node    13  7.32    7.10    3.66    mass    0.0001  0.0001  0.0001  0.0001  0.0001  0.0001
node    14  7.32    7.10    0.20    mass    0.0001  0.0001  0.0001  0.0001  0.0001  0.0001

node    15  7.32    3.66    0.20    mass    0.0001  0.0001  0.0001  0.0001  0.0001  0.0001
node    16  7.32    3.44    3.66    mass    0.0001  0.0001  0.0001  0.0001  0.0001  0.0001
node    17  7.32    3.44    0.20    mass    0.0001  0.0001  0.0001  0.0001  0.0001  0.0001
```

© Ricles and Malik, 2025

HyCoM-3D model and input file (Fix Constraints)

- Use fix command to define constraints
- Syntax – fix [nodeID] [X] [Y] [Z] [thetaX] [thetaY] [thetaZ]
- 0 = free and 1 = fixed

```
# Define constraints for fix condition at base (1 = fixed, 0 = free)
#fix nodeID [X] [Y] [Z] [thetaX] [thetaY] [thetaZ]
fix 1   1   1   1   1   1   1
```

Fix conditions at base

```
##### Do it yourself #################
# Assign boundary condition to remaining nodes at base of structure?
#########################################
```

```
# Fix out of plane motion of all remaining nodes (1 = fixed, 0 = free)
# HyCoM-3D is a Multi-directional analysis software.
# For 2D analysis: Contraint Out of plane motions (i.e., in X direction)
#fix nodeID [X] [Y] [Z] [thetaX] [thetaY] [thetaZ]
fix 3 1 0 0 0 1 1
fix 4 1 0 0 0 1 1
fix 5 1 0 0 0 1 1
fix 6 1 0 0 0 1 1
fix 7 1 0 0 0 1 1
fix 8 1 0 0 0 1 1
```

Fix out of plane DOFs of remaining nodes

```
fix 10 1 1 0 1 1 1
fix 11 1 1 0 1 1 1
```

Only keep translational DOF for 3Ddummy column nodes

HyCoM-3D model and input file (Define Sections for Force-based fiber beam & column elements)

- Sections need a uniaxialMaterial for the fibers
  - We are using BelPlastic, however other materials can also be used
- uniaxialMaterial BelPlastic [MATID] [E] [SigmaY] [Alpha] [sigInit]
- section3D [Units] WFSection [SECID] [nomDepth] [nomWeight] [NFFT] [NFWD] [NFFW] [NFWT] [MATID]

```
# Define the BelPlastic material
# uniaxialMaterial BelPlastic [MATID] [E] [SigmaY] [Alpha] [sigInit]
uniaxialMaterial BelPlastic 1 2.1e8 415000 0.01 0.0

# Define Sections
# Beams use W12X45 and Columns use W14X43
# section3D [Units] WFSection [SECID] [nomDepth] [nomWeight] [NFFT] [NFWD] [NFFW] [NFWT] [MATID]
section3D m WFSection 1 14 43 5 10 10 5 1
```

```
############################ Do it yourself ######################
# Define WFSection for Beams. Note: Use Section Tag = 2 and Use same number of fibers and materialID as that used for the columns?
##################################################################
```

Need to define WF sections for beams. Note: Use section tag = 2

HyCoM-3D model and input file (Define force-based fiber elements for beams and columns)

- element 3DforceBeamColumn [Units] [ELID] [NODEI] [NODEJ] [DAMPK] [DAMPM] [G] [A] [Asy] [Asz] [J] [Wt/L] [Theta] [eI] [eJ] integration [intType] [intData] iter [MaxIter] [TOL] [CO] elementDamping [elmDamp]

```
# Define elements for the columns and beams
# We are using explicit force-based fiber elements with Lobatto integration scheme and 5 integration points
# Element damping matrix based on current stiffness
# All elements are massless
# Since the Frame is defined in YZ plane. The columns need to be rotated to align the strong axis with bending in YZ plane
# This is achieved by using Theta = 90 degree in the column elements
# element 3DforceBeamColumn [Units] [ELID] [NODEI] [NODEJ] [DAMPK] [DAMPM] [G] [A] [Asy] [Asz] [J] [Wt/L] [Theta] [eI] [eJ] integration [intType] [intData] iter [MaxIter] [TOL] [CO] elementDamping [elmDamp]
element 3DforceBeamColumn  m/sec2  1  1 3  1 1  21e6  0.1858 0  0  5.7939e-3  0  90  0.0000  0.0556  integration Lobatto 5 1 1 1 1 1 iter  2  1.0E-16 1  elementDamping  CurrentStiffness
element 3DforceBeamColumn  m/sec2  2  2 4  1 1  21e6  0.1858 0  0  5.7939e-3  0  90  0.0000  0.0556  integration Lobatto 5 1 1 1 1 1 iter  2  1.0E-16 1  elementDamping  CurrentStiffness
element 3DforceBeamColumn  m/sec2  3  3 5  1 1  21e6  0.1858 0  0  5.7939e-3  0  90  0.0556  0.0556  integration Lobatto 5 1 1 1 1 1 iter  2  1.0E-16 1  elementDamping  CurrentStiffness
element 3DforceBeamColumn  m/sec2  4  4 6  1 1  21e6  0.1858 0  0  5.7939e-3  0  90  0.0556  0.0556  integration Lobatto 5 1 1 1 1 1 iter  2  1.0E-16 1  elementDamping  CurrentStiffness
element 3DforceBeamColumn  m/sec2  5  3 7  1 1  21e6  0.1703 0  0  5.7506e-3  0  0   0.0556  0.0000  integration Lobatto 5 2 2 2 2 2 iter  2  1.0E-16 1  elementDamping  CurrentStiffness
element 3DforceBeamColumn  m/sec2  6  4 7  1 1  21e6  0.1703 0  0  5.7506e-3  0  0   0.0556  0.0000  integration Lobatto 5 2 2 2 2 2 iter  2  1.0E-16 1  elementDamping  CurrentStiffness
element 3DforceBeamColumn  m/sec2  7  5 8  1 1  21e6  0.1393 0  0  3.5063e-3  0  0   0.0556  0.0000  integration Lobatto 5 2 2 2 2 2 iter  2  1.0E-16 1  elementDamping  CurrentStiffness
element 3DforceBeamColumn  m/sec2  8  6 8  1 1  21e6  0.1393 0  0  3.5063e-3  0  0   0.0556  0.0000  integration Lobatto 5 2 2 2 2 2 iter  2  1.0E-16 1  elementDamping  CurrentStiffness
```

HyCoM-3D model and input file: Rigid floor diaphragm modeling

- We are connecting the 3Ddummy column to middle of the bay with equalDOF constraints to model rigid floor diaphragms
- equalDOF [MNODE] [SNODE] [UX] [UY] [UZ] [THETAX] [THETAY] [THETAZ]

```
# #################################### Define Rigid Floor Diaphragms #####################
# We define floor diaphragms between the center of the MRF bay and the P-delta column
# This is acheived by slaving the 3Ddummy column nodes to center of the bays by equalDOF constraints in Translation (i.e., Z axis)

#equalDOF [MNODE] [SNODE] [UX] [UY] [UZ] [THETAX] [THETAY] [THETAZ]
equalDOF 7 10 0 0 1 0 0 0
equalDOF 8 11 0 0 1 0 0 0
# ########################################################################################
```

© Ricles and Malik, 2025

HyCoM-3D model and input file: 3Ddummy columns

- Required for P-Δ effects
- element 3Ddummy [ELID] [NODEI] [NODEJ] [DAMPK] [DAMPM] [W] [Mi] [Mj] [MDOF]
- MDOF is the direction: 1 for X, 2 for Z and 3 for both X and Z
- W is the cumulative floor weight i.e., at floor 1 it will be sum of floor 1 and floor 2 weights

```
# Use 3Ddummy column (Lean on column for P-delta effects)
set FW1 824.56
set FW2 411.21
# set command is used to define user defined variables
# FW1 and FW2 are cummulative weigths at floors 1 and 2 i.e., FW1 is sum of floor 1 weight and floor 2 weight
#element 3Ddummy [ELID] [NODEI] [NODEJ] [DAMPK] [DAMPM] [W] [Mi] [Mj] [MDOF]
# MDOF is the direction: 1 for X, 2 for Z and 3 for both X and Z
element 3Ddummy 9    9  10  1   1  $FW1  0   0   2
```

```
############# Do it yourself ######################
#Define Lean on column for second story using FW2 as axial load. Use Element Tag = 10
###################################################
#If you have time to do it, consider increasing the the gravity load on 3Ddummy columns to increase the P-Delta effect and see what happens?
```

- Define the second lean on P-Δ column.
- Additional exercise: If you have time, increase the weight on P-Δ Column by a factor of 2 to see what happens

HyCoM-3D model and input file: Damper connection system

- Includes damper clevises, connection to columns, and diagonal braces
- Using elastic beam-column elements for the current model
- # element 3DelasticBeamColumn [Units] [ELID] [NODEI] [NODEJ] [DAMPK] [DAMPM] [A] [E] [G] [Asy] [Asz] [Iz] [Iy] [J] [Wt/L] [Theta] [eI] [eJ] <[EndR] [opt]> <AxialRel> <TorqueRel>

```
# Define Elastic beam column elements for the diagonal bracing system (Includes braces, clevises, and connections)
# The braces are pin ended
# Here this is achieved by using end releases at both ends on one brace and only one end of other brace
# Braces at second story
# element 3DelasticBeamColumn [Units] [ELID] [NODEI] [NODEJ] [DAMPK] [DAMPM] [A] [E] [G] [Asy] [Asz] [Iz] [Iy] [J] [Wt/L] [Theta] [eI] [eJ] <[EndR] [opt]> <AxialRel> <TorqueRel>
element 3DelasticBeamColumn m/sec2 11  5   12 1 1 0.1393 2.0e9 76903068.7654  0  0  1.60e-3 1.60e-3 3.5063e-3  0  0  0  0
element 3DelasticBeamColumn m/sec2 12  4   13 1 1 0.0038 2.0e8 76903068.7654  0  0  1.08e-5 1.08e-5 1.7565e-5  0  0  0  0 EndR RIJz
element 3DelasticBeamColumn m/sec2 13  3   13 1 1 0.0038 2.0e8 76903068.7654  0  0  1.08e-5 1.08e-5 1.7565e-5  0  0  0  0 EndR RIz
element 3DelasticBeamColumn m/sec2 14  12  14 1 1 0.1393 2.0e9 76903068.7654  0  0  2.43e-3 2.43e-3 4.4300e-3  0  0  0  0
```

- Connection (i.e., element 11) and clevis (i.e., element 14) are modeled as stiff elements: Youngs modulus = 2e9
- Diagonal braces are pin ended – Moment end release at one end (i.e., connection with column) of brace 1 (i.e., element 13) and both ends (i.e., connection with column and connection with another brace) of brace 2 (i.e., element 12)

```
element 3DelasticBeamColumn m/sec2 15   3   15 1 1 0.1393 2.0e9 76903068.7654  0  0  1.60e-3 1.60e-3 3.5063e-3  0  0  0  0
############# Do it yourself ######################
# Define the two diagonal braces of the first story. Note: Use element tags of 16 and 17. They are similar to that of the second story.
##################################################
element 3DelasticBeamColumn m/sec2 18   15   17 1 1 0.1393 2.0e9 76903068.7654  0  0  2.43e-3 2.43e-3 4.4300e-3  0  0  0  0
```


HyCoM-3D model and input file: Dampers

- Dampers are modeled using 3Dzerolength element for numerical analysis
  - Zerolength elements can have nonzero length
  - Needs a uniaxial material – We are using BelPlastic
  - # element 3DzeroLength [ELID] [NODEI] [NODEJ] [NODEK] [DAMPK] [DAMPM] [DOF] [Theta] [MATID] [Type=1 (Always)]
- Damper are modeled with 3Duni_exptElement for RTHS
  - Used to add C_{eq} and K_{eq} of the damper to the integration parameter matrices
  - Restoring forces are obtained from the actual damper

```
# ################################## Define Dampers ################################
set NumDampers 0
# set command is used to assing the number of dampers
# We are using 0 dampers for current analysis (i.e., the case without dampers)
# Alternatively you can use 10 dampers for the case with dampers
# ####### Numerical dampers ######
# A numerical damper is defined by adding a bilinear plastic material to zerolength elements (zerolength element can have nonzero length)
# #uniaxialMaterial BelPlastic [MATID] [E] [SigmaY] [Alpha] [sigInit]
uniaxialMaterial BelPlastic 2 16000*$NumDampers 16*$NumDampers 0.001 0.0
# element 3DzeroLength [ELID] [NODEI] [NODEJ] [NODEK] [DAMPK] [DAMPM] [DOF] [Theta] [MATID] [Type=1 (Always)]
element 3DzeroLength 19 13 14 14 1 1 1 0 2 1
element 3DzeroLength 20 17 16 16 1 1 1 0 2 1
```

Additional exercise: Use NumDamper 10 and compare the results to the RTHS results.

HyCoM-3D model and input file: Inherent damping and analysis

- Inherent damping modeled using modalDamping – Applies a constant damping ratio to all modes
  - Alternatively, Rayleigh damping can be used

```
# Add damping to the model (We are using 3% Modal Damping in all the modes)
modalDamping 36 0.03
```

- Gravity loads applied on the frame columns (for moment-axial force interaction) using static loading command
  - StaticLoad [NodeID] [DOF] [StaticLOAD] [ASNum] [NumLoadInc] [IterMax] [IterTol]

```
#Static Analysis
#Applies gravity load to the columns
#Runs deflection due to dead load point load determined by tributary area
#StaticLoad [NodeID] [DOF] [StaticLOAD] [ASNum] [NumLoadInc] [IterMax] [IterTol]
set F1  -413.35/2
set F2  -411.21/2
StaticLoad  3   2   $F1   1   10   200 1e-05
StaticLoad  4   2   $F1   1   10   200 1e-05
```

```
############### Do it yourself #########################
#Apply Gravity load to second story. Use same ASNum, NumLoadInc, IterMax, and IterTol as above and use $F2 as the gravity load.
#######################################################
```

At what nodes should the gravity load be applied to?

- Dynamic analysis done using MKR-α integration algorithm

```
# integrator MKRAlpha [rhoInfy] [SF] [DT] [INT]
integrator MKRAlpha 0.25 9.81*$SF 5/1024 5
```


HyCoM-3D model and input file: Misc commands

- Set command used for user defined variables
  - e.g., set SF 1
- recorders – used to record element responses

```
# ############ Record moment and curvature/ Axial force and deformation in columns and beams
# recorder Element [elmIDs] section [secID]
recorder Element  1 2 3 4 5 6 7 8  section 1
recorder Element  1 2 3 4 5 6 7 8  section 2
recorder Element  1 2 3 4 5 6 7 8  section 3
recorder Element  1 2 3 4 5 6 7 8  section 4
recorder Element  1 2 3 4 5 6 7 8  section 5
# ############ Record global force in beams, columns and braces
# recorder Element [elmIDs] globalForce
recorder Element  1 2 3 4 5 6 7 8 12 13 16 17 globalForce

# recorder Element [elmIDs] basicForce
recorder Element  19 20  basicForce
recorder Element  19 20  basicDeformation
# ##########################################################################################
```

LEHIGH UNIVERSITY ATLSS | RTMD REAL-TIME MULTI-DIRECTIONAL SIMULATION | NSF 75 YEARS OF DISCOVERY & INNOVATION | NSF NHERI CYBER-PHYSICAL SIMULATION

How to run the numerical analysis (MATLAB mode)

MainFile.m

```
addlibraries
clc;clear;
EQFile = 'RSN779_LOMAP_LGP000_0_unit_g_MCE.txt';
INP_File='2D_Steel_Frame_With_Dampers.tcl';
TARGET = 'Matlab';
RUNMODE = 'Simulation';
UICheck = 1;
LoadEQ
RunModel
```

- EQFile → Name of the earthquake
- INP_File → Name of the input file (i.e., the model)
- TARGET / RUNMODE → Run on MATLAB (Numerical) or Create a block for RTHS
- UICheck → Keep 1 for now

- Once everything is set (Press Run button in MATLAB)

What happens after pressing run button (Preprocessing)

Command Window

```
[FEM] Is this a Restart simulation? y/[n]:
```

We are not doing a restart simulation (so type n and press enter)

Command Window

```
[FEM] Is this a Restart simulation? y/[n]: n
[FEM] Clear workspace? y/[n]:
```

Clears the workspace

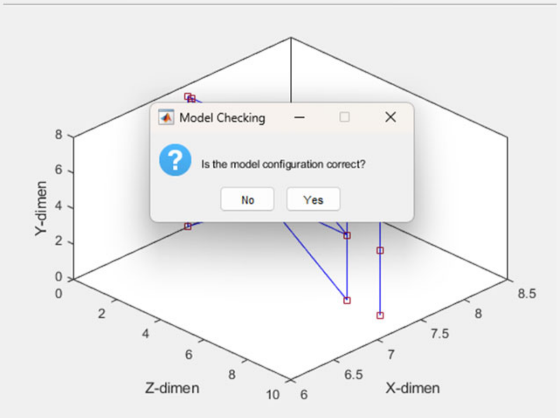
Command Window

```
[FEM] Is this a Restart simulation? y/[n]: n
[FEM] Clear workspace? y/[n]: y
[FEM] Workspace cleared, no Restart
[FEM] Proceeding with initial(non-Restart) simulation
  Hit Enter to continue
```

Press enter to continue with modeling

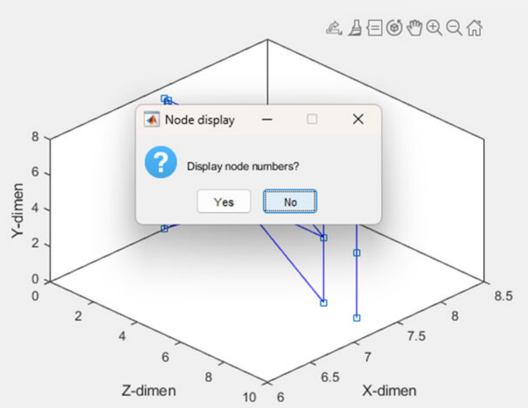
LEHIGH UNIVERSITY ATLSS RTMD REAL-TIME MULTI-DIRECTIONAL SIMULATION NSF 75 YEARS OF DISCOVERY & INNOVATION NSF NHERI CYBER-PHYSICAL SIMULATION

What happens after pressing run button (Preprocessing)



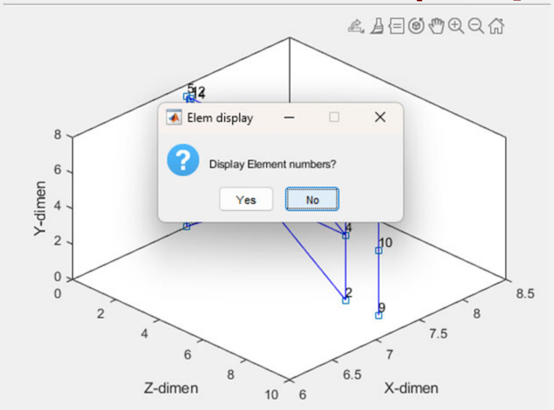
Shows you the model. User needs to press Yes

What happens after pressing run button (Preprocessing)



Press Yes to see node numbers

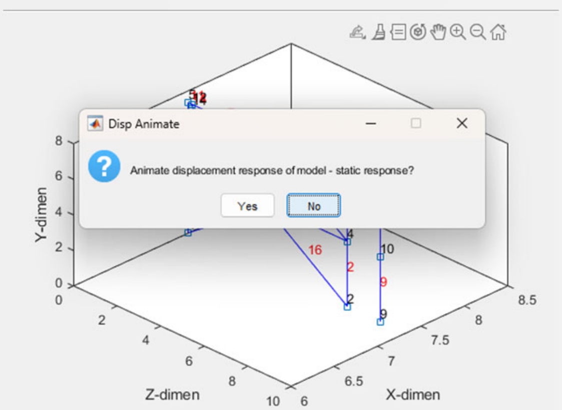
What happens after pressing run button (Preprocessing)



- Press Yes to see element numbers
- After pressing Yes – It will do a gravity analysis and ask if you want to visualize the gravity response

```
Command Window
Static Analysis Completed
   Load Increment number = 10
   Iteration Number= 1  Initial Iteration Cycle Energy Error = 1.371818e-24
   ***Convergence Achieved
Static Analysis Completed
Analysis Stage 2
 : generate initial conditions for start of dynamic analysis
   Load Increment number = 1
Element 1 failed to converge; carrying over
Element 2 failed to converge; carrying over
Element 3 failed to converge; carrying over
Element 4 failed to converge; carrying over
   Iteration Number= 1  Initial Iteration Cycle Energy Error = 5.821893e-25
   ***Convergence Achieved
Static Analysis Completed
```


What happens after pressing run button (Preprocessing)



- Press Yes to visualize gravity response

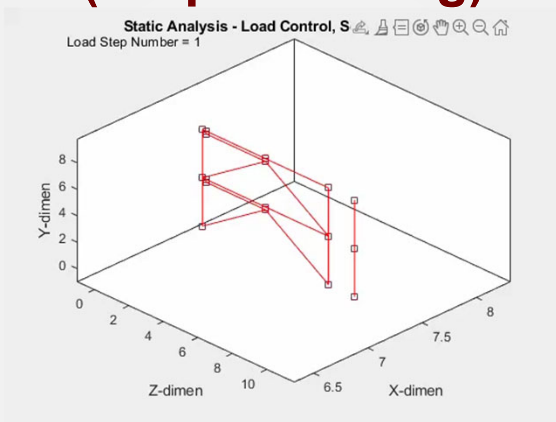
```
Static Displacement Animation Configuration:
    scale factor                                : 100.00
    step interval for decimating data in animation: 1
    pause for playback of animation             : 0.500
Animation configuration (scale factor, decimation, pause) okay? [y]/n:
```

- Used to set scale factor, step interval and speed. Automatically set. Type ‘y’

```
Axes ranges for animation plot (based on HyCom-3D: Y-axis in vertical elevation, Right-hand Rule applies):
    axis range values, x-axis                  : 6.320000, 8.320000
    axis range values, y-axis                  : -1.000000, 9.784000
    axis range values, z-axis                  : -1.000000, 11.608000
Axes ranges okay? [y]/n:
```

- Asks if the axis are set correctly. Done automatically. type ‘y’
- After typing ‘y’ it will ask if you want to store a video of the animation

What happens after pressing run button (Preprocessing)



Proceed for eigenvalue analysis

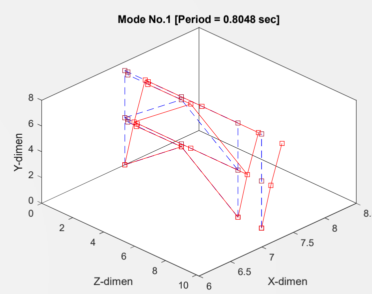
Command Window

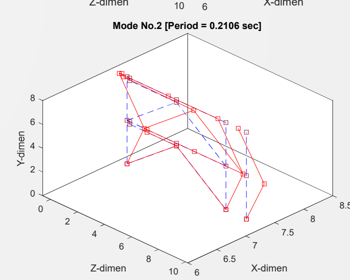
```
[FEM] Configuration    : 2D_Steel_Frame_With_Dampers.tcl
[FEM] Structure
[FEM]  Nodes              : 17
[FEM]  Elements           : 20
[FEM]  Materials          : 2
[FEM]  Sections           : 2
[FEM]  Restrained DOFs    : 64
[FEM]  Slaved DOFs        : 2
[FEM]  Free DOFs          : 36
[FEM]  Gravity Nodes      : 4
[FEM] View Mode shapes and Periods? y/[n]:
```

- Asks if you want to visualize modeshapes
- y or n

```
[FEM] View Mode shapes and Periods? y/[n]: y
Enter number of modes shapes to view =
```

- If you press y. How many modes to be visualized. Since we have lumped mass at only two nodes. Therefore only 2 modes are relevant





Proceed to do transient dynamic analysis

```
Command Window
[FEM] View Mode shapes and Periods? y/[n]: y
Enter number of modes shapes to view = 2
[FEM] X-Direction EQ History      : EQRec.txt
[FEM] Z-Direction EQ History      : EQRec.txt
[FEM] EQ Scale Factor : 9.810000e+00
[FEM] Integration algorithm: MKRAlpha
[FEM] Integrator type: ExplicitAlpha
[FEM] Steps          : 6001
[FEM] Timestep       : 0.004883 seconds
[FEM] Target         : Matlab
[FEM] Hit Enter to begin numerical simulation
```

- Hit enter to start transient dynamic analysis

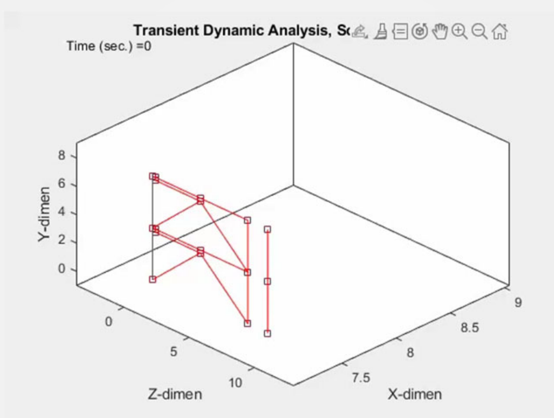
```
Command Window
Dyanamic Analysis Step 27/ of 6001
Dyanamic Analysis Step 28/ of 6001
Dyanamic Analysis Step 29/ of 6001
Dyanamic Analysis Step 30/ of 6001
Dyanamic Analysis Step 31/ of 6001
Dyanamic Analysis Step 32/ of 6001
Dyanamic Analysis Step 33/ of 6001
Dyanamic Analysis Step 34/ of 6001
Dyanamic Analysis Step 35/ of 6001
Dyanamic Analysis Step 36/ of 6001
Dyanamic Analysis Step 37/ of 6001
```

- Shows the progress of dynamic analysis

- Once the analysis is done. You can visualize the dynamic response.
- Similar to what was shown for static analysis

LEHIGH UNIVERSITY | ATLSS | RTMD REAL-TIME MULTI-DIRECTIONAL SIMULATION | NSF 75 YEARS OF DISCOVERY & INNOVATION | NSF NHERI CYBER-PHYSICAL SIMULATION

Proceed to do transient dynamic analysis



LEHIGH UNIVERSITY | ATLSS | RTMD REAL-TIME MULTI-DIRECTIONAL SIMULATION | NSF 75 YEARS OF DISCOVERY & INNOVATION | NSF NHERI CYBER-PHYSICAL SIMULATION

Post-process transient dynamic analysis data

- The data from transient dynamic analysis will be stored into a workspace file

```
%%
filesave = 'NoDampers_MCE.mat';
save(filesave,'Am_Out_Displacement','Am_Out_Velocity','Am_Out_Acceleration','Am_Out_RestoringForce',...
    'Am_Out_ElementResponse',"Am_Out_Steps",'time','Structure','Nodes','Materials','Elements','Sections')
```

- A script is provided to you for post processing the data from transient dynamic analysis

PostProcess_MATLAB.m

```
% Code developed by Faisal Nissar Malik
% Use this script file as a reference to post process data obtained from HyCoM
% For Post Processing Data from Simulink, a different file is provided

% Load data
load('NoDampers_MCE.mat');

%%
% Displacement, Velocity, and Acceleration
Node = 8;
DOF = 'Horz'; % use Horz for Translation, Vert for Vertical and Rot for rotational

Displacement = get_disp_accel_vel(Node,DOF,Am_Out_Displacement,Nodes);
Velocity = get_disp_accel_vel(Node,DOF,Am_Out_Velocity,Nodes);
Acceleration = get_disp_accel_vel(Node,DOF,Am_Out_Acceleration,Nodes);

figure('Units','normalized','Position',[0.1 0.1 0.6 0.3])
plot(time,Displacement)
xlabel('Time (sec)'); ylabel('Displacement (m)')
grid on; box on;
set(gca,'fontsize',14)
title(sprintf('%s Displacement of Node %d',DOF,Node))
```

To obtain the node displacement, velocity or acceleration

© Ricles and Malik, 2025

Post-process transient dynamic analysis data

- The data from transient dynamic analysis will be stored into a workspace file

```
%%
filesave = 'NoDampers_MCE.mat';
save(filesave,'Am_Out_Displacement','Am_Out_Velocity','Am_Out_Acceleration','Am_Out_RestoringForce',...
    'Am_Out_ElementResponse',"Am_Out_Steps",'time','Structure','Nodes','Materials','Elements','Sections')
```

- A script is provided to you for post processing the data from transient dynamic analysis

PostProcess_MATLAB.m

```
%%
% Get Element Response
% Force Based Fiber Elements --> Tag = 1,2,3,4,5,6,7,8
Element = 2; Section = 1;
ElementIDX = find([Am_Out_ElementResponse.ID] == Element, 1);

Moment = Am_Out_ElementResponse(ElementIDX).sections(Section).force(:,2);
AxialForce = Am_Out_ElementResponse(ElementIDX).sections(Section).force(:,1);
Curvature = Am_Out_ElementResponse(ElementIDX).sections(Section).deformation(:,2);
AxialDeformation = Am_Out_ElementResponse(ElementIDX).sections(Section).deformation(:,1);
RestoringForce = Am_Out_ElementResponse(ElementIDX).globalForce;

figure('Units','normalized','Position',[0.1 0.1 0.4 0.3])
plot(Curvature,Moment)
xlabel('Curvature (1/m)');
ylabel('Moment (kNm)')
grid on; box on;
set(gca,'fontsize',14)
title(sprintf('Moment Curvature for Element %d at Section %d',Element,Section))
```

To obtain the section response/ global force of FBE element (1-8)

Post-process transient dynamic analysis data

- The data from transient dynamic analysis will be stored into a workspace file

```
%%
filesave = 'NoDampers_MCE.mat';
save(filesave,'Am_Out_Displacement','Am_Out_Velocity','Am_Out_Acceleration','Am_Out_RestoringForce',...
    'Am_Out_ElementResponse','Am_Out_Steps','time','Structure','Nodes','Materials','Elements','Sections')
```

- A script is provided to you for post processing the data from transient dynamic analysis

PostProcess_MATLAB.m

```
%%
% Get restoring force force for braces
% Braces, Tags = 12,13, 16,17
Element = 12;
ElementIDX = find([Am_Out_ElementResponse.ID] == Element, 1);
RestoringForce = Am_Out_ElementResponse(ElementIDX).globalForce;
```

To obtain the global force of the diagonal brace elements

```
%%
% Dampers, Tags = 19, 20
%if NumDampers=0; Damper force will be zero
Element = 19;
ElementIDX = find([Am_Out_ElementResponse.ID] == Element, 1);

DamperForce = Am_Out_ElementResponse(ElementIDX).basicForce;
DamperDeformation = Am_Out_ElementResponse(ElementIDX).basicDeformation;
```

To obtain basic force and deformation of dampers. If you use NumDampers = 0 in your numerical analysis, the recorded damper force will be zero

Post-processing RTHS data

- Each group will go one by one (Group A, B, C and then D) to conduct RTHS in the control room.
- The RTHS data will be placed in your Workstation and a script file is provided to post-process the data
- The script is similar to the script provided for post-processing the transient dynamic analysis data (i.e., from MATLAB simulation)

PostProcessRTHSData.m

```
clc;clear; close all;
file = 'RTHSData_Soil_MCE.mat';
load(file);
% Note: Data from structure is recorded at 5/1024 s and from DAQ is recorded at 1/1024 s
%%
% Get the Data from DAQ -- i.e., Recorded for the damper and UKF

%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
% Calulated coefficients of LuGre Model from the UKF
LuGre_Sigma0 = RTHSDAQRecordedData.UKFSigma0;
LuGre_Sigma1 = RTHSDAQRecordedData.UKFSigma1;
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%

% ATS coefficients -- Not Needed %%%%%%%%
% ATS_a0 = RTHSDAQRecordedData.ATS_a0;
% ATS_a1 = RTHSDAQRecordedData.ATS_a1;
% ATS_a2 = RTHSDAQRecordedData.ATS_a2;

%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
% Get Damper deformation Story 2
ExperimentalDamperDeformationStory2 = RTHSDAQRecordedData.DamperDisplacement;
% Experimental Damper force and LuGre Damper force for Story2
ExperimentalDamperForceStory2 = RTHSDAQRecordedData.DamperForce;
UKFLuGre_DamperForceStory2 = RTHSDAQRecordedData.DamperForceStory2;
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%

%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
% Calculated Damper force for Story 1
UKFLuGre_DamperForceStory1 = RTHSDAQRecordedData.DamperForceStory1;
% Damper Deformation for Story 1
DamperDeformationStory1 = A_xpcDispData.data(:,Nodes(16).UZ)-A_xpcDispData.data(:,Nodes(17).UZ);
DamperDeformationStory1 = interp1(StructralRecorderTime,DamperDeformationStory1,RTHSDAQRecordedData.DAQRecorderTime);
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
```

Process data from the DAQ which includes data of experimental dampers and dampers modeled by OMU

Post-processing RTHS data

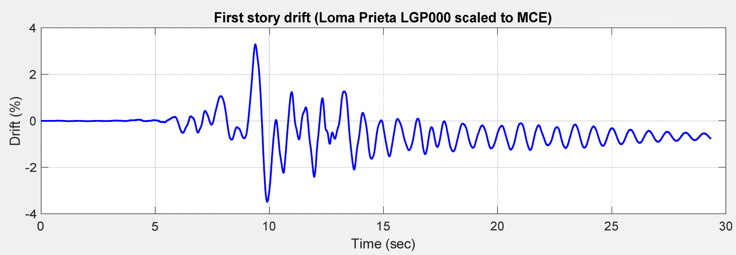
- Each group will go one by one (Group 1, 2 and then 3) to conduct RTHS in the control room.
- The RTHS data will be placed in your Workstation and a script file is provided to post-process the data
- The script is similar to the script provided for post-processing the transient dynamic analysis data (i.e., from MATLAB simulation)

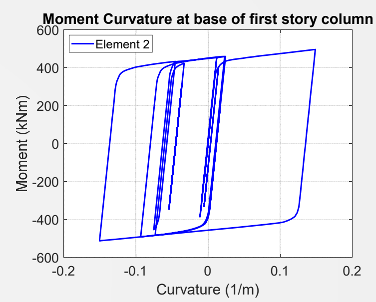
PostProcessRTHSData.m

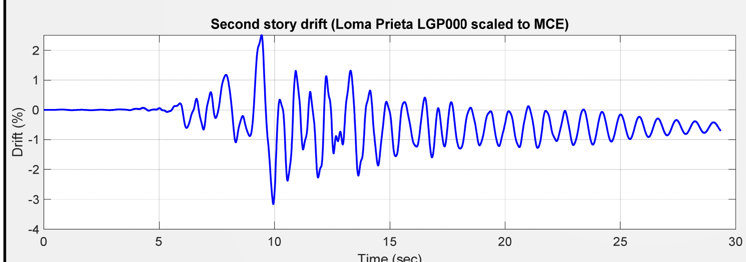
```
%%
% Axial Force, Moment, and Restoring Force From Force-based Fiber Elements
% For FBE Elements Element Tags are -- 1,2,3,4,5,6,7,8
%%%%%%%%%%%%%%%%%% Modify the element number and section ID %%%%%%%%%%
Element = 1; Section = 1;                                              %
%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%%
Moment = A_xpcElementResponse{Element}{Section}.force(2,:);
AxialForce = A_xpcElementResponse{Element}{Section}.force(1,:);
AxialDeformation = A_xpcElementResponse{Element}{Section}.deformation(1,:);
Curvature = A_xpcElementResponse{Element}{Section}.deformation(2,:);
RestoringForce = A_xpcElementRestoringForce{Element};

figure('Units','normalized','Position',[0.1 0.5 0.4 0.4])
plot(Curvature,Moment)
xlabel('Curvature (1/m)');
ylabel('Moment (kNm)')
grid on; box on;
set(gca,'fontsize',14)
title(sprintf('Moment Curvature for Element %d at Section %d',Element,Section))
```

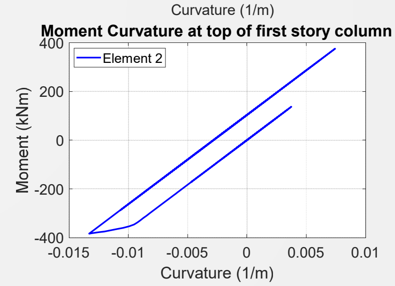
Element level data process is similar with a slight modification. It does not use the command
ElementIDX = find([Am_Out_ElementResponse.ID] == Element, 1);
Rather, it uses the element number to identify the element.
This is due to slightly different structures of the data stored in MATLAB and RTHS

Results – Case with no dampers

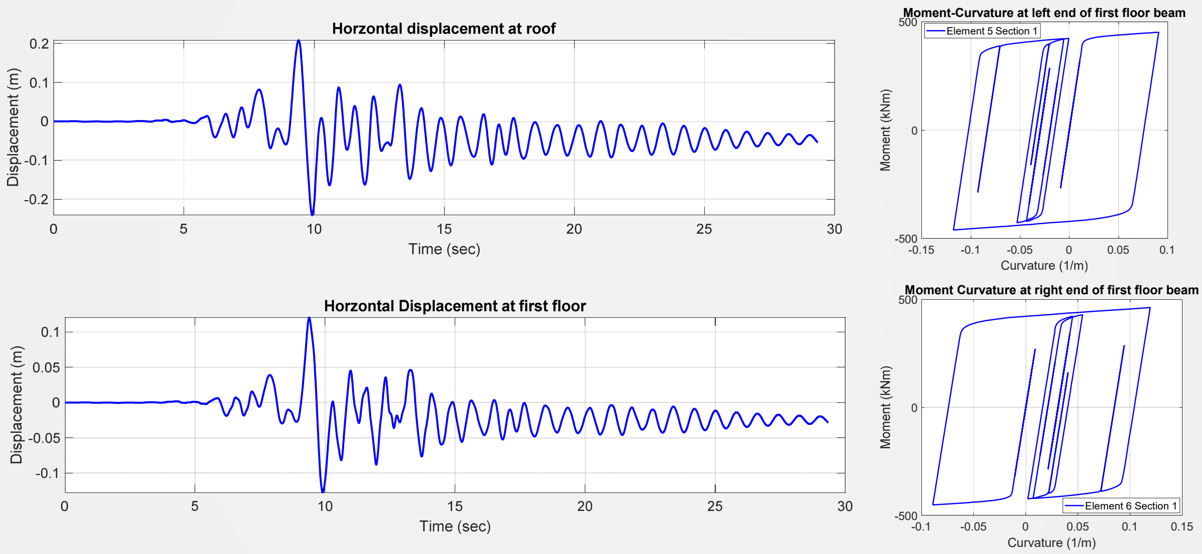




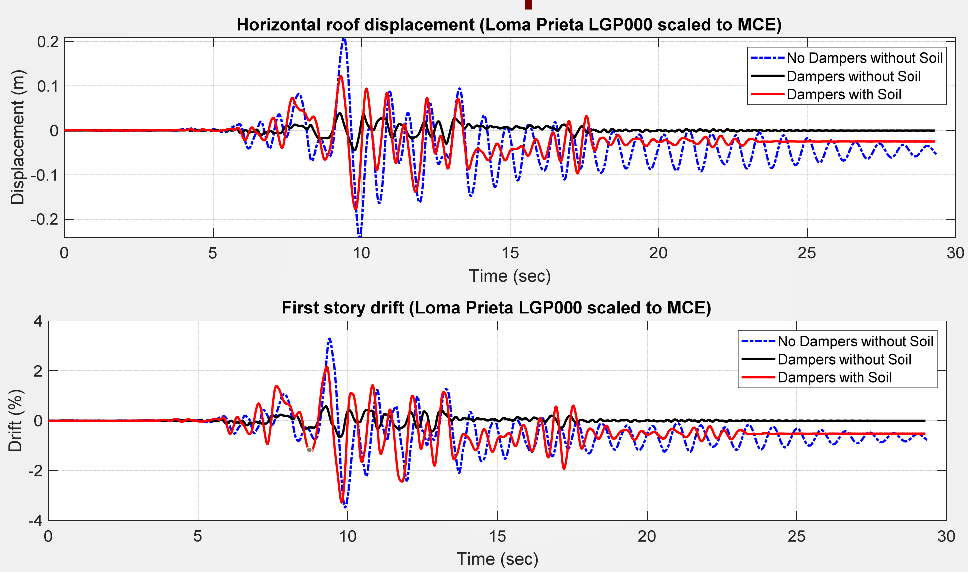




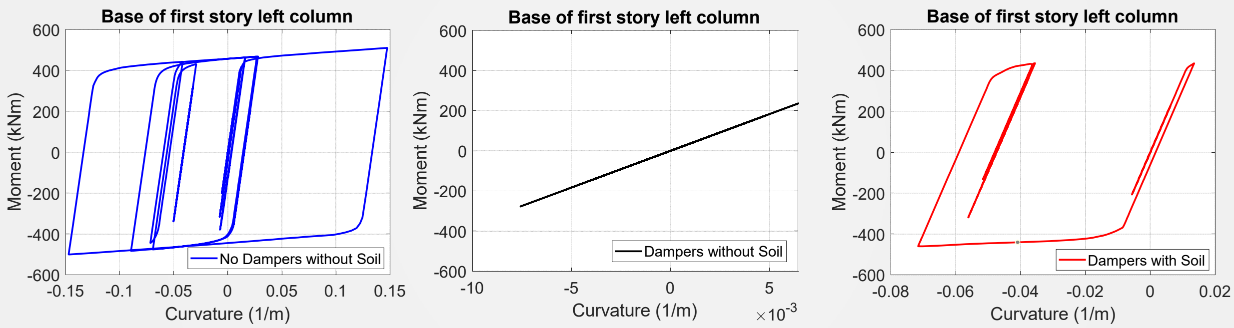
Results – Case with no dampers



RTHS Results: Displacements & Drifts

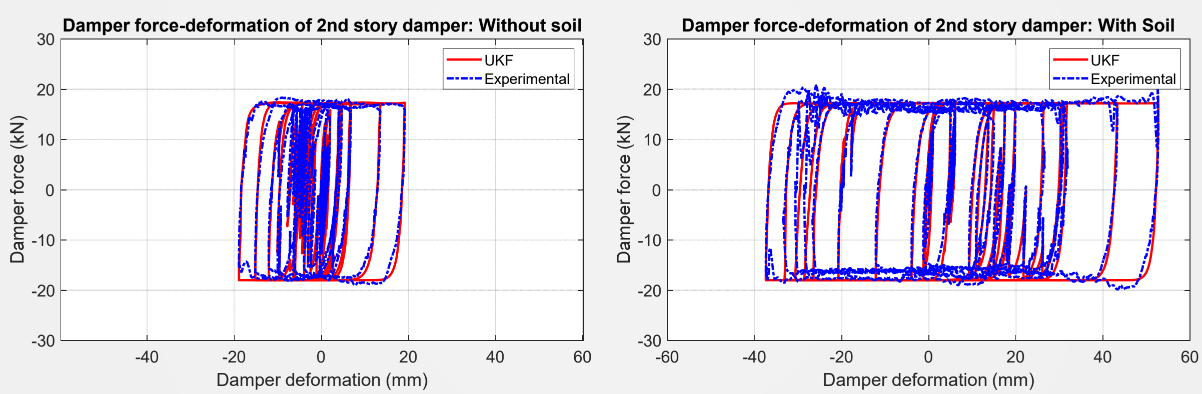


RTHS Results: Moment-Curvature



Loma Prieta (LGP000) scaled to MCE hazard level

RTHS Results: Damper-force deformation



Loma Prieta (LGP000) scaled to MCE hazard level

Session 5: Groups Perform RTHS

Group Presentation Guidelines

- 15-minute duration
- Present results and an assessment of them from RTHS case study
  - ➢ What is the efficacy of the dampers in improving performance of the structural system?
  - ➢ What is the effect of soil-foundation-interaction on the structural system's performance?
  - ➢ What is the difference in performance under the different seismic hazard levels?
- Consider evaluating global response and local (member) response.
- Include photographs of test setup and control room; plots of data; animations of structure.
- Summarize your assessment of the performance; draw conclusions, supporting them with plots and any animations.
- Suggest looking at:
  - Floor accelerations
  - Floor displacements
  - Inter-story drifts
  - Member level response – Moment curvature at ends of beams and columns
  - Damper force-deformation response (for RTHS results)

Short Course Assignment Logistics

All groups will proceed to the Life Cycle Computational Lab (1:30 pm to 4:30 pm):

1. Create HyCom-3D input file
   - Run HyCom-3D in MATLAB script form
     - DBE of structure without dampers and no SFSI
     - MCE of structure without dampers and no SFSI
2. Each group, when called, will conduct 4 RTHS in the NEES Control Room
   - DBE of structure with dampers and no SFSI
   - DBE of structure with dampers and with SFSI
   - MCE of structure with dampers and no SFSI
   - MCE of structure with dampers and with SFSI
3. Each group post-process data and prepare their group presentation

At 4:30 pm all groups will return to B-101 to make their 15-minute presentation.

Session 6: Groups Presentations