Real ESSI Simulator
Three Day Short Course Examples

Yuan Feng and Boris Jeremić
University of California, Davis, CA

Version: July 28, 2020, 17:44
http://real-essi.us/
This document is an excerpt from: http://sokocalo.engr.ucdavis.edu/~jeremic/LectureNotes/
Contents

1 Three Day Short Course Examples (2017) 5

1.1 Nonlinear Analysis Steps .......................................................... 6
  1.1.1 Free Field 1C ................................................................. 6
  1.1.2 Free Field 3C ................................................................. 10
  1.1.3 Soil-Foundation Interaction 3D ........................................... 14
  1.1.4 Soil-Structure Interaction 3D ............................................. 17
  1.1.5 Analysis of a Structure without Soil .................................. 24
    Eigen Analysis ................................................................. 24
    Imposed Motion .............................................................. 26

1.2 Day 1: Overview ....................................................................... 29
  1.2.1 Nuclear Power Plant with 3C motions from SW4 ................. 29
  1.2.2 Nuclear Power Plant with 1C motions from Deconvolution ..... 31
  1.2.3 Nuclear Power Plant with 3×1C motions from Deconvolution ... 33
  1.2.4 Single Element Models: Illustration of the Elastic-Plastic Behavior 35
  1.2.5 Pushover for Nonlinear Frame ........................................... 37
  1.2.6 Pre-Processing examples with Gmsh ................................. 39
    Cantilever Example ............................................................ 39
    Brick-shell-beam Example ................................................. 40
    DRM 2D Example ............................................................... 41
    DRM 3D Example ............................................................... 42
  1.2.7 Post-processing examples with ParaView ............................ 43
    Slice Visualization .............................................................. 43
    Stress Visualization ........................................................... 44
    Pore Pressure Visualization with upU Element ......................... 45
    Eigen Visualization ........................................................... 46
  1.2.8 Check Model and Visualization of Boundary Conditions ........ 47
1.2.9 Restart Simulation
- Restart in the next stage
- Restart inside the stage

1.3 Day 2: Seismic Motions
1.3.1 Deconvolution and Propagation of 1C Motions, 1D Model
1.3.2 Convolution and Propagation of 1C Motions, 1D Model
1.3.3 Convolution, Deconvolution and Propagation of 1C Motions, 2D Model
1.3.4 Deconvolution 3×1C Motions
1.3.5 Mesh Dependence of Wave Propagation Frequencies
1.3.6 Application of 3C Motions from SW4

1.4 Day 3: Inelastic, Nonlinear Analysis
1.4.1 Single Element Models: Illustration of the Elastic-Plastic Behavior
- von-Mises Perfectly Plastic Material Model
- von-Mises Armstrong-Frederick Material Model
- von-Mises G/Gmax Material Model
- Drucker-Prager Perfectly Plastic Material Model
- Drucker-Prager Armstrong-Frederick Non-Associated Material Model
- Drucker-Prager G/Gmax Non-Associated Material Model
1.4.2 Wave Propagation Through Elasto-plastic Soil
1.4.3 Contact Examples
- Axial Behavior: Stress-Based Hard Contact Example
- Axial Behavior: Stress-Based Soft Contact Example
- Shear behavior: Stress-Based Elastic Perfectly Plastic Contact
- Shear behavior: Stress-Based Elastic-Hardening Contact
- Shear behavior: Stress-Based Elastic-Hardening-Softening Contact
- Force Based Contact Example: Base Isolator
1.4.4 Inelastic Frame Pushover
1.4.5 Inelastic Wall Pushover
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.4.6</td>
<td>Viscous Nonlinear behavior</td>
<td>83</td>
</tr>
<tr>
<td>1.4.7</td>
<td>Numerical Damping Example</td>
<td>85</td>
</tr>
<tr>
<td>1.4.8</td>
<td>Realistic Nuclear Power Plant Example with Nonlinearities</td>
<td>87</td>
</tr>
</tbody>
</table>
Chapter 1

Three Day Short Course Examples

(In collaboration with Mr. Yuan Feng)
1.1 Nonlinear Analysis Steps

1.1.1 Free Field 1C

**Elastic Material.** The Real-ESSI input files for elastic example are available [HERE](#). The compressed package of input files is [HERE](#).

The modeling parameters are listed below:

- Elastic Material Properties
  - Mass Density, $\rho$, 2000 kg/m$^3$
  - Shear wave velocity, $V_s$, 500 m/s
  - Young's modulus, $E$, 1.1 GPa
  - Poisson's ratio, $\nu$, 0.1

**Elastoplastic Material.** The Real-ESSI input files for elastoplastic material example are available [HERE](#). The compressed package of input files is [HERE](#).

The modeling parameters are listed below

- von-Mises nonlinear hardening material model
  - Mass density, $\rho$, 2000 kg/m$^3$
  - Shear wave velocity, $V_s$, 500 m/s
  - Young's modulus, $E$, 1.1 GPa
  - Poisson's ratio, $\nu$, 0.1
  - von Mises radius, $k$, 60 kPa
  - Nonlinear kinematic hardening, $H_a$, 30 MPa
  - Nonlinear kinematic hardening, $C_r$, 60
  - Shear strength ($\approx \sqrt{2/3} H_a/C_r$), $S_u$, 408 kPa
  - Isotropic hardening rate, $K_{iso}$, 0 Pa

Results of the simulation are shown in Fig. 1.1.

The time series of simulation results is shown in Fig. 1.3.

The response spectrum of motion is shown in Fig. 1.4.
Figure 1.1: Simulation model.

Figure 1.2: Simulation model.
Figure 1.3: Simulation results: acceleration time series.
Figure 1.4: Simulation results: response spectrum at soil top.
1.1.2 Free Field 3C

Elastic Material. The compressed package of input files for this example is HERE.

The Modeling parameters are listed below:

- Elastic Material Properties
  - Mass density, $\rho$, 2000 kg/m$^3$
  - Shear wave velocity, $V_s$, 500 m/s
  - Young's modulus, $E$, 1.1 GPa
  - Poisson's ratio, $\nu$, 0.1

SIMULATION TIME: With 8 cores on AWS EC2 c4.2xlarge instance, the running time for this example is 5 minutes.

von-Mises Armstrong-Frederick Material. The compressed package of input files is HERE.

The Modeling parameters are listed below:

- von-Mises nonlinear hardening material model
  - Mass density, $\rho$, 2000 kg/m$^3$
  - Shear wave velocity, $V_s$, 500 m/s
  - Young's modulus, $E$, 1.1 GPa
  - Poisson's ratio, $\nu$, 0.1
  - von Mises radius, $k$, 60 kPa
  - Nonlinear kinematic hardening, $H_a$, 30 MPa
  - Nonlinear kinematic hardening, $C_r$, 60
  - Shear strength ($\approx \sqrt{2/3} \frac{H_a}{C_r}$), $S_u$, 408 kPa
  - Isotropic hardening rate, $K_{iso}$, 0 Pa

SIMULATION TIME: With 8 cores on AWS EC2 c4.2xlarge instance, the running time for this example is 17 minutes.

von-Mises G/Gmax Material. The compressed package of input files is HERE.

The Modeling parameters are listed below:

- von-Mises G/Gmax material model
- Mass density, $\rho$, 2000 kg/m$^3$
- Shear wave velocity, $V_s$, 500 m/s
- Young’s modulus, $E$, 1.1 GPa
- Poisson’s ratio, $\nu$, 0.1
- Total number of shear modulus 9
- $G$ over $G_{\text{max}}$, 1, 0.995, 0.966, 0.873, 0.787, 0.467, 0.320, 0.109, 0.063
- Shear strain gamma, 0, 1E-6, 1E-5, 5E-5, 1E-4, 0.0005, 0.001, 0.005, 0.01

SIMULATION TIME: With 8 cores on AWS EC2 c4.2xlarge instance, the running time for this example is 565 minutes.

**Drucker-Prager G/Gmax Material.** The compressed package of input files is [HERE](#).

The Modeling parameters are listed below:

- Drucker-Prager G/Gmax material model
  - Mass density, $\rho$, 2000 kg/m$^3$
  - Shear wave velocity, $V_s$, 500 m/s
  - Young’s modulus, $E$, 1.1 GPa
  - Poisson’s ratio, $\nu$, 0.1
  - Initial confining stress, $p_0$, 100 kPa
  - Reference pressure, $p_{\text{ref}}$, 100 kPa
  - Pressure exponential, $n$, 0.5
  - Cohesion, $c$, 1 kPa
  - Total number of Shear Modulus 9
  - $G$ over $G_{\text{max}}$, 1, 0.995, 0.966, 0.873, 0.787, 0.467, 0.320, 0.109, 0.063
  - Shear strain gamma, 0, 1E-6, 1E-5, 5E-5, 1E-4, 0.0005, 0.001, 0.005, 0.01

SIMULATION TIME: With 8 cores on AWS EC2 c4.2xlarge instance, the running time for this example is 565 minutes.

Results are shown in Fig. 1.56.

SIMULATION TIME: With 8 cores on AWS EC2 c4.2xlarge instance, the running time for this example is 871 minutes.

The time series of simulation results is shown in Fig. 1.7.

The response spectrum of motion is shown in Fig. 1.8.
Figure 1.5: Simulation model.

Figure 1.6: Simulation model.

Figure 1.7: Simulation results: acceleration time series.
Figure 1.8: Simulation results: response spectrum at soil top.
1.1.3 Soil-Foundation Interaction 3D

**Elastic Material.** The compressed package of input files is HERE.

The Modeling parameters are listed below:

- Elastic Material Properties
  - Mass density, $\rho$, 2000 $kg/m^3$
  - Shear wave velocity, $V_s$, 500 $m/s$
  - Young’s modulus, $E$, 1.1 GPa
  - Poisson’s ratio, $\nu$, 0.1

SIMULATION TIME: With 8 cores on AWS EC2 c4.2xlarge instance, the running time for this example is 13 minutes.

**von-Mises Armstrong-Frederick Material.** The compressed package of input files is HERE.

The Modeling parameters are listed below:

- von-Mises nonlinear hardening material model
  - Mass density, $\rho$, 2000 $kg/m^3$
  - Shear wave velocity, $V_s$, 500 $m/s$
  - Young’s modulus, $E$, 1.1 GPa
  - Poisson’s ratio, $\nu$, 0.1
  - von Mises radius, $k$, 60 kPa
  - Nonlinear kinematic hardening, $H_a$, 30 MPa
  - Nonlinear kinematic hardening, $C_r$, 60
  - Shear strength ($\approx \sqrt{2/3} H_a/C_r$), $S_u$, 408 kPa
  - Isotropic hardening rate, $K_{iso}$, 0 Pa

SIMULATION TIME: With 8 cores on AWS EC2 c4.2xlarge instance, the running time for this example is 36 minutes.

**von-Mises G/Gmax Material.** The compressed package of input files is HERE.

The Modeling parameters are listed below:

- von-Mises G/Gmax material model
- Mass density, $\rho$, 2000 $kg/m^3$
- Shear wave velocity, $V_s$, 500 $m/s$
- Young’s modulus, $E$, 1.1 GPa
- Poisson’s ratio, $\nu$, 0.1
- Total number of shear modulus, 9
- $G$ over $G_{max}$, 1, 0.995, 0.966, 0.873, 0.787, 0.467, 0.320, 0.109, 0.063
- Shear strain gamma, 0, 1E-6, 1E-5, 5E-5, 1E-4, 0.0005, 0.001, 0.005, 0.01

SIMULATION TIME: With 8 cores on AWS EC2 c4.2xlarge instance, the running time for this example is 726 minutes.

**Drucker-Prager G/Gmax Material.** The compressed package of input files is [HERE](#).

The Modeling parameters are listed below:

- Drucker-Prager G/Gmax material model
  - Mass density, $\rho$, 2000 $kg/m^3$
  - Shear wave velocity, $V_s$, 500 $m/s$
  - Young’s modulus, $E$, 1.1 GPa
  - Poisson’s ratio, $\nu$, 0.1
  - Initial confining stress, $p_0$, 100 kPa
  - Reference pressure, $p_{ref}$, 100 kPa
  - Pressure exponential, $n$, 0.5
  - Cohesion, $n$, 1 kPa
  - Total number of Shear Modulus, 9
  - $G$ over $G_{max}$, 1, 0.995, 0.966, 0.873, 0.787, 0.467, 0.320, 0.109, 0.063
  - Shear strain gamma, 0, 1E-6, 1E-5, 5E-5, 1E-4, 0.0005, 0.001, 0.005, 0.01

SIMULATION TIME: With 8 cores on AWS EC2 c4.2xlarge instance, the running time for this example is 1252 minutes.

**Contact Elements.** The compressed package of input files is [HERE](#).

The Modeling parameters are listed below:

- Elastic Material Properties
- Mass density, $\rho$, 2000 kg/m$^3$
- Shear wave velocity, $V_s$, 500 m/s
- Young’s modulus, $E$, 1.1 GPa
- Poisson’s ratio, $\nu$, 0.1

SIMULATION TIME: With 8 cores on AWS EC2 c4.2xlarge instance, the running time for this example is 24 minutes.

Both Elastoplastic Material and Contact Elements. The compressed package of input files is HERE.

SIMULATION TIME: With 8 cores on AWS EC2 c4.2xlarge instance, the running time for this example is 41 minutes.

Figure 1.9: Simulation model.

Results of the simulation are shown in Fig. 1.12.

Figure 1.10: Soil foundation interaction results.
1.1.4 Soil-Structure Interaction 3D

**Elastic Material.** The compressed package of input files is [HERE](#).

The Modeling parameters are listed below:

- Elastic Material Properties
  
  - Mass density, $\rho$, 2000 kg/m$^3$
  
  - Shear wave velocity, $V_s$, 500 m/s
  
  - Young's modulus, $E$, 1.1 GPa
  
  - Poisson's ratio, $\nu$, 0.1

SIMULATION TIME: With 8 cores on AWS EC2 c4.2xlarge instance, the running time for this example is 10 minutes.

**von-Mises Armstrong-Frederick Material.** The compressed package of input files is [HERE](#).

The Modeling parameters are listed below:

- von-Mises nonlinear hardening material model
  
  - Mass density, $\rho$, 2000 kg/m$^3$
  
  - Shear wave velocity, $V_s$, 500 m/s
  
  - Young's modulus, $E$, 1.1 GPa
  
  - Poisson's ratio, $\nu$, 0.1
  
  - von Mises radius, $k$, 60 kPa
  
  - Nonlinear kinematic hardening, $H_a$, 30 MPa
  
  - Nonlinear kinematic hardening, $C_r$, 60
  
  - Shear strength ($\approx \sqrt{2/3} H_a/C_r$), $S_u$, 408 kPa
  
  - Isotropic hardening rate, $K_{iso}$, 0 Pa

SIMULATION TIME: With 8 cores on AWS EC2 c4.2xlarge instance, the running time for this example is 46 minutes.

**von-Mises G/Gmax Material.** The compressed package of input files is [HERE](#).

The Modeling parameters are listed below:

- von-Mises G/Gmax material model
– Mass density, $\rho$, 2000 $kg/m^3$
– Shear wave velocity, $V_s$, 500 $m/s$
– Young’s modulus, $E$, 1.1 GPa
– Poisson’s ratio, $\nu$, 0.1
– Total number of shear modulus 9
– $G$ over $G_{max}$, 1, 0.995, 0.966, 0.873, 0.787, 0.467, 0.320, 0.109, 0.063
– Shear strain gamma, 0, 1E-6, 1E-5, 5E-5, 1E-4, 0.0005, 0.001, 0.005, 0.01

SIMULATION TIME: With 8 cores on AWS EC2 c4.2xlarge instance, the running time for this example is 755 minutes.

**Drucker-Prager G/Gmax Material.** The compressed package of input files is [HERE](#).

SIMULATION TIME: With 8 cores on AWS EC2 c4.2xlarge instance, the running time for this example is 1178 minutes.

The Modeling parameters are listed below:

- Drucker-Prager G/Gmax material model
  – Mass density, $\rho$, 2000 $kg/m^3$
  – Shear wave velocity, $V_s$, 500 $m/s$
  – Young’s modulus, $E$, 1.1 GPa
  – Poisson’s ratio, $\nu$, 0.1
  – Initial confining stress, $p_0$, 100 kPa
  – Reference pressure, $p_{ref}$, 100 kPa
  – Pressure exponential, $n$, 0.5
  – Cohesion, $n$, 1 kPa
  – Total number of Shear Modulus 9
  – $G$ over $G_{max}$, 1, 0.995, 0.966, 0.873, 0.787, 0.467, 0.320, 0.109, 0.063
  – Shear strain gamma, 0.0005, 0.001, 0.005, 0.01

SIMULATION TIME: With 8 cores, the running time for this example is 15 minutes.

**Contact Elements.** The compressed package of input files is [HERE](#).

SIMULATION TIME: With 8 cores on AWS EC2 c4.2xlarge instance, the running time for this example is 15 minutes.
Both Elastoplastic Material and Contact Elements. The compressed package of input files is HERE.

The thickness of the shell structure is 2 meters.

Results of the simulation are shown in Fig. 1.12.

SIMULATION TIME: With 8 cores on AWS EC2 c4.2xlarge instance, the running time for this example is 47 minutes.

Simulation with 1C motion. The time series of simulation results is shown in Fig. 1.13.

The response spectrum of motion is shown in Fig. 1.14.

Simulation with 3 × 1C motion. The time series of simulation results is shown in Fig. 1.15.

The response spectrum of motion is shown in Fig. 1.16.
Figure 1.12: Simulation Model.

Figure 1.13: Simulation Results: Acceleration Time Series with 1C motion.
Figure 1.14: Simulation Results: Response Spectrum of Structure Top with 1C motion.
Figure 1.15: Simulation Results: Acceleration Time Series with 3C motion.
Figure 1.16: Simulation Results: Response Spectrum of Structure Top with 3C motion.
1.1.5 Analysis of a Structure without Soil

Eigen Analysis

Eigen analysis of a fixed base structural model should provide a good check of the structural model, natural (eigen) frequencies, and natural (eigen) modes.

The compressed package of input files is HERE.

For this particular example, eigen modes and frequencies are given in Figures 1.18 and 1.19

Figure 1.17: Structure on a fixed based simulation model.

Figure 1.18: Eigen frequencies: $f_1 = 3.47\text{Hz}$, $f_2 = 3.47\text{Hz}$, $f_3 = 6.88\text{Hz}$ (eigen mode 1 to 3 from left to right).

Input files for eigen analysis of the fixed base structure are available at this LINK, and can be directly simulated using Real-ESSI Simulator, http://real-essi.us/, that is available on Amazon Web Services,
Figure 1.19: Eigen frequencies: \( f_4 = 11.50\text{Hz} \) \( f_5 = 11.50\text{Hz} \) \( f_6 = 12.13\text{Hz} \) (eigen modes 4 to 6 from left to right).

Imposed Motion

The Real-ESSI input files for this example are available HERE. The compressed package of input files is HERE.

In addition to eigen analysis, fixed base structural model is used to test response of a fixed base structure. This is important as it provides an opportunity to compare results between different finite element programs, some of which can only model dynamics of fixed base structures.

The simulation model is shown below.

![Simulation Model](image)

Figure 1.20: Simulation Model.

The simulation results:

![Simulation Results](image)

Figure 1.21: Simulation Results.

The time series of simulation results is shown in Fig. 1.22.
The response spectrum of motion is shown in Fig. 1.23.
Figure 1.22: Simulation Results: Acceleration Time Series with 1C imposed motion.
Figure 1.23: Simulation Results: Response Spectrum of Structure Top with 1C imposed motion.
1.2 Day 1: Overview

1.2.1 Nuclear Power Plant with 3C motions from SW4

The Real-ESSI input files for this example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

![Simulation Model](image)

Figure 1.24: Simulation Model.

The Modeling parameters are listed below:

- **Soil**
  - Unit weight, $\gamma$, 21.4 kPa
  - Shear velocity, $V_s$, 500 m/s
  - Young’s modulus, $E$, 1.3 GPa
  - Poisson’s ratio, $\nu$, 0.25
  - Shear strength, $S_u$, 650 kPa
  - von Mises radius, $k$, 60 kPa
  - Kinematic hardening, $H_a$, 30 MPa
  - Kinematic hardening, $C_r$, 25

- **Structure**
  - Unit weight, $\gamma$, 24 kPa
- Young’s modulus, $E$, 20 GPa
- Poisson’s ratio, $\nu$, 0.21

The input motion at the bottom is a 3C wave from SW4.

SIMULATION TIME: With 32 cores on AWS EC2 c4.8xlarge instance, the running time for this example is 17 hours.
1.2.2 Nuclear Power Plant with 1C motions from Deconvolution

The Real-ESSI input files for this example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

![Simulation Model](image1)

Figure 1.25: Simulation Model.

The input motion at the bottom is the deconvolution of the Northridge earthquake records.

![Motion Deconvolution](image2)

Figure 1.26: Motion Deconvolution.

The Modeling parameters are listed below:

- **Soil**
  - Unit weight, $\gamma$, 21.4 kPa
  - Shear velocity, $V_s$, 500 m/s
  - Young’s modulus, $E$, 1.3 GPa
  - Poisson’s ratio, $\nu$, 0.25
  - Shear strength, $S_u$, 650 kPa
- von Mises radius, $k$, $60$ kPa
- kinematic hardening, $H_a$, $30$ MPa
- kinematic hardening, $C_r$, $25$

- Structure
  - Unit weight, $\gamma$, $24$ kPa
  - Young's modulus, $E$, $20$ GPa
  - Poisson's ratio, $\nu$, $0.21$
1.2.3 Nuclear Power Plant with $3 \times 1C$ motions from Deconvolution

The Real-ESSI input files for this example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

![Simulation Model](image)

Figure 1.27: Simulation Model.

The input motion at the bottom is the deconvolution of the Northridge earthquake records.

![Acceleration Deconvolution](image)

Figure 1.28: Acceleration Deconvolution, from left to right in x, y, z directions respectively.

![Displacement Deconvolution](image)

Figure 1.29: Displacement Deconvolution, from left to right in x, y, z directions respectively.

The Modeling parameters are listed below:
• Soil
  – Unit weight, $\gamma$, 21.4 kPa
  – Shear velocity, $V_s$, 500 m/s
  – Young’s modulus, $E$, 1.3 GPa
  – Poisson’s ratio, $\nu$, 0.25
  – Shear strength, $S_u$, 650 kPa
  – von Mises radius, $k$, 60 kPa
  – kinematic hardening, $H_a$, 30 MPa
  – kinematic hardening, $C_r$, 25

• Structure
  – Unit weight, $\gamma$, 24 kPa
  – Young’s modulus, $E$, 20 GPa
  – Poisson’s ratio, $\nu$, 0.21
1.2.4 Single Element Models: Illustration of the Elastic-Plastic Behavior

The compressed package of Real-ESSI input files for this example with von-Mises material model are available HERE.

The compressed package of Real-ESSI input files for this example with Drucker-Prager material model are available HERE.

The Modeling parameters are listed below:

- von-Mises linear hardening material model
  - Mass Density, $\rho$, 0.0 $\text{kg/m}^3$
  - Young’s modulus, $E$, 20 MPa
  - Poisson’s ratio, $\nu$, 0.0
  - von Mises radius, $k$, 100 kPa
  - kinematic hardening rate, $K_{\text{kin}}$, 2 MPa
  - isotropic hardening rate, $K_{\text{iso}}$, 0 Pa

- Drucker-Prager nonlinear hardening material model
  - Mass Density, $\rho$, 0.0 $\text{kg/m}^3$
  - Young’s modulus, $E$, 20 MPa
  - Poisson’s ratio, $\nu$, 0.0
  - Drucker-Prager, $k$, 0.179527
  - nonlinear kinematic hardening, $H_a$, 20 MPa
  - nonlinear kinematic hardening, $C_r$, 100
  - isotropic hardening rate, $K_{\text{iso}}$, 0 Pa
  - initial confining stress, $p_0$, 1 Pa

Inelastic/nonlinear material behavior is shown in Fig. 1.31.
Figure 1.30: Simulation Model of Single Element.

Figure 1.31: Inelastic/Nonlinear material behavior.
1.2.5 Pushover for Nonlinear Frame

The Real-ESSI input files for this example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

![Model for pushover simulation and the cross section of fiber beam (concrete and reinforcement).](image1)

Figure 1.32: Model for pushover simulation and the cross section of fiber beam (concrete and reinforcement).

Result are shown in Fig. 1.33.

![Results for fiber beam pushover.](image2)

Figure 1.33: Results for fiber beam pushover.

The Modeling parameters are listed below:

- Uniaxial concrete
  - Compressive strength, 24 MPa
  - Strain at compressive strength, 0.001752
  - Crushing strength, 0.0 Pa
Figure 1.34: Boundary condition $u_x$ for fiber beam pushover.

- Strain at compressive strength, 0.003168
- lambda, 0.5
- Tensile strength, 0 Pa
- Tension softening stiffness, 0 Pa

- Uniaxial steel

  - Yield strength, 413.8 MPa
  - Young's modulus, 200 GPa
  - Strain hardening ratio, 0.01
  - R0, 18.0
  - cR1, 0.925
  - cR2, 0.15
  - a1, 0.0
  - a2, 55.0
  - a3, 0.0
  - a4, 55.0
1.2.6 Pre-Processing examples with Gmsh

Cantilever Example

The Real-ESSI input files for this example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

![Simulation Model Cantilever](image1)

Figure 1.35: Simulation Model Cantilever.

Results are shown in Fig. 1.36.

![Simulation Model. Cantilever, results](image2)

Figure 1.36: Simulation model. cantilever, results.
Brick-shell-beam Example

The Real-ESSI input files for this example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

Figure 1.37: Simulation Model Brick-Shell-Beam.

Results are shown in Fig. 1.38.

Figure 1.38: Brick-Shell-Beam, Results.
DRM 2D Example

The Real-ESSI input files for this example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

Figure 1.39: Simulation Model DRM 2D.

Results of free field DRM 2D Model under 1C motion are shown in Fig. 1.40.

Figure 1.40: Simulation Model DRM 2D.
DRM 3D Example

The Real-ESSI input files for this example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

![Simulation Model DRM 3D](image)

Figure 1.41: Simulation Model DRM 3D.

Results of free field DRM 3D Model under 1C motion are shown in Fig. 1.42.

![Simulation Model DRM 2D](image)

Figure 1.42: Simulation Model DRM 2D.
1.2.7 Post-processing examples with ParaView

Slice Visualization

The Real-ESSI input files for this example are available [HERE](#). The compressed package of Real-ESSI input files for this example is available [HERE](#).

![Slice Visualization with ParaView](#)

Figure 1.43: Slice Visualization with ParaView.
Stress Visualization

The Real-ESSI input files for this example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

Figure 1.44: Stress Visualization with ParaView.
Pore Pressure Visualization with upU Element

The Real-ESSI input files for this example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

Figure 1.45: Pore Pressure Visualization with Paraview.
Eigen Visualization

The Real-ESSI input files for this example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

Figure 1.46: Eigen Mode Visualization with Paraview.
1.2.8 Check Model and Visualization of Boundary Conditions

The Real-ESSI input files for this example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

Figure 1.47: Partition Information Visualization with Paraview.

Figure 1.48: Partition Information Visualization with Paraview.
1.2.9 Restart Simulation

Restart in the next stage

The Real-ESSI input files for this example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

![Figure 1.49: Restart Simulation.](image)

This group of examples illustrates the restart functionality between loading stages. There are three test cases in this example. The two loading stages in the first test case is split into two test cases to show the restart feature.

- The first test case run through two loading stages.
- The second test case only run the first loading stage and saves model state at the end.
- The third test case restart the simulation from the saved model state of the second test case. Then, with the restart model state, the test case run the second loading stage only.

Results of the third test case are exactly the same to the first test case.
Restart inside the stage

For the case of lack of convergence, restart with the previous loading stage.

The Real-ESSI input files for this example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

![Figure 1.50: Restart Simulation.](image)

This group of examples illustrate the restart functionality inside one loading stage when the simulation cannot converge in the nonlinear analysis. The nonlinear material model, von-Mises Armstrong-Frederick, is used in all test cases.

There are three test cases in this example.

- The first test case run through the whole simulation with a relatively big tolerance of the unbalanced force.

- The second test case failed in the middle of the simulation with a relatively small tolerance of the unbalanced force. When the second test failed, the model reverted to the last commit model state and saved model state.

- The third test case load the saved model state, increased the tolerance of the unbalanced force, and added the remaining load to the model to continue the simulation.

Results of the third test case are exactly the same to the first test case.

Note that in the third test case only the remaining load should be added to the model. Whenever the new loading stage is used, the previous loading are all finished, which means that the static loading becomes constant and the dynamic loading vanishes.
1.3 Day 2: Seismic Motions

1.3.1 Deconvolution and Propagation of 1C Motions, 1D Model

Various deconvolution and propagation 1D models for one component (1C) wave propagation are provided through links below.

**Note:** Please make sure that the input acceleration record is baseline corrected and the displacement record has no permanent deformation. Otherwise, the unrealistic high frequency components can be brought into the simulation results.

- Deconvolution of Ormsby wavelet, input files are available [HERE](#).
- Deconvolution of Northridge earthquake, input files are available [HERE](#).
- Deconvolution of and DRM propagation of Ormsby wavelet, input files are available [HERE](#).
- Deconvolution of and DRM propagation of Northridge earthquake, input files are available [HERE](#).

1.3.2 Convolution and Propagation of 1C Motions, 1D Model

Various convolution and propagation 1D models for one component (1C) wave propagation are provided through links below:

**Note:** Please make sure that the input acceleration record is baseline corrected and the displacement record has no permanent deformation. Otherwise, the unrealistic high frequency components can be brought into the simulation results.

- Convolution of Ormsby wavelet, input files are available [HERE](#).
- Convolution of Northridge earthquake, input files are available [HERE](#).
- Convolution of and DRM propagation of Ormsby wavelet, input files are available [HERE](#).
- Convolution of and DRM propagation of Northridge earthquake, input files are available [HERE](#).

1.3.3 Convolution, Deconvolution and Propagation of 1C Motions, 2D Model

Various convolution, deconvolution and propagation 2D models for one component (1C) wave propagation are provided through links below.

**Note #1:** Please make sure that the input acceleration record is baseline corrected and the displacement record has no permanent deformation. Otherwise, the unrealistic high frequency components can be brought into the simulation results.
**Note #2:** Please make sure that you develop seismic motions by doing deconvolution and then convolution before analyzing the actual model. File `run.sh` in examples directory has a proper sequence of commands, that is one should first run Real-ESSI on `Deconvolution_DRM_motion.fei` and then, when motions are developed, analyze model.

Examples are available through links below:

- Convolution/Deconvolution of and DRM propagation of Ormsby wavelet, input files are available [HERE](#).
- Convolution/Deconvolution of and DRM propagation of Kobe earthquake records, input files are available [HERE](#).
ESSI 3D building model, deconvolution 1C model, shell model with DRM

The Real-ESSI input files for this example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

The Modeling parameters are listed below:

- **Elastic Soil Material Properties**
  - Mass density, $\rho$, 2000 kg/m$^3$
  - Shear Wave Velocity, $V_s$, 500 m/s
  - Young’s modulus, $E$, 1.1 GPa
  - Poisson’s ratio, $\nu$, 0.1

- **Elastic Structure Material Properties**
  - Mass density, $\rho$, 2500 kg/m$^3$
  - Young’s modulus, $E$, 20 GPa
  - Poisson’s ratio, $\nu$, 0.1

Results of DRM 3D shell Structure Model under 1C motion are shown in Fig. 1.52.
Figure 1.52: Simulation Model.
1.3.4 Deconvolution $3 \times 1$C Motions

Free field 1C model, deconvolution $3 \times 1$C motion, model with DRM

The Real-ESSI input files for this example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

The Modeling parameters are listed below:

- Elastic Material Properties
  - Mass density, $\rho$, 2000 $kg/m^3$
  - Shear Wave Velocity, $V_s$, 500 $m/s$
  - Young’s modulus, $E$, 1.1 GPa
  - Poisson’s ratio, $\nu$, 0.1

Figure 1.53: Simulation Model.

Results of the simulation are shown in Fig. 1.1.
Figure 1.54: Simulation Model.
Free field 3D model, deconvolution 3×1C motion, model with DRM

The Real-ESSI input files for this example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

The Modeling parameters are listed below:

- Elastic Soil Material Properties
  - Mass density, $\rho$, 2000 kg/m$^3$
  - Shear Wave Velocity, $V_s$, 500 m/s
  - Young’s modulus, $E$, 1.1 GPa
  - Poisson’s ratio, $\nu$, 0.1

Results of the simulation are shown in Fig. 1.56.
**ESSI 3D building model, deconvolution 3×1C motion, shell model with DRM**

The Real-ESSI input files for this example are available [HERE](#). The compressed package of Real-ESSI input files for this example is available [HERE](#).

The Modeling parameters are listed below:

- **Elastic Soil Material Properties**
  - Mass density, \( \rho \), \( 2000 \text{ kg/m}^3 \)
  - Shear Wave Velocity, \( V_s \), \( 500 \text{ m/s} \)
  - Young’s modulus, \( E \), \( 1.1 \text{ GPa} \)
  - Poisson’s ratio, \( \nu \), \( 0.1 \)

- **Elastic Structure Material Properties**
  - Mass density, \( \rho \), \( 2500 \text{ kg/m}^3 \)
  - Young’s modulus, \( E \), \( 20 \text{ GPa} \)
  - Poisson’s ratio, \( \nu \), \( 0.1 \)

![Figure 1.57: Simulation Model.](image)

Results of DRM 3D shell Structure Model under 1C motion are shown in Fig. 1.58.
Figure 1.58: Simulation Model.
1.3.5 Mesh Dependence of Wave Propagation Frequencies

The Real-ESSI input files for this example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

Show the mesh dependence of high frequency wave with Ormsby wavelet.

Figure 1.59: Simulation Model.

Results of mesh dependence are shown in Fig. 1.60.
Figure 1.60: Convolution Results and Mesh Dependence.
1.3.6 Application of 3C Motions from SW4

3C Seismic Motion from SW4

A 3C seismic motion field has been developed by using SW4. The characteristic parameters of the seismic motion are given below:

- Geological model: length 3\( km \), width 3\( km \), height 1.7\( km \), grid size 50\( m \), width of super grid damping layer 30\( m \).
- Material model: Elastic material, First 1\( km \): \( V_p = 4630.76 m/s \), \( V_s = 2437.56 m/s \), \( \rho = 2600 kg/m^3 \). 1\( km \sim 1.7 km \): \( V_p = 6000 m/s \), \( V_s = 3464 m/s \), \( \rho = 2700 kg/m^3 \)
- Source type: point moment source, moment seismic moment \( M_{xy} = 5e^{15} N \cdot m \), moment magnitude 4.5.
- Time function: Gaussian function, with dominant frequency 2.5\( Hz \) and maximum frequency 6.5\( Hz \).

The time series displacement and acceleration response at the center of the model is shown below in figure 1.61. And figure 1.62 gives corresponding FFT response.

![Time series response of 3C motion.](image)

During the simulation of SW4, the time series motions at many ESSI nodes (basically are some pre-defined record stations) of an ESSI box (300\( m \times 300 m \times 100 m \)) are recorded and written into SAC files. Then an
transition program SW42ESSI has been developed to interpolate these motions to DRM nodes of localized ESSI model by specifying some geometric translational and rotational transformation, as shown in figure ??.

To launch SW42ESSI, following parameters are needed:

- DRM input: specify the name of DRM input files. This DRM file just contains the geometric information of DRM layer in ESSI model (e.g. DRM node IDs, nodal coordinates, etc).
- SW4 motion directory: specify the output directory of SW4, that contains SAC files.
- origin coordinates of ESSI box (x, y, z): the SW4 coordinates of the origin of ESSI box, i.e. the coordinates of ESSI nodes, whose station ID is (0, 0, 0).
- dimensions of ESSI Box (length, width, height): specify the dimension (length, width and height) of ESSI box.
- spacing of ESSI nodes: specify the grid spacing of ESSI nodes (i.e. motion recording stations)
- interval of time steps for sampling: specify the sampling frequency, if 1 is used here, ESSI simulation time step is the same as the simulation time step of SW4.
- reference point in ESSI model for translational transformation (x, y, z): specify the coordinate of reference point for translational transformation in ESSI model.
• reference point in SW4 model for translational transformation \((x, y, z)\): specify the coordinate of reference point for translational transformation in SW4 model.

• conduct rotational transformation (yes/no): input yes and provide more rotational transformation parameters to enable rotational transformation. If input no, no more parameters are required.

• reference point in SW4 model for rotational transformation \((x, y, z)\): specify the coordinate of reference point for rotational transformation in SW4 model.

• degrees of rotation along three axes \((x, y, z)\): specify the degrees of rotation along three axes. The sign of rotation degrees follows right hand rule.

Figure 1.63: Illustration of transition from SW4 to Real-ESSI.

Free field 3D model, 3C motion, model with DRM

The Real-ESSI input files for this example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

Results of free field DRM 3D Model under 3C motion are shown in figure 1.65.
Figure 1.64: Simulation Model.

Figure 1.65: Simulation of 3D free field model under 3C seismic motion.
ESSI 3D building model, 3C motion, shell model with DRM

The Real-ESSI input files for this example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

Figure 1.66: Simulation Model.
1.4 Day 3: Inelastic, Nonlinear Analysis

1.4.1 Single Element Models: Illustration of the Elastic-Plastic Behavior

von-Mises Perfectly Plastic Material Model.

The Real-ESSI input files for von-Mises perfectly plastic example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

von-Mises Armstrong-Frederick Material Model.

The Real-ESSI input files for von-Mises Armstrong-Frederick example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

The Modeling parameters are listed below:

- **Left:** von-Mises linear hardening material model
  - Mass Density, $\rho$, 0.0 kg/m$^3$
  - Young’s modulus, $E$, 20 MPa
  - Poisson’s ratio, $\nu$, 0.0
  - von Mises radius, $k$, 100 kPa
  - kinematic hardening rate, $K_{\text{kine}}$, 2 MPa
  - isotropic hardening rate, $K_{\text{iso}}$, 0 Pa

- **Right:** Drucker-Prager nonlinear hardening material model
  - Mass Density, $\rho$, 0.0 kg/m$^3$
  - Young’s modulus, $E$, 20 MPa
  - Poisson’s ratio, $\nu$, 0.0
  - Drucker-Prager, $k$, 0.179527
  - nonlinear kinematic hardening, $H_a$, 20 MPa
  - nonlinear kinematic hardening, $C_r$, 100
  - isotropic hardening rate, $K_{\text{iso}}$, 0 Pa
  - initial confining stress, $p_0$, 1 Pa

Results are shown in Fig. 1.68.
Figure 1.67: Simulation Model of Single Element.

Figure 1.68: Simulation Results of Single Element.
von-Mises $G/G_{\text{max}}$ Material Model

The Real-ESSI input files for this example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

The Modeling parameters are listed below:

- von-Mises $G/G_{\text{max}}$ material model
  - Mass density, $\rho$, $2000 \text{ kg/m}^3$
  - Young's modulus, $E$, $200 \text{ MPa}$
  - Poisson's ratio, $\nu$, $0.1$
  - Total number of shear modulus $9$
  - $G$ over $G_{\text{max}}$, $1, 0.995, 0.966, 0.873, 0.787, 0.467, 0.320, 0.109, 0.063$
  - Shear strain gamma, $0, 1\times10^{-6}, 1\times10^{-5}, 5\times10^{-5}, 1\times10^{-4}, 0.0005, 0.001, 0.005, 0.01$

![Figure 1.69: Simulation Model of Single Element.](image-url)
Material Behavior: Stress-Strain

Multi-Yield-Surface vonMises G/Gmax

Damping Plot
Drucker-Prager Perfectly Plastic Material Model

The Real-ESSI input files for this Drucker-Prager perfectly plastic example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

Drucker-Prager Armstrong-Frederick Non-Associated Material Model

The Real-ESSI input files for this Drucker-Prager Armstrong-Frederick example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

![Simulation model, single element.](image)

Figure 1.70: Simulation model, single element.

Results are shown in Fig. 1.71.

![Simulation results for single element.](image)

Figure 1.71: Simulation results for single element.
Drucker-Prager G/Gmax Non-Associated Material Model

The Real-ESSI input files for this example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

The Modeling parameters are listed below:

- Drucker-Prager G/Gmax material model
  - Mass density, \( \rho \), 2000 kg/m\(^3\)
  - Young’s modulus, \( E \), 200 MPa
  - Poisson’s ratio, \( \nu \), 0.1
  - Initial confining stress, \( p_0 \), 100 kPa
  - Reference pressure, \( p_{\text{ref}} \), 100 kPa
  - Pressure exponential, \( n \), 0.5
  - Cohesion, \( n \), 1 kPa
  - Total number of Shear Modulus \( G \) over \( G_{\text{max}} \), 9
  - \( G \) over \( G_{\text{max}} \), 1.0, 0.995, 0.966, 0.873, 0.787, 0.467, 0.320, 0.109, 0.063
  - Shear strain gamma, 0, 1E-6, 1E-5, 5E-5, 1E-4, 0.0005, 0.001, 0.005, 0.01

Results are shown in Fig. 1.73.
Figure 1.73: Simulation Results of Single Element.
1.4.2 Wave Propagation Through Elasto-plastic Soil

The Real-ESSI input files for this example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

Figure 1.74: Wave Propagation through elastoplastic Soils.

The displacement series at the surface are plotted in time and frequency domain.

Figure 1.75: Simulation Results of Wave Propagation.
1.4.3 Contact Examples

Axial Behavior: Stress-Based Hard Contact Example

The Real-ESSI input files for hard contact/interface example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

Axial Behavior: Stress-Based Soft Contact Example

The Real-ESSI input files for soft contact/interface example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

The axial behavior of hard contact/interface and soft contact/interface is illustrated in Fig. 1.76.

![Figure 1.76](image-url)  

Figure 1.76: Simulation results for axial behavior of (left) soft contact/interface and (right) hard contact.
Shear behavior: Stress-Based Elastic Perfectly Plastic Contact

The Real-ESSI input files for the the elastic-perfectly plastic example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

Shear behavior: Stress-Based Elastic-Hardening Contact

The Real-ESSI input files for the elastic-hardening contact/interface example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

Shear behavior: Stress-Based Elastic-Hardening-Softening Contact

The Real-ESSI input files for the elastic-hardening-softening example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

The shear behavior of elastic-perfectly plastic, elastic-hardening plastic, elastic and hardening and softening plastic is illustrated in Fig. 1.77.
Figure 1.77: Simulation results for shear behavior for stress based contact elements: elastic-perfectly plastic, elastic-hardening plastic, elastic, hardening and softening plastic.
Force Based Contact Example: Base Isolator

The Real-ESSI input files for this example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

![Simulation Model](image1)

Figure 1.78: Simulation Model.

Results are show in Fig.1.79.

![Simulation Results](image2)

Figure 1.79: Simulation Results of Contact Examples.
1.4.4 Inelastic Frame Pushover

The Real-ESSI input files for this example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

The Modeling parameters are listed below:

- **Uniaxial concrete**
  - Compressive strength, 24 MPa
  - Strain at compressive strength, 0.001752
  - Crushing strength, 0.0 Pa
  - Strain at compressive strength, 0.003168
  - \( \lambda \), 0.5
  - Tensile strength, 0 Pa
  - Tension softening stiffness, 0 Pa

- **Uniaxial steel**
  - Yield strength, 413.8 MPa
  - Young's modulus, 200 GPa
  - Strain hardening ratio, 0.01
  - \( R_0 \), 18.0
  - \( c_{R1} \), 0.925
  - \( c_{R2} \), 0.15
  - \( a_1 \), 0.0
  - \( a_2 \), 55.0
  - \( a_3 \), 0.0
  - \( a_4 \), 55.0

Result is shown in Fig. 1.81.
Figure 1.80: Model for pushover simulation and the cross section of fiber beam (concrete and reinforcement).

Figure 1.81: Results for fiber pushover.
Figure 1.82: Boundary condition $u_x$ for a fiber beam pushover.
1.4.5 Inelastic Wall Pushover

The Real-ESSI input files for this example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

The Modeling parameters are listed below:

- **Concrete Wall**
  - Young’s modulus, 36.9 GPa
  - Poisson’s ratio, 0.2
  - Tensile yield strength, 5 MPa
  - Compressive yield strength, 56 MPa
  - Plastic deformation rate, 0.4
  - Damage parameter Ap, 0.1
  - Damage parameter An, 1.5
  - Damage parameter Bn, 0.75

- **Uniaxial steel**
  - Yield strength, 457.5 MPa
  - Young’s modulus, 200 GPa
  - Strain hardening ratio, 0.011042
  - a1, 0.0
  - a2, 55.0
  - a3, 0.0
  - a4, 55.0
Figure 1.83: Model for wall element pushover.
1.4.6 Viscous Nonlinear behavior

The Real-ESSI input files for this example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

![Simulation Model](image1)

Figure 1.84: Simulation Model.

Result are shown in Fig. 1.85 and Fig. 1.86.

![Results for low viscous damping](image2)

Figure 1.85: Results for low viscous damping.
Figure 1.86: Results for high viscous.
1.4.7 Numerical Damping Example

The Real-ESSI input files for this example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

![Simulation Model](image)

Figure 1.87: Simulation Model.

Result are shown in Fig. 1.85 and Fig. 1.89.

![Results of low numerical damping](image)

Figure 1.88: Results of low numerical damping.
Figure 1.89: Results of high numerical damping.
1.4.8 Realistic Nuclear Power Plant Example with Nonlinearities

The Real-ESSI input files for this example are available HERE. The compressed package of Real-ESSI input files for this example is available HERE.

The Modeling parameters are listed below:

- **Soil**
  - Unit weight, $\gamma$, 21.4 kPa
  - Shear velocity, $V_s$, 500 m/s
  - Young’s modulus, $E$, 1.3 GPa
  - Poisson’s ratio, $\nu$, 0.25
  - Shear strength, $S_u$, 650 kPa
  - von Mises radius, $k$, 60 kPa
  - kinematic hardening, $H_a$, 30 MPa
  - kinematic hardening, $C_r$, 25

- **Structure**
  - Unit weight, $\gamma$, 24 kPa
  - Young’s modulus, $E$, 20 GPa
  - Poisson’s ratio, $\nu$, 0.21

- **Contact**
  - Initial axial stiffness, $k_{n_{init}}$, 1e9 N/m
  - Stiffening rate, $S_r$, 1000 /m
  - Maximum axial stiffness, $k_{n_{max}}$, 1e12 N/m
  - Shear stiffness, $k_t$, 1e7 N/m
  - Axial viscous damping, $C_n$, 100 N · s/m
  - Shear viscous damping, $C_t$, 100 N · s/m
  - Friction ratio, $\mu$, 0.25

**SIMULATION TIME:** With 32 cores on AWS EC2 c4.8xlarge instance, the running time for this example is 30 hours.
Figure 1.90: Simulation Model.