

# **Practice:**

dynamic rupture in 2D and 2.5D

Huihui Weng

Jean-Paul Ampuero

ICTP, Trieste, Italy, 2-14 Sep.

# Goals

- Learn to simulate dynamic rupture model
- Explore the effects of fault heterogeneities on dynamic ruptures
- Discussion: seismology problems


# 2D and 2.5D numerical codes


## Download from GitHub



**Jean Paul Ampuero**

jpampuero

 California Institute of Technology

 Pasadena, CA, USA

 [Sign in to view email](#)

 <http://www.seismolab.caltech.edu/a...>

**Overview**

Repositories **4**

Projects **0**

Stars **1**

Followers **15**

Following **0**

### Popular repositories

#### **sem2dpack**


SEM2DPACK - A spectral element method for 2D wave propagation and fracture dynamics, with emphasis on computational seismology and earthquake source dynamics.

 Fortran  4  3

#### **specfem3d**




Forked from geodynamics/specfem3d

SPECFEM3D\_Cartesian simulates acoustic (fluid), elastic (solid), coupled acoustic/elastic, poroelastic or seismic wave propagation in any type of conforming mesh of hexahedra (structured or not). I...

 Fortran  1

#### **semlab**

Spectral Element Method for wave propagation and rupture dynamics in Matlab.

 MATLAB  3  1

#### **jpampuero.github.io**

J. P. Ampuero - Seismology and Earthquake Dynamics


 HTML

# 2D and 2.5D numerical codes

## Download from GitHub



**Jean Paul Ampuero**  
jpampuero

 California Institute of Technology

 Pasadena, CA, USA

 [Sign in to view email](#)

<http://www.seismolab.caltech.edu/a...>

## Overview

## Repositories 4

Projects 0

Stars 1

Followers 15

Following 0

~~Popular repositories~~

**sem2dpack**

SEM2DPACK - A spectral element method for 2D wave propagation and fracture dynamics, with emphasis on computational seismology and earthquake source dynamics.

● Fortran ★ 4 🍷 3

semlab

Spectral Element Method for wave propagation and rupture dynamics in Matlab.

● MATLAB ★ 3 🔗 1

specfem3d

Forked from [geodynamics/specfem3d](#)

SPECFEM3D\_Cartesian simulates acoustic (fluid), elastic (solid), coupled acoustic/elastic, poroelastic or seismic wave propagation in any type of conforming mesh of hexahedra (structured or not). I...

● Fortran 1

[jpampuero.github.io](http://jpampuero.github.io)

J. P. Ampuero - Seismology and Earthquake Dynamics

● HTML

The tutorial script will be uploaded to GitHub soon.

MATLAB code **sem1ab**

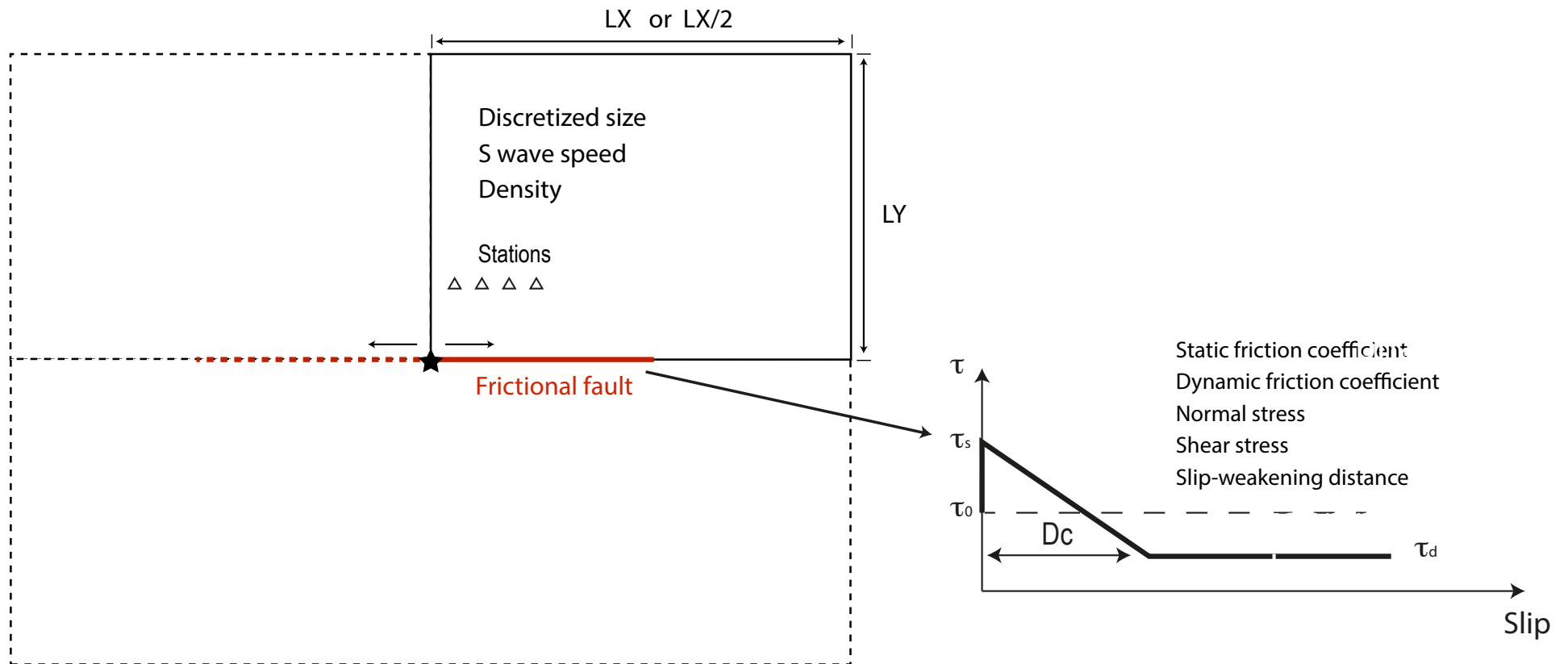
----- run it directly

Fortran code **sem2dpack**

----- installation is simple

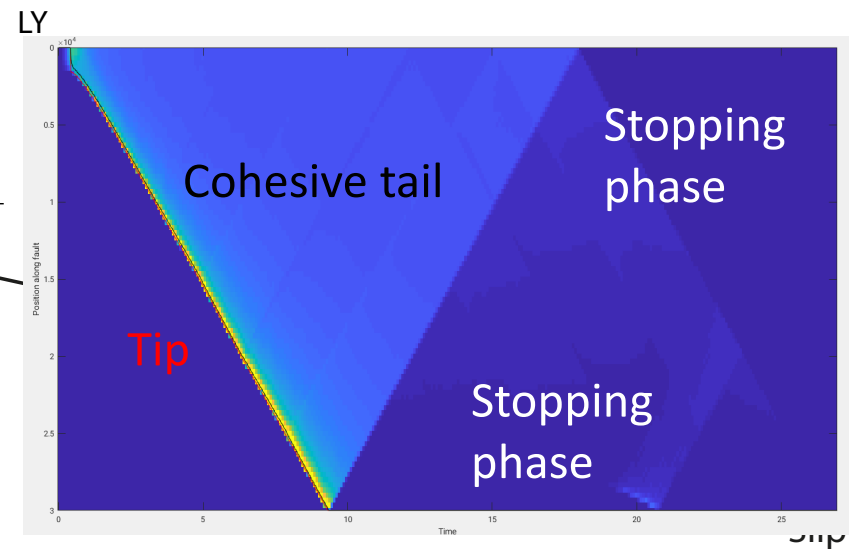
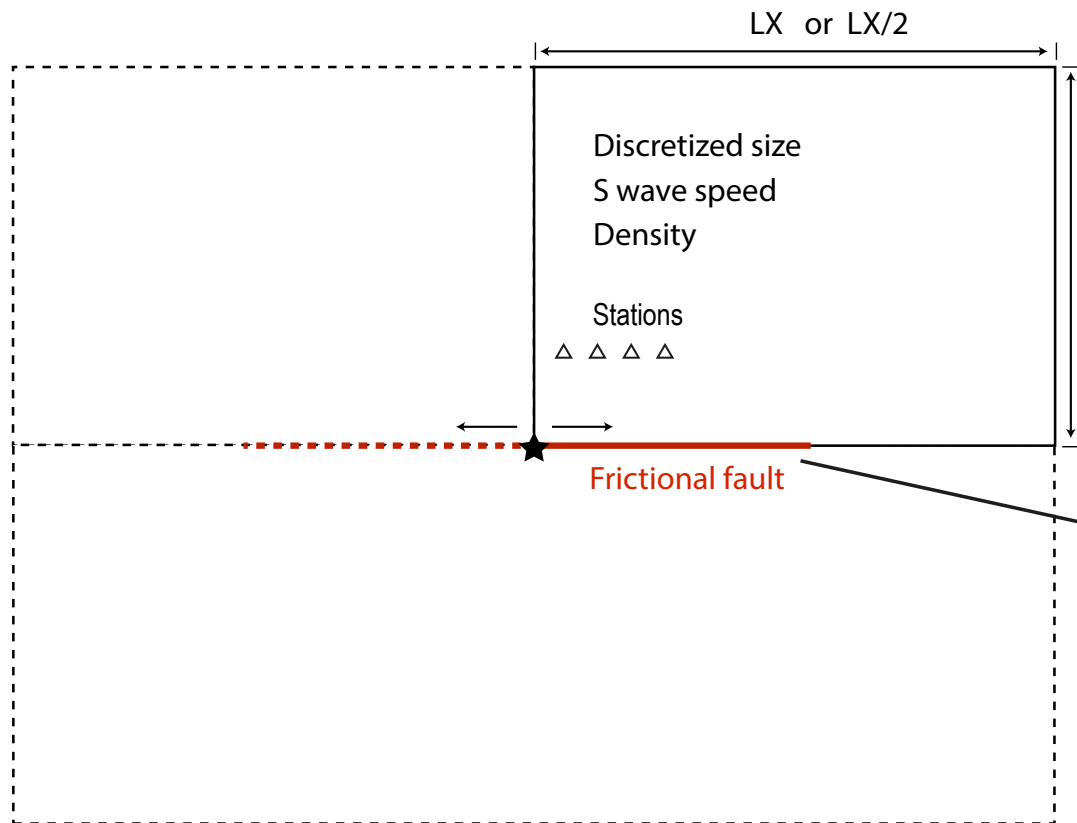
----- simulation is fast

# Model parameters



Second stopping phase

# Model parameters



Second stopping phase

# Model parameters

$$\sigma_{ij,j} = \rho \ddot{u}_i \quad (3 \text{ equations})$$



Reduce to 1 equation

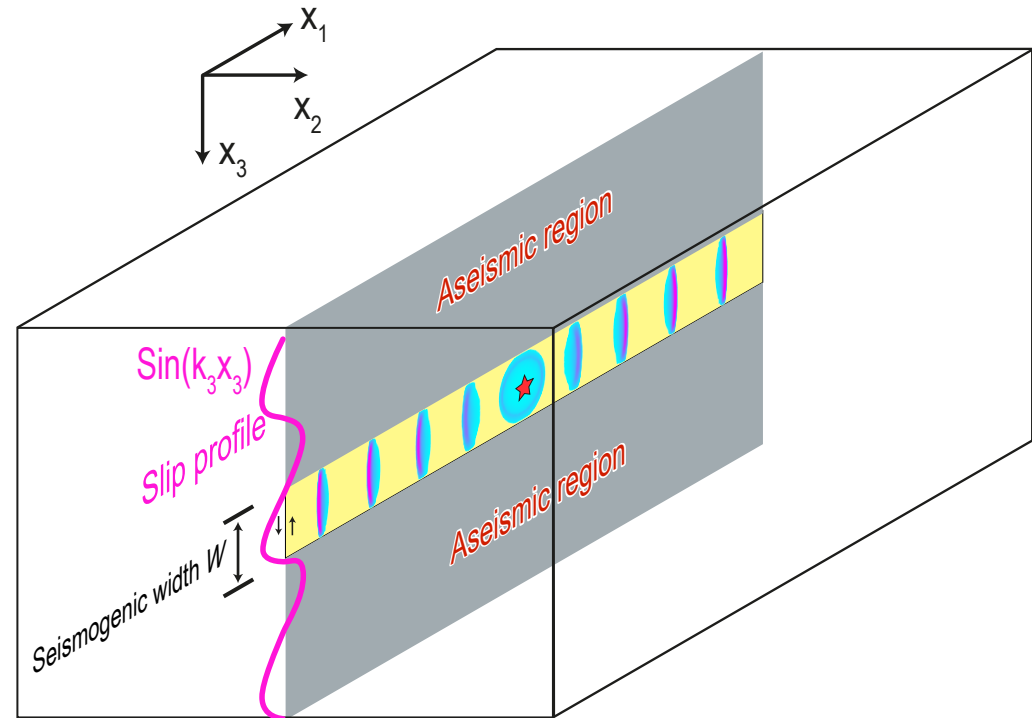
$$\frac{\partial^2 u}{\partial x_1^2} + \frac{\partial^2 u}{\partial x_2^2} + \frac{\partial^2 u}{\partial x_3^2} = \frac{1}{v_s^2} \frac{\partial^2 u}{\partial t^2}$$

Slip approximation

$$u(x_1, x_2, x_3) = u(x_1, x_2, t) e^{ik_3 x_3}$$

$$k_3 = \pi/W$$

$$\frac{\partial^2 u}{\partial x_1^2} + \frac{\partial^2 u}{\partial x_2^2} - k_3^2 u = \frac{1}{v_s^2} \frac{\partial^2 u}{\partial t^2}$$





Tutorial of semlab

Tutorial of sem2dpack

# How to run semlab?

```
cp -r /home/nfs3/seismology_lectures/hweng .
```

```
ls ./hweng
```

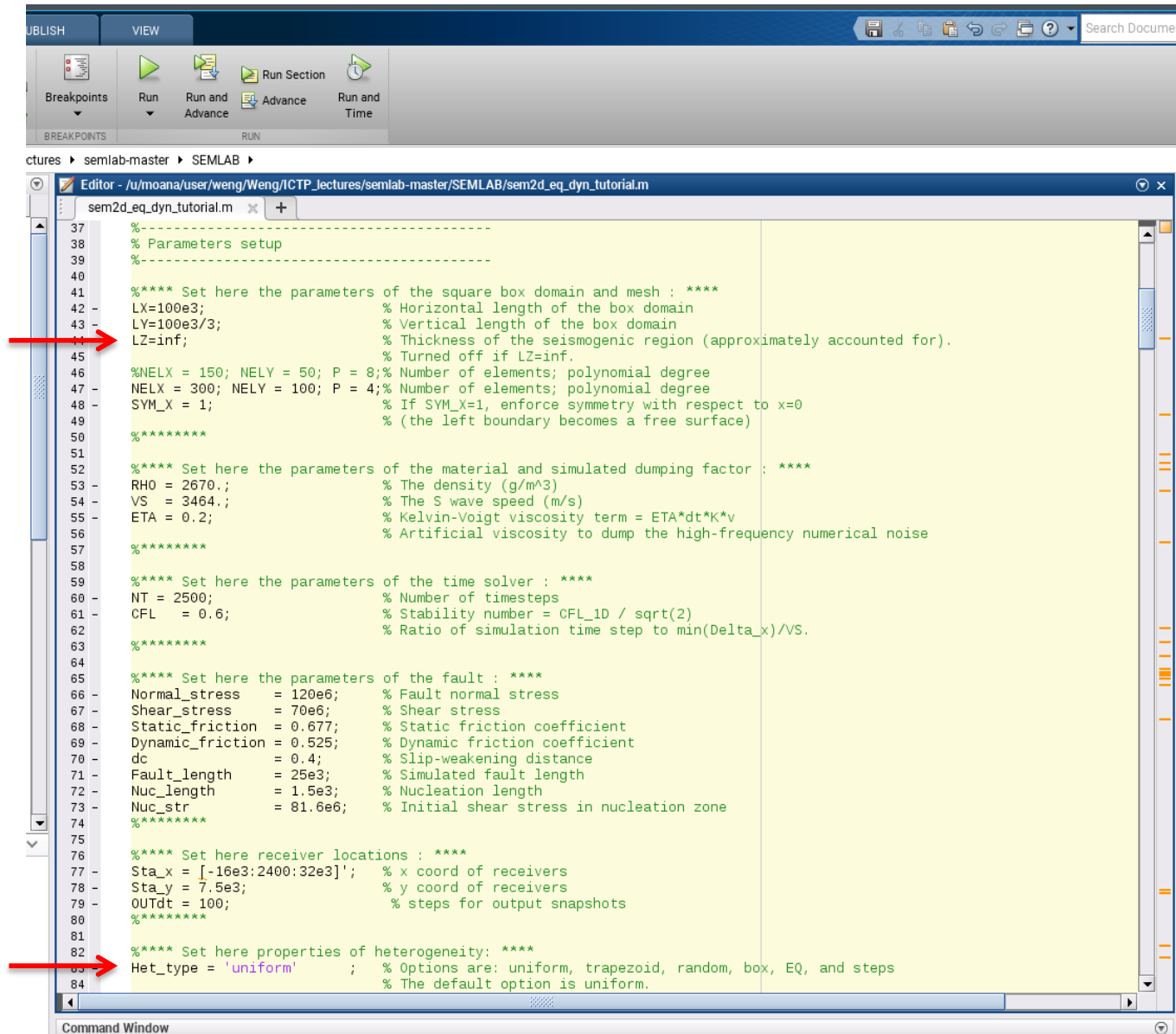
```
cd ./hweng/semlab-master/SEMLAB
```

```
matlab &
```

open this file from matlab:

sem2d\_eq\_dyn\_tutorial.m

# Parameter setup in semlab



```
37 %-----
38 % Parameters setup
39 %-----
40
41 %*** Set here the parameters of the square box domain and mesh : ***
42 LX=100e3; % Horizontal length of the box domain
43 LY=100e3/3; % Vertical length of the box domain
44 LZ=inf; % Thickness of the seismogenic region (approximately accounted for).
45 % Turned off if LZ=inf.
46 %NELX = 150; NELY = 50; P = 8; % Number of elements; polynomial degree
47 NELX = 300; NELY = 100; P = 4; % Number of elements; polynomial degree
48 SYM_X = 1; % If SYM_X=1, enforce symmetry with respect to x=0
49 % (the left boundary becomes a free surface)
50 %*****
51
52 %*** Set here the parameters of the material and simulated dumping factor : ***
53 RHO = 2670.; % The density (g/m^3)
54 VS = 3464.; % The S wave speed (m/s)
55 ETA = 0.2; % Kelvin-Voigt viscosity term = ETA*dt*K*v
56 % Artificial viscosity to dump the high-frequency numerical noise
57 %*****
58
59 %*** Set here the parameters of the time solver : ***
60 NT = 2500; % Number of timesteps
61 CFL = 0.6; % Stability number = CFL_1D / sqrt(2)
62 % Ratio of simulation time step to min(Delta_x)/VS.
63 %*****
64
65 %*** Set here the parameters of the fault : ***
66 Normal_stress = 120e6; % Fault normal stress
67 Shear_stress = 70e6; % Shear stress
68 Static_friction = 0.677; % Static friction coefficient
69 Dynamic_friction = 0.525; % Dynamic friction coefficient
70 dc = 0.4; % Slip-weakening distance
71 Fault_length = 25e3; % Simulated fault length
72 Nuc_length = 1.5e3; % Nucleation length
73 Nuc_str = 81.6e6; % Initial shear stress in nucleation zone
74 %*****
75
76 %*** Set here receiver locations : ***
77 Sta_x = [-16e3:2400:32e3]'; % x coord of receivers
78 Sta_y = 7.5e3; % y coord of receivers
79 OUTdt = 100; % steps for output snapshots
80 %*****
81
82 %*** Set here properties of heterogeneity: ***
83 Het_type = 'uniform'; % Options are: uniform, trapezoid, random, box, EQ, and steps
84 % The default option is uniform.
```

# Add a heterogeneity

```
%**** Set here properties of heterogeneity: ****
Het_type = 'uniform'      ; % options are: uniform, box, trapezoid, random, EQ, and steps
                          % The default option is uniform.
Het_para = 'stress'       ; % The options are: stress and dc. If Het_type='EQ',
                          % this option shall be stress.
Het_loc   = 20e3           ; % The beginning location of heterogeneity (m)
Het_len   = 10e3           ; % The length of heterogeneity (m)
Het_val   = 50e6           ;
                          % The value of heterogeneity. The unit is Pa for stress and m for dc.
%Het_val  = [70e6, 50e6]   ;
                          % For trapezoid, the format is [val1,val2]
                          % For random, this value indicates perturbation range
                          % For EQ, this value indicates the stress drop of the previous event.
%Het_val  = [60e6,75e6,53e6] ;
                          % For steps, the format is [val1, val2, ...]
%*****
```

# Add a heterogeneity

```
***** Set here properties of heterogeneity: *****
Het_type = 'uniform'      ; % options are: uniform, box, trapezoid, random, EQ, and steps
                          % The default option is uniform.
Het_para = 'stress'       ; % The options are: stress and dc. If Het_type='EQ',
                          % this option shall be stress.
Het_loc  = 20e3            ; % The beginning location of heterogeneity (m)
Het_len  = 10e3            ; % The length of heterogeneity (m)
Het_val  = 50e6            ;
                          % The value of heterogeneity. The unit is Pa for stress and m for dc.
%Het_val = [70e6, 50e6]   ;
                          % For trapezoid, the format is [val1,val2]
                          % For random, this value indicates perturbation range
                          % For EQ, this value indicates the stress drop of the previous event.
%Het_val = [60e6,75e6,53e6] ;
                          % For steps, the format is [val1, val2, ...]
%*****
```

# Model name meaning

2D or 2.5D

stress or dc

length

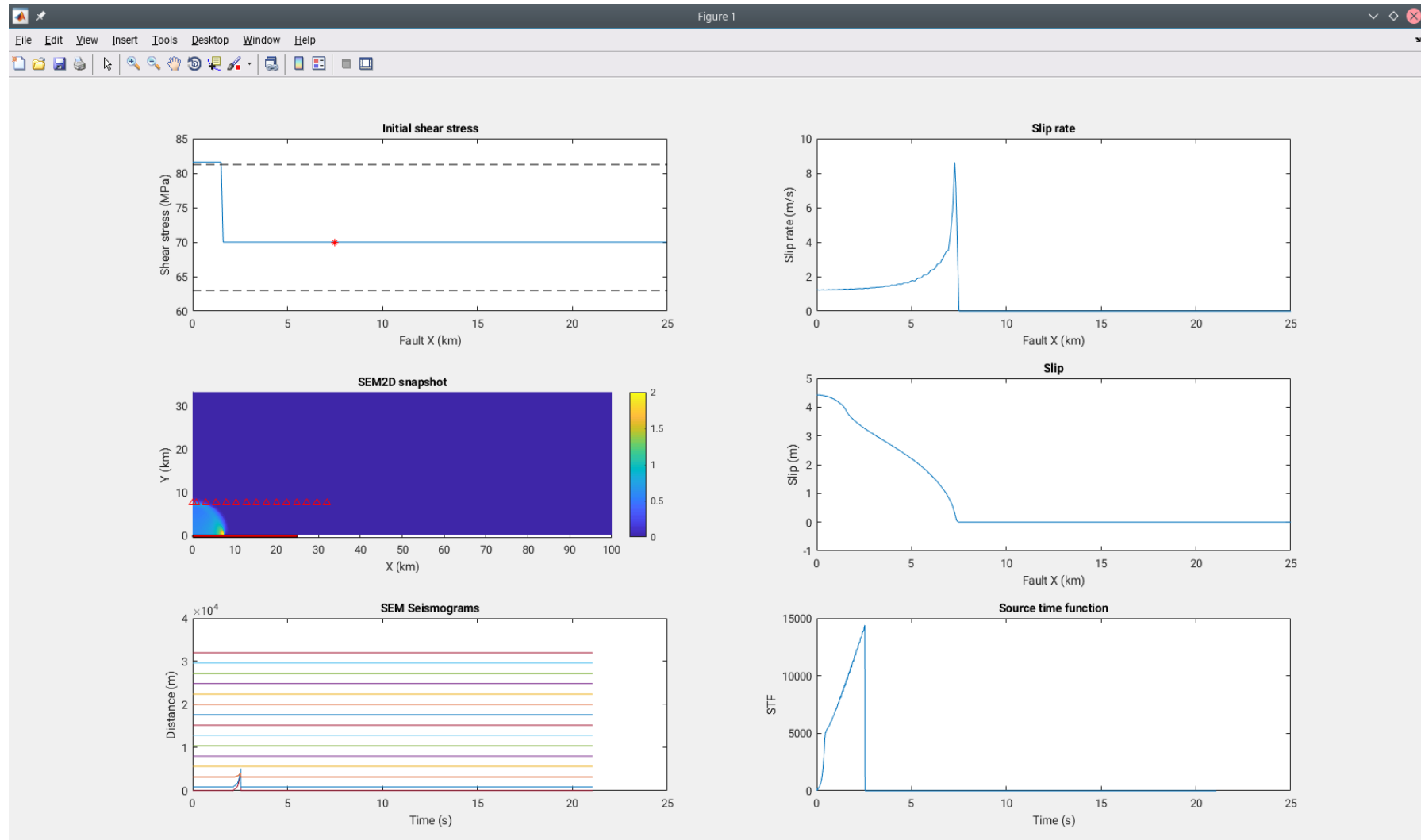
2.5D-8-box-stress-20-30-50

uniform, box, random ...

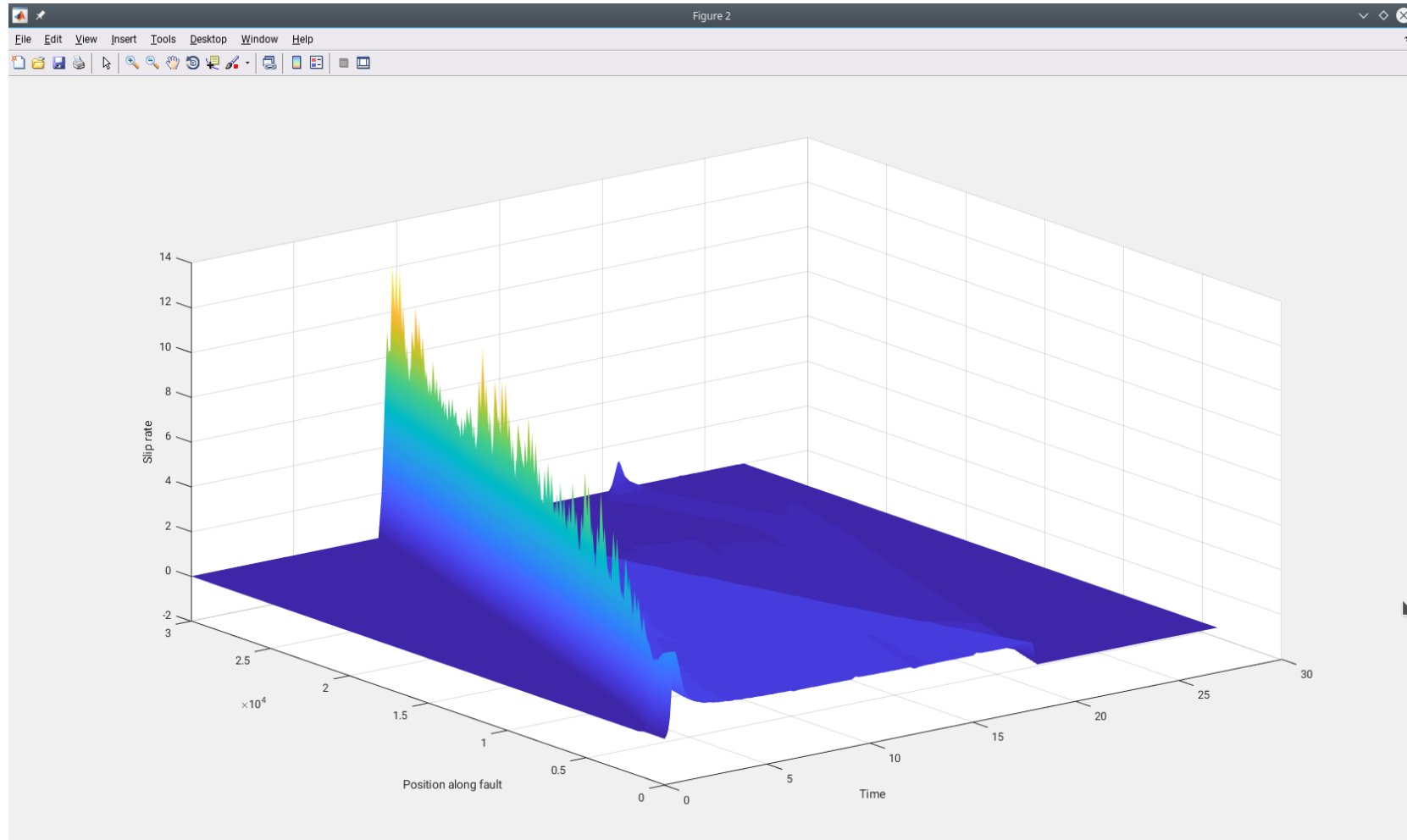
location

value

# Result presentation (figure1)

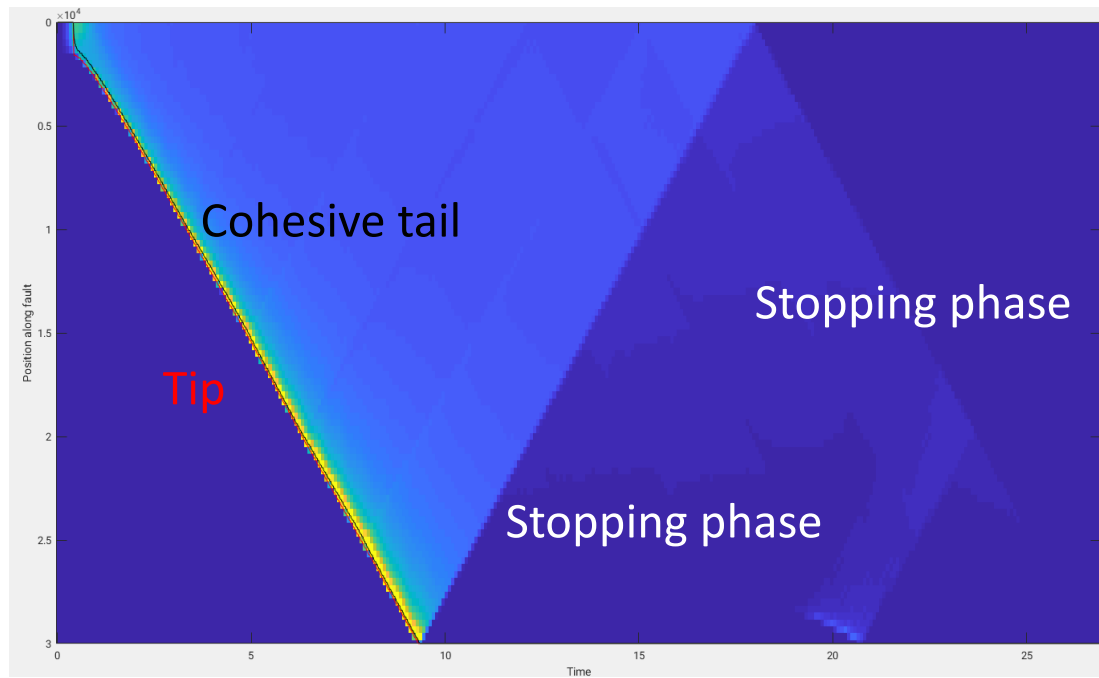


# Result presentation (figure2)

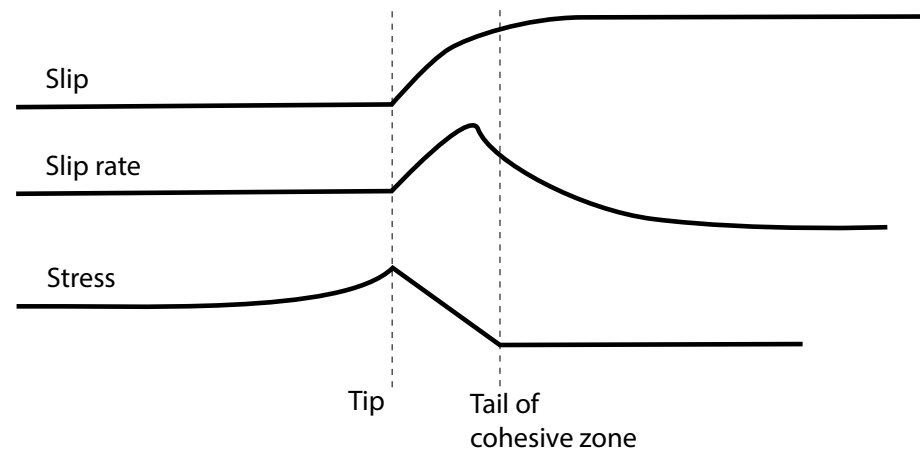
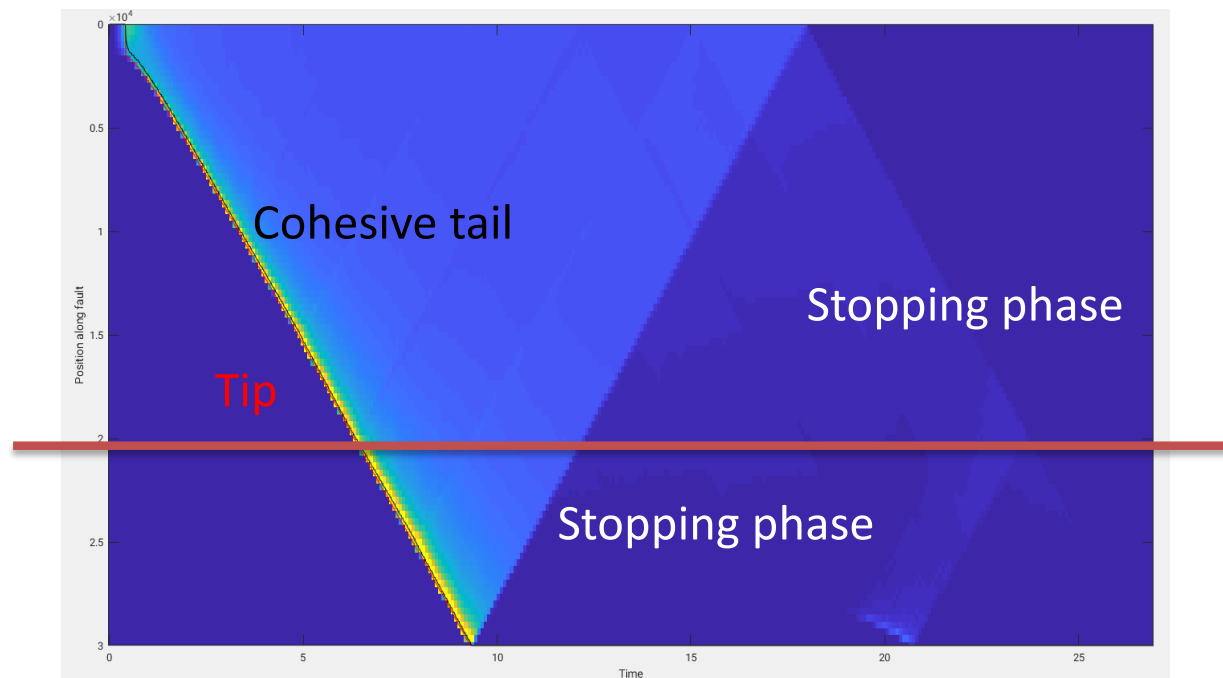




# Result presentation (figure3)



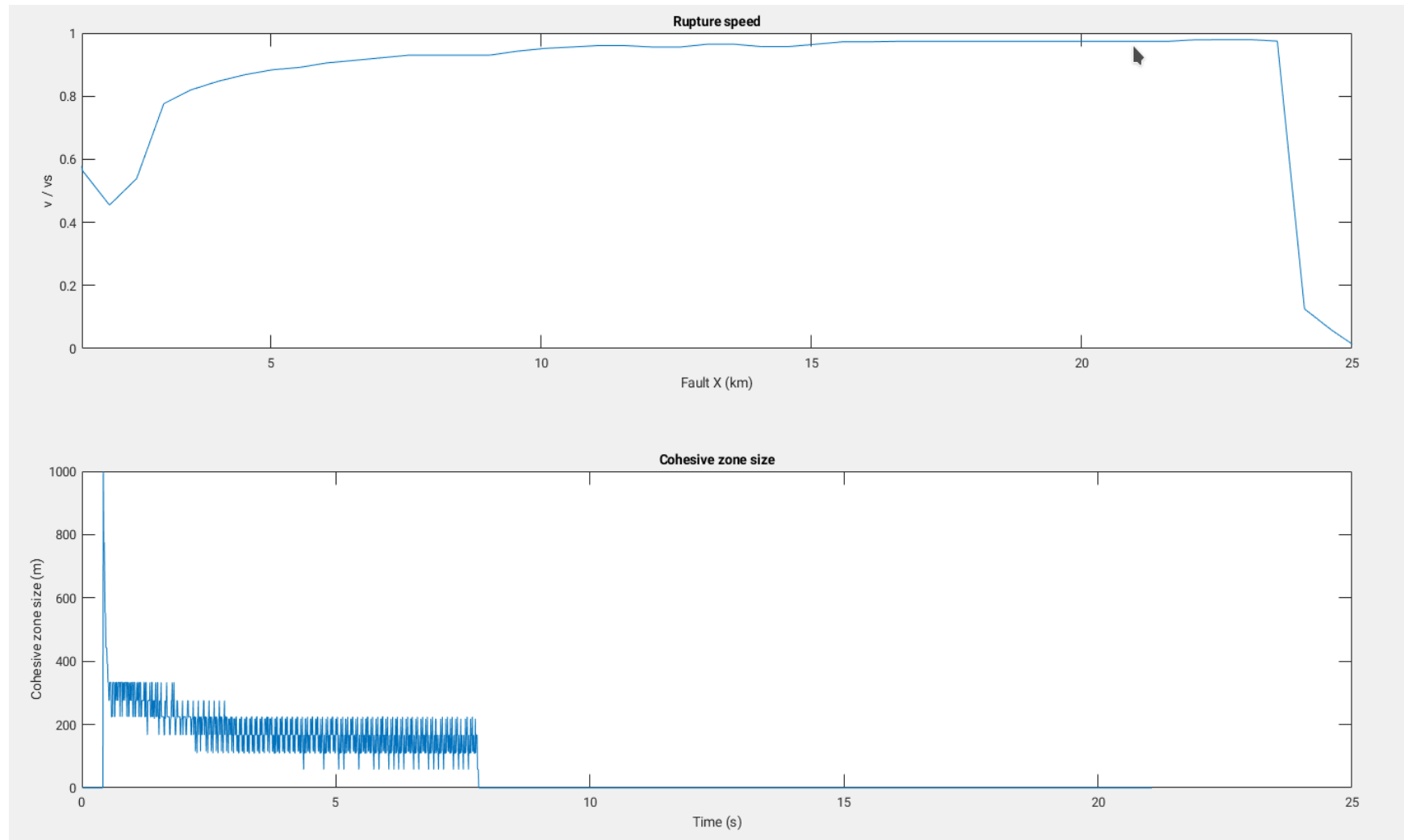
# Result presentation (figure3)



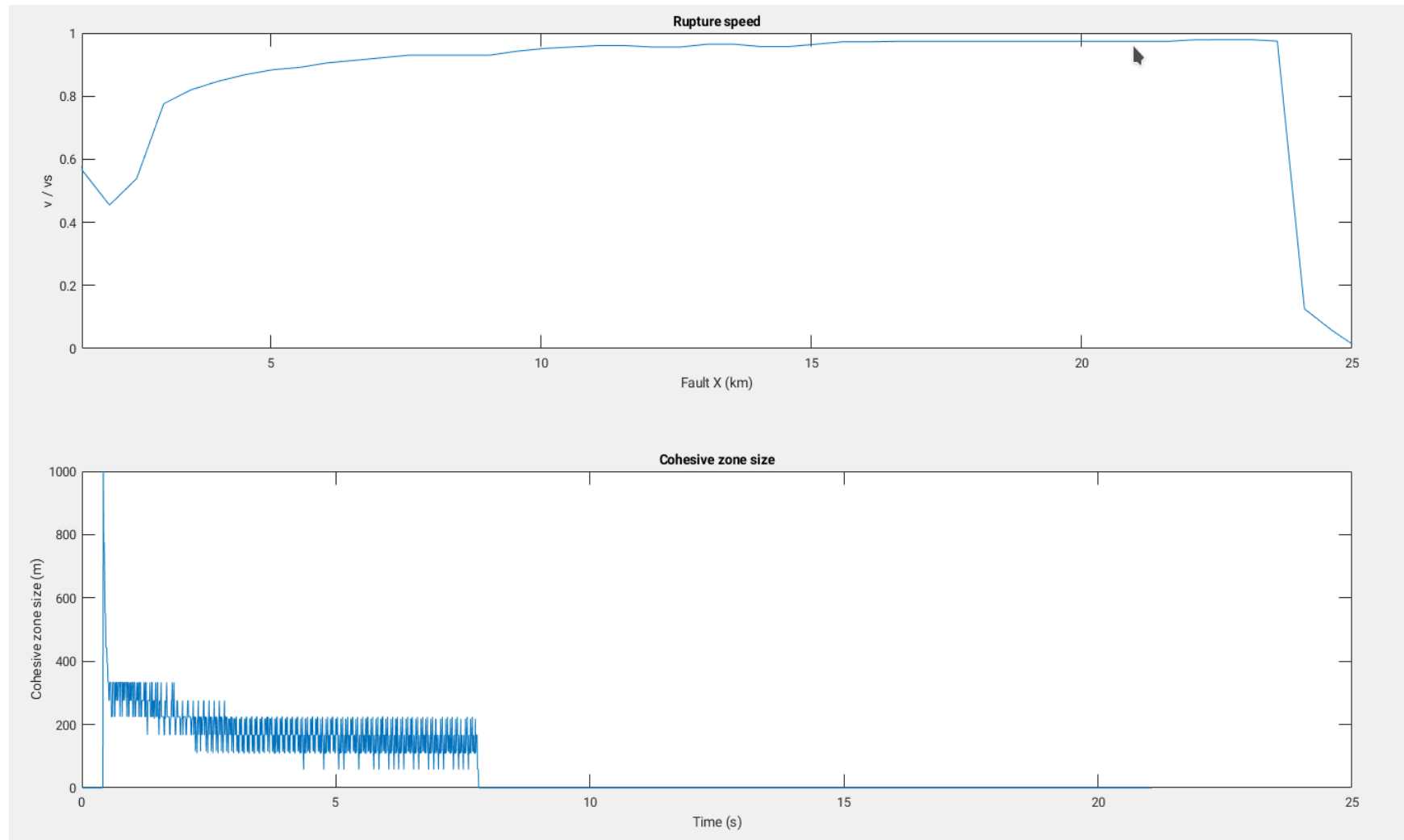
Definition of tip:  
Slip rate  $>$  threshold

Definition of cohesive tail:  
Slip =  $D_c$

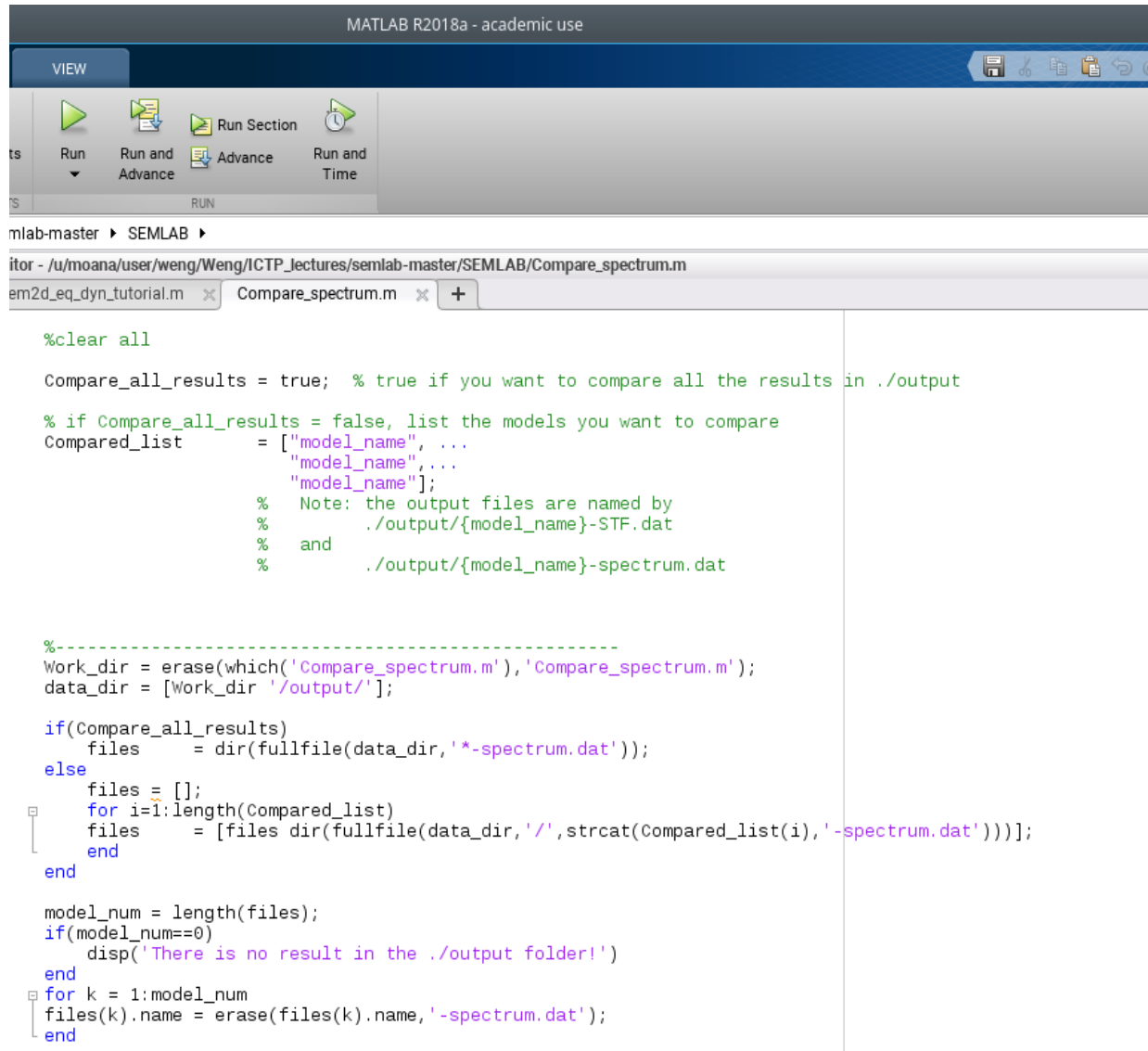
# Result presentation (figure4)



# Result presentation (figure4)



# STF and its spectrum (final)



The image shows the MATLAB R2018a - academic use interface. The top toolbar includes a 'VIEW' button and icons for saving, cutting, copying, pasting, and undo. Below the toolbar is a 'RUN' section with buttons for 'Run', 'Run and Advance', 'Run Section', 'Advance', and 'Run and Time'. The main window displays the script 'Compare\_spectrum.m' with the following code:

```
%clear all

Compare_all_results = true; % true if you want to compare all the results in ./output

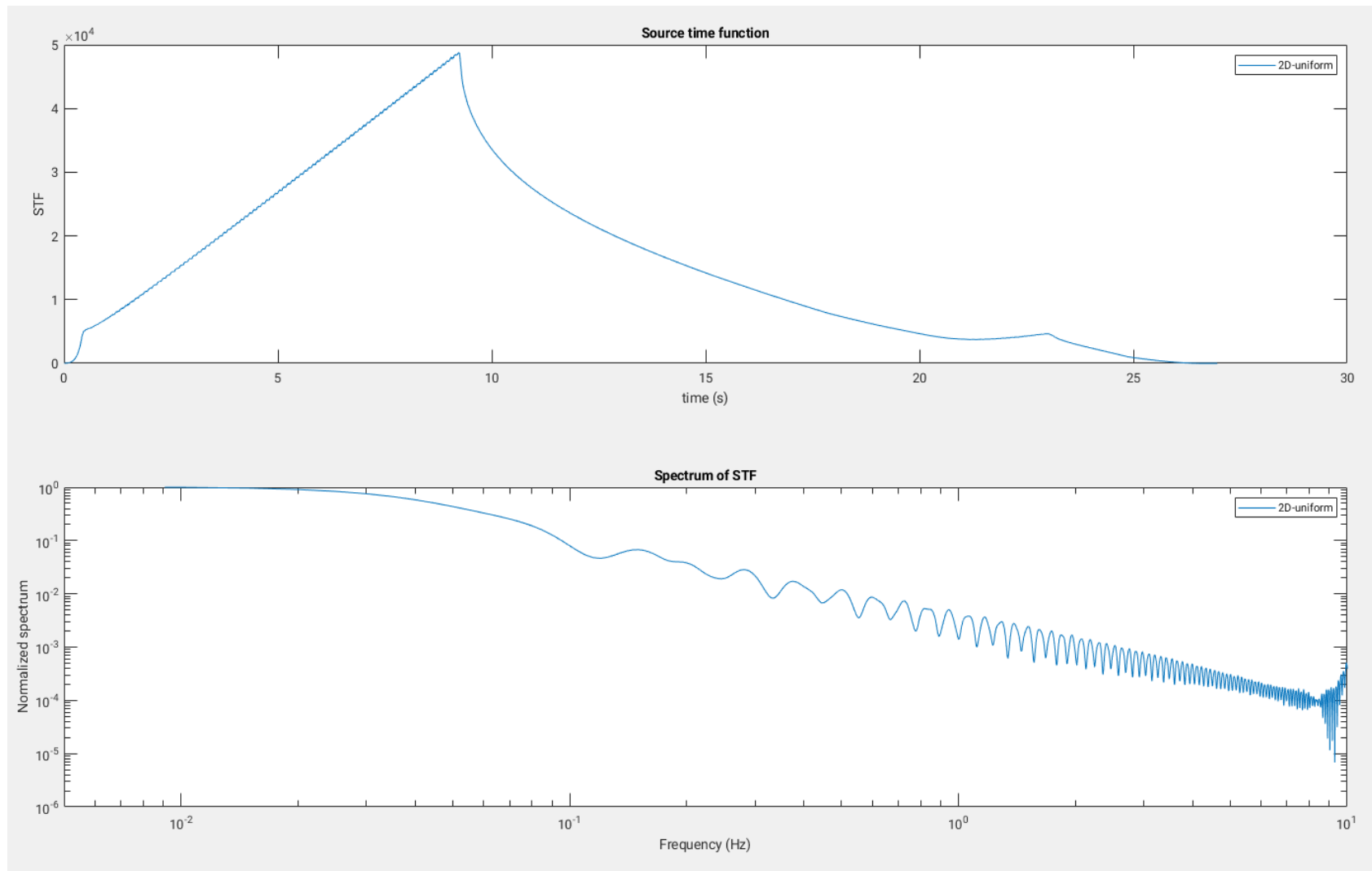
% if Compare_all_results = false, list the models you want to compare
Compared_list      = ["model_name", ...
                      "model_name",...
                      "model_name"];
% Note: the output files are named by
%       ./output/{model_name}-STF.dat
% and
%       ./output/{model_name}-spectrum.dat

%-----
Work_dir = erase(which('Compare_spectrum.m'), 'Compare_spectrum.m');
data_dir = [Work_dir './output/'];

if(Compare_all_results)
    files = dir(fullfile(data_dir, '*-spectrum.dat'));
else
    files = [];
    for i=1:length(Compared_list)
        files = [files dir(fullfile(data_dir, '/', strcat(Compared_list(i), '-spectrum.dat')))];
    end
end

model_num = length(files);
if(model_num==0)
    disp('There is no result in the ./output folder!')
end
for k = 1:model_num
    files(k).name = erase(files(k).name, '-spectrum.dat');
end
```

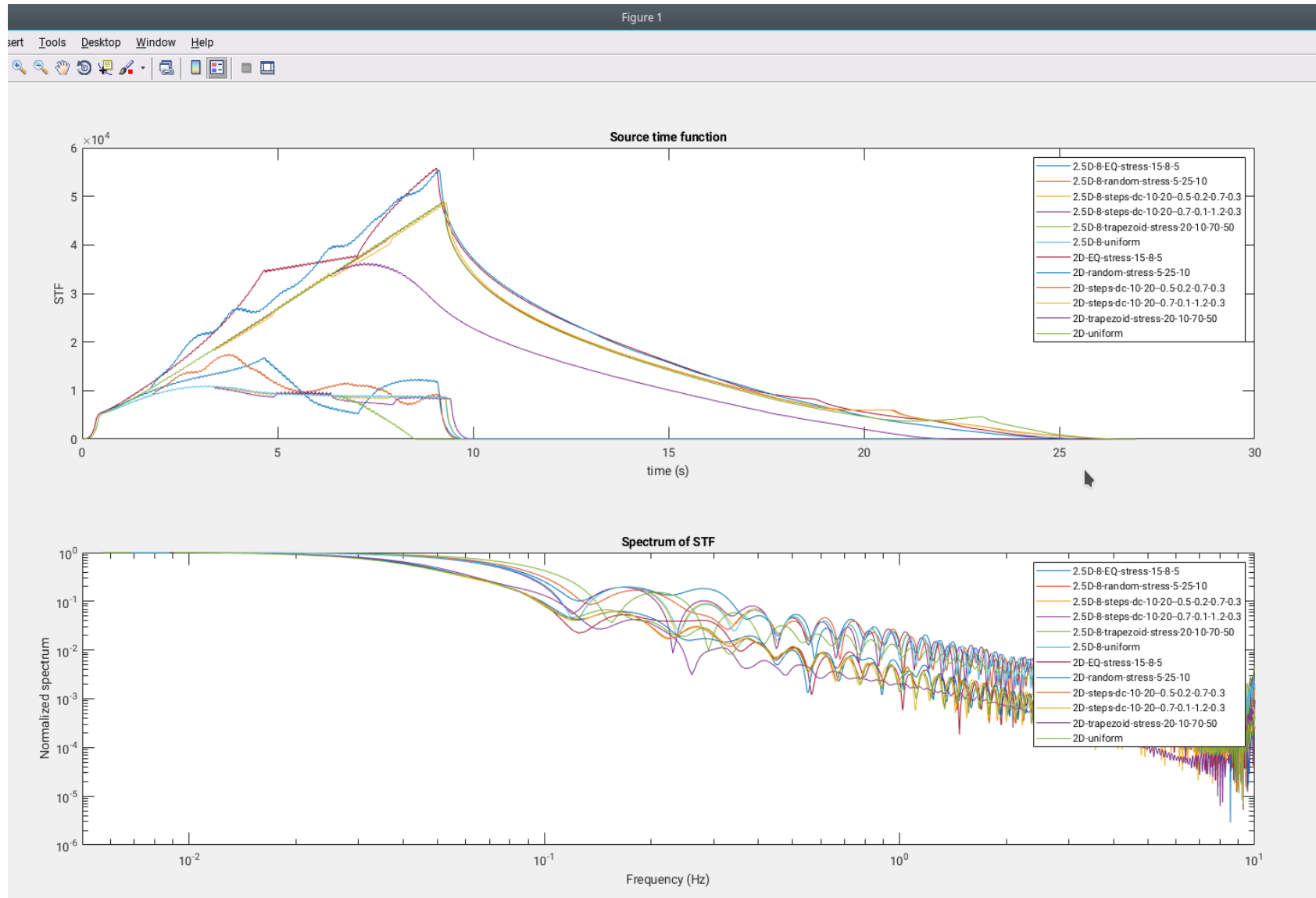
# STF and spectra



# Add a heterogeneity

```
%**** Set here properties of heterogeneity: ****
Het_type = 'uniform'      ; % options are: uniform, box, trapezoid, random, EQ, and steps
                          % The default option is uniform.
Het_para = 'stress'       ; % The options are: stress and dc. If Het_type='EQ',
                          % this option shall be stress.
Het_loc   = 20e3           ; % The beginning location of heterogeneity (m)
Het_len   = 10e3           ; % The length of heterogeneity (m)
Het_val   = 50e6           ;
                          % The value of heterogeneity. The unit is Pa for stress and m for dc.
%Het_val  = [70e6, 50e6]   ;
                          % For trapezoid, the format is [val1,val2]
                          % For random, this value indicates perturbation range
                          % For EQ, this value indicates the stress drop of the previous event.
%Het_val  = [60e6,75e6,53e6] ;
                          % For steps, the format is [val1, val2, ...]
%*****
```

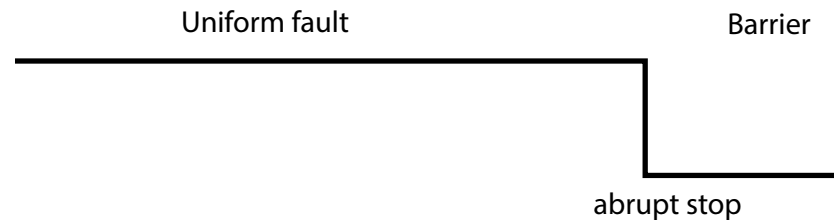
# STF comparison



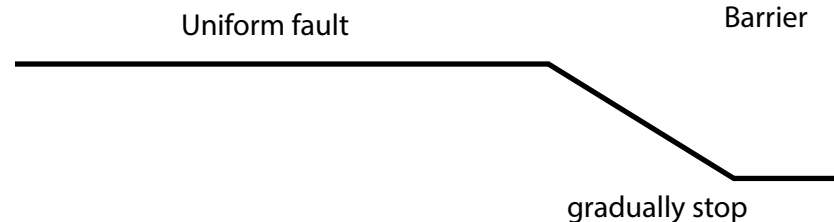


# Models to test

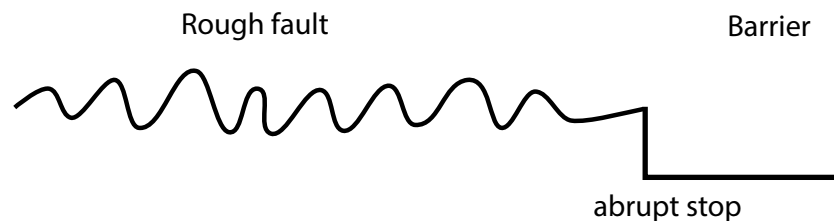
## Model 1



## Model 2



## Model 3



- Heterogeneity can be initial stress or Dc
- Question: what are the differences of STF and spectrum?
- Discussion: what are the differences between 2D and 2.5D models?

# Discussion

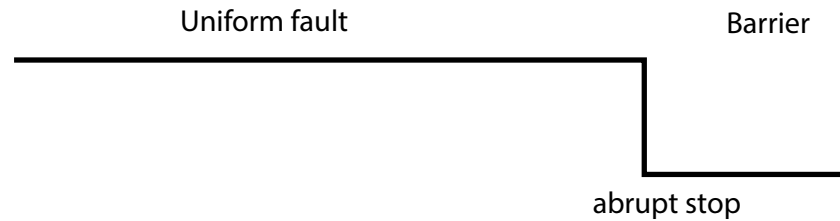
## Apple to apple

➤ model 1 vs. model 2

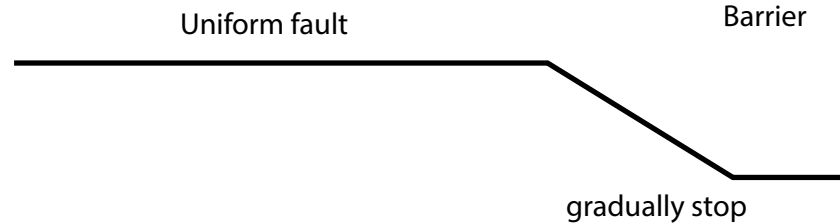
➤ model 1 vs. model 3

➤ 2D vs. 2.5D

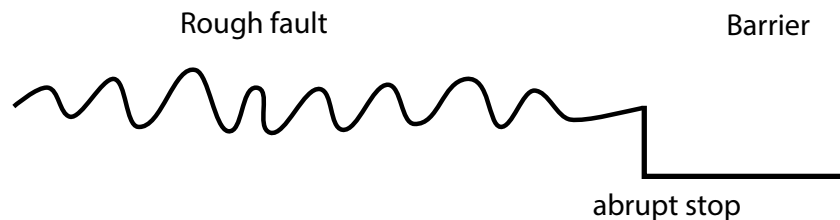
**Model 1**



**Model 2**

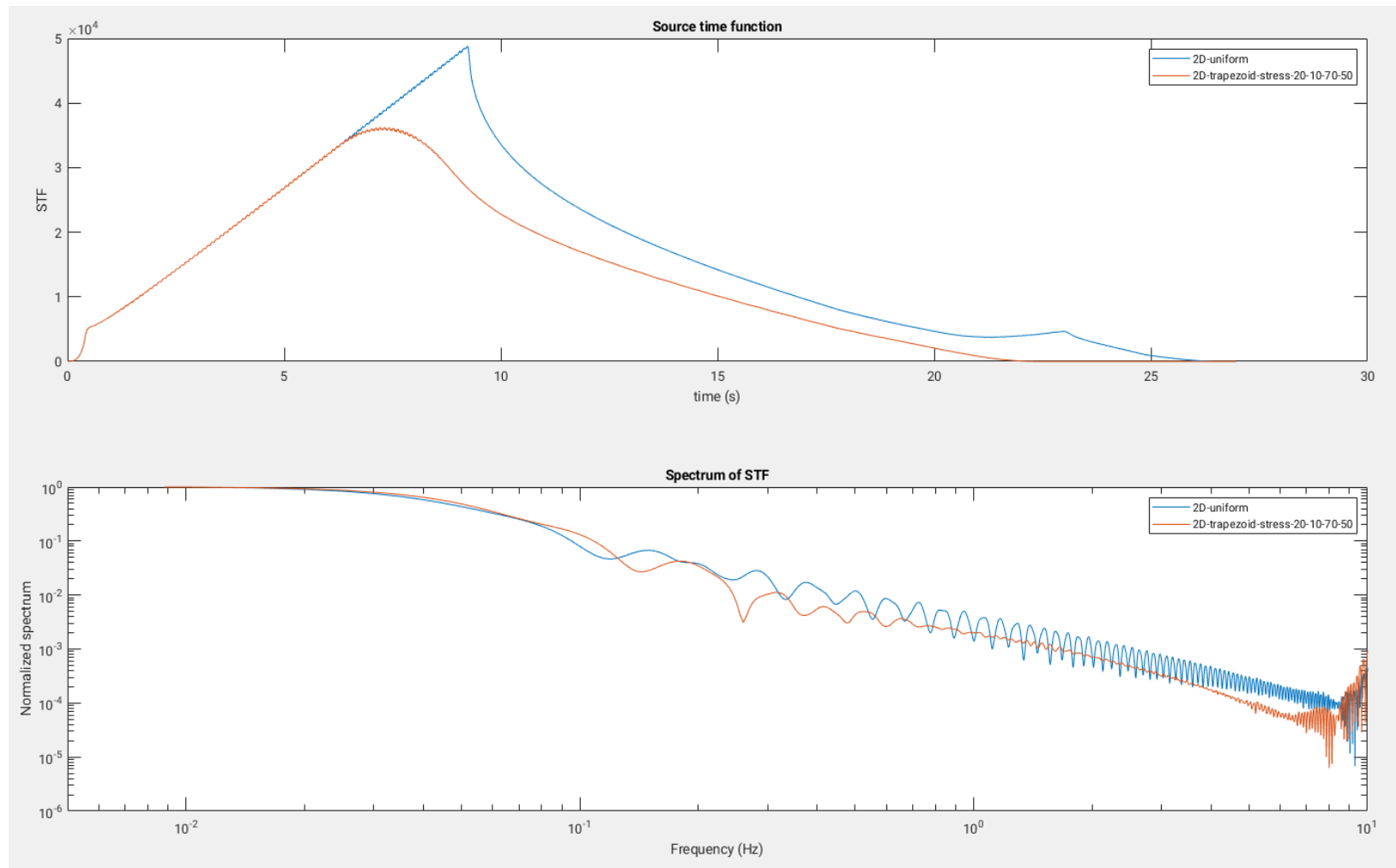


**Model 3**



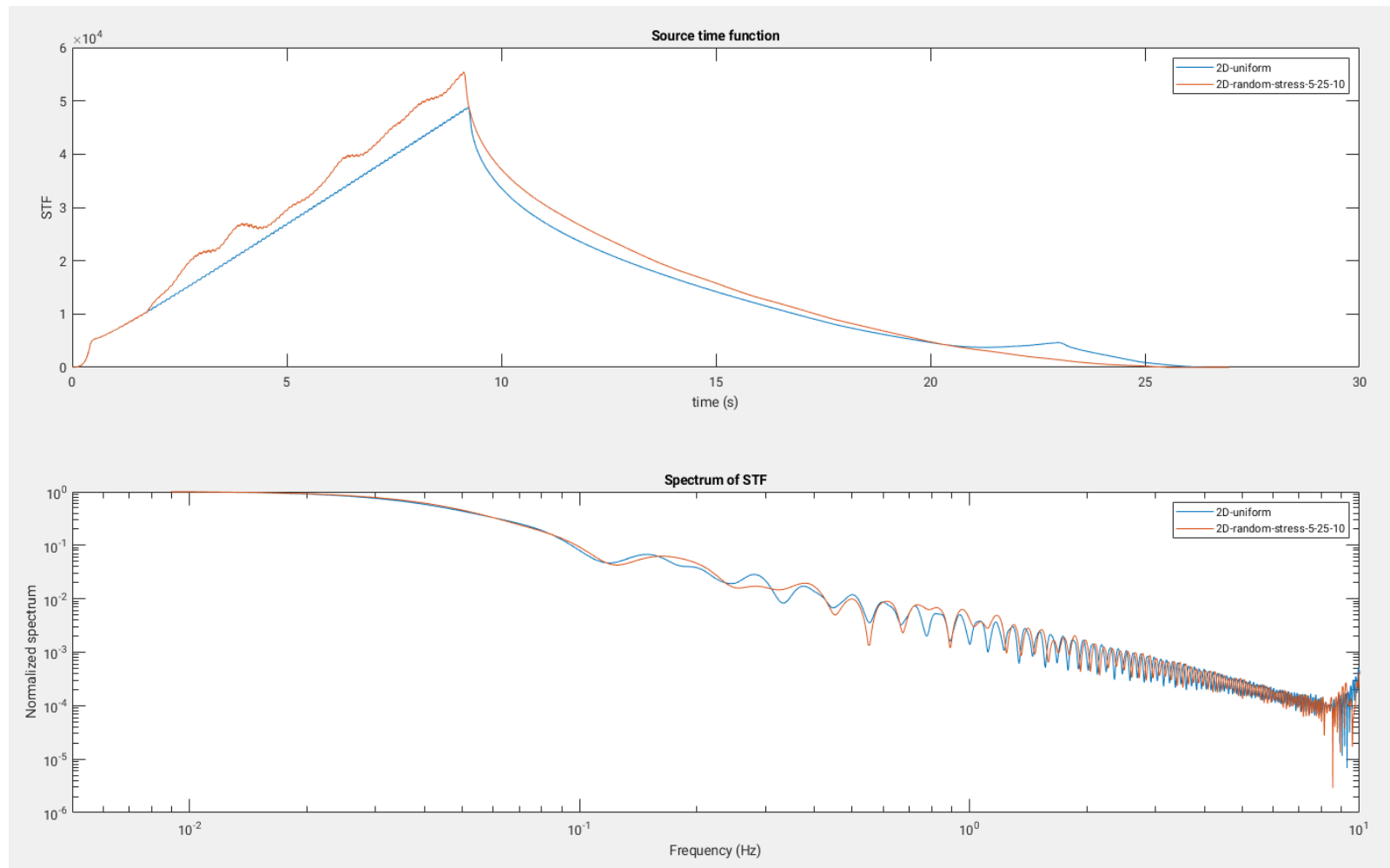
# Discussion

## ➤ model 1 vs. model 2



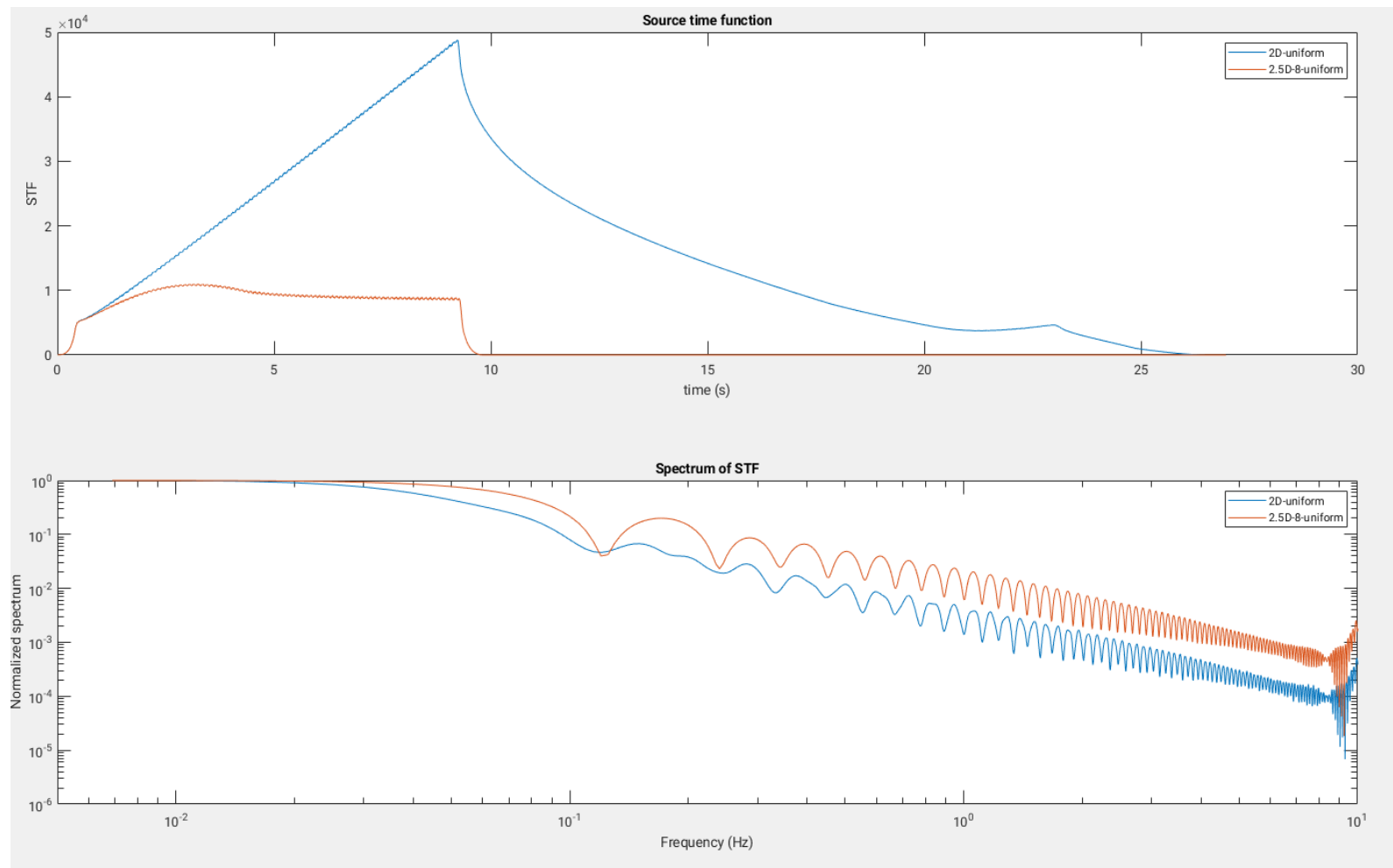
# Discussion

## ➤ model 1 vs. model 3



# Discussion

## ➤ 2D vs. 2.5D



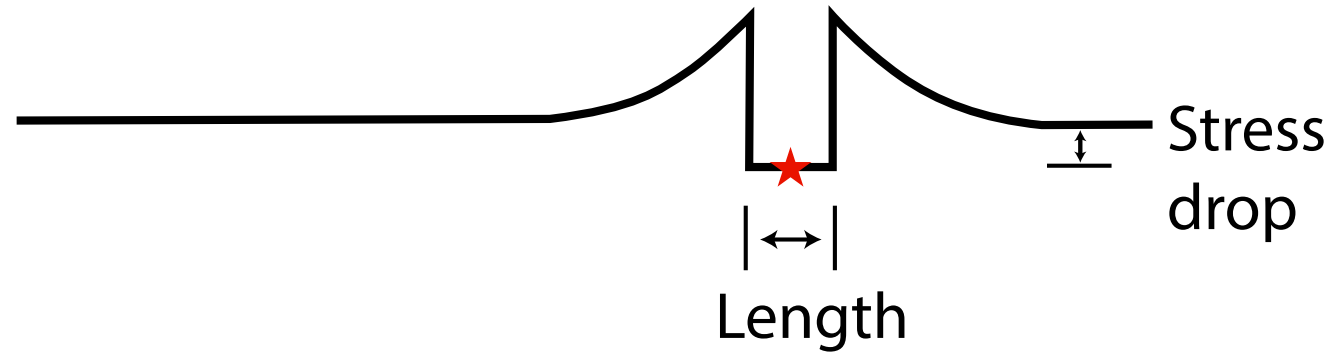
Uniform initial stress

$t_1$



One small earthquake occurred

$t_2$



The model you are going to simulate

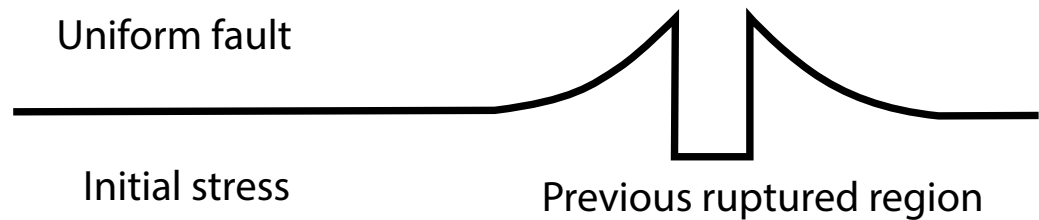
$t_3$



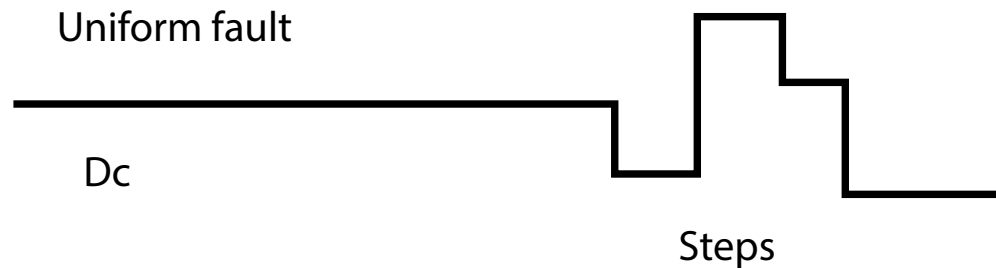
# Models to test

- Heterogeneity is initial stress
- Question: do these heterogeneities increase high-frequency radiation?
- Discussion: what parameters may control the high-frequency radiation?

## Model 4



## Model 5



# Discussion

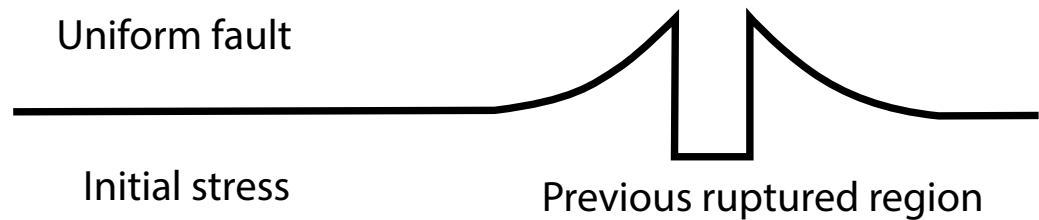
## Apple to apple

➤ model 1 vs. model 4

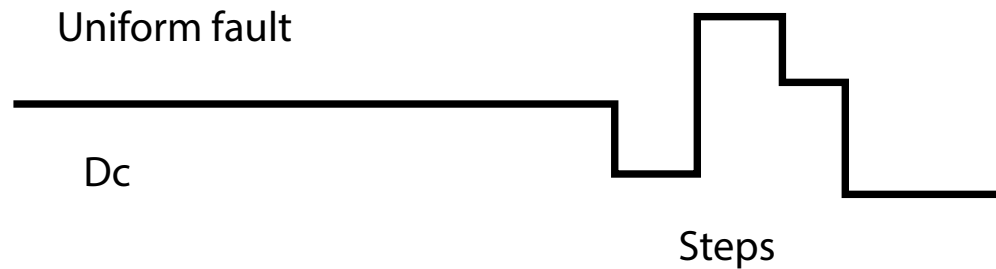
➤ model 1 vs. model 5

➤ 2D vs. 2.5D

### Model 4



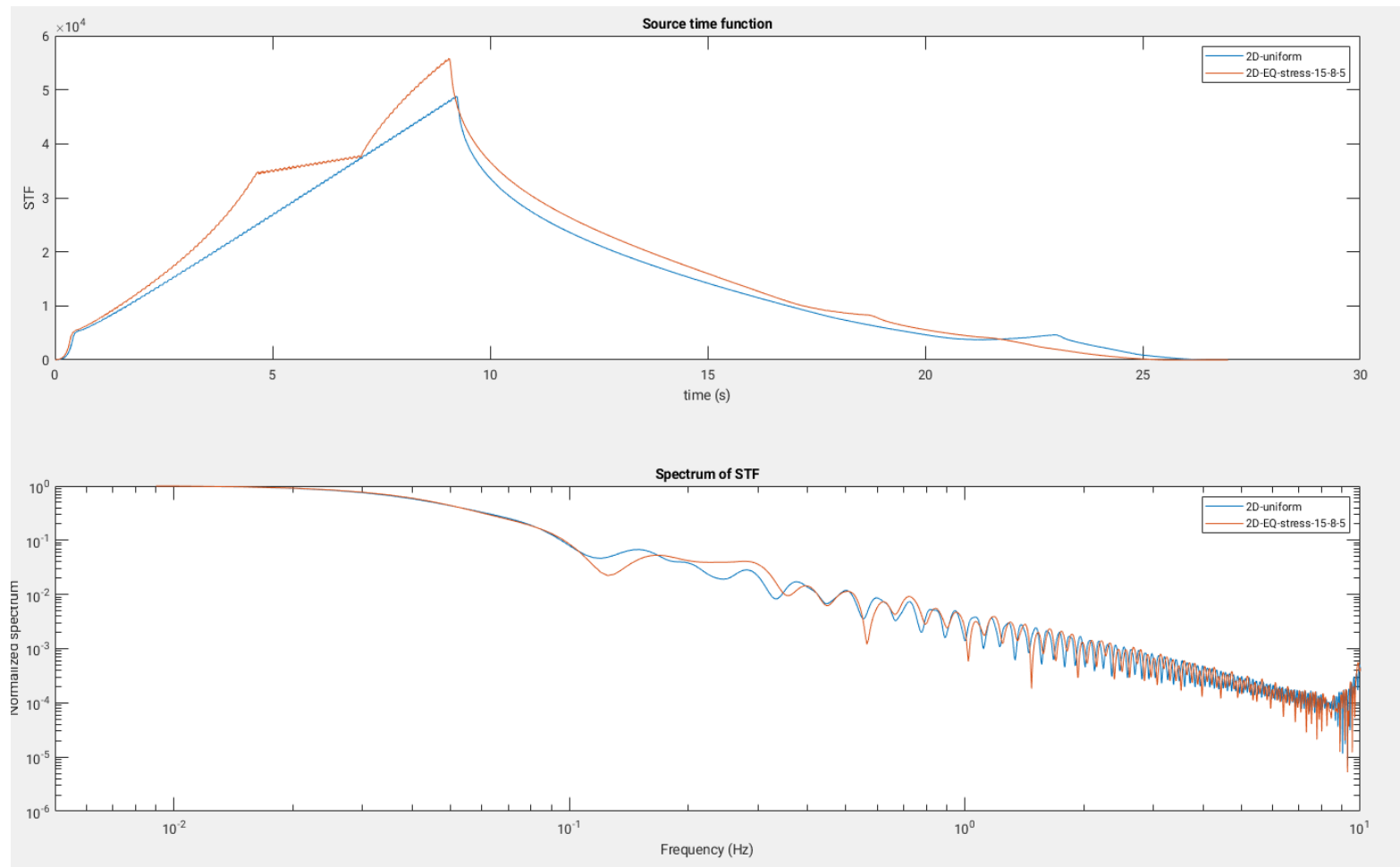
### Model 5





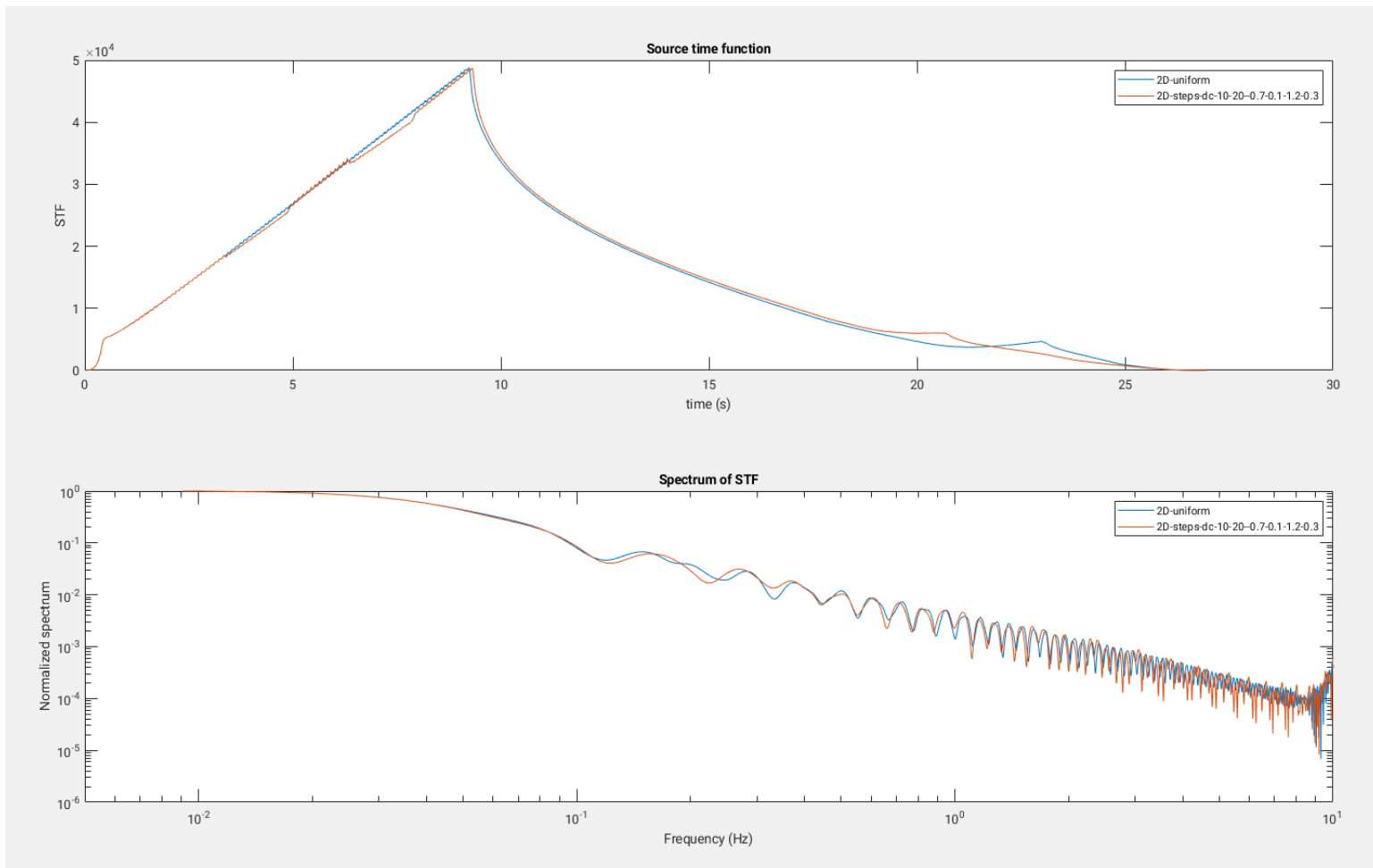
# Discussion

## ➤ model 1 vs. model 4



# Discussion

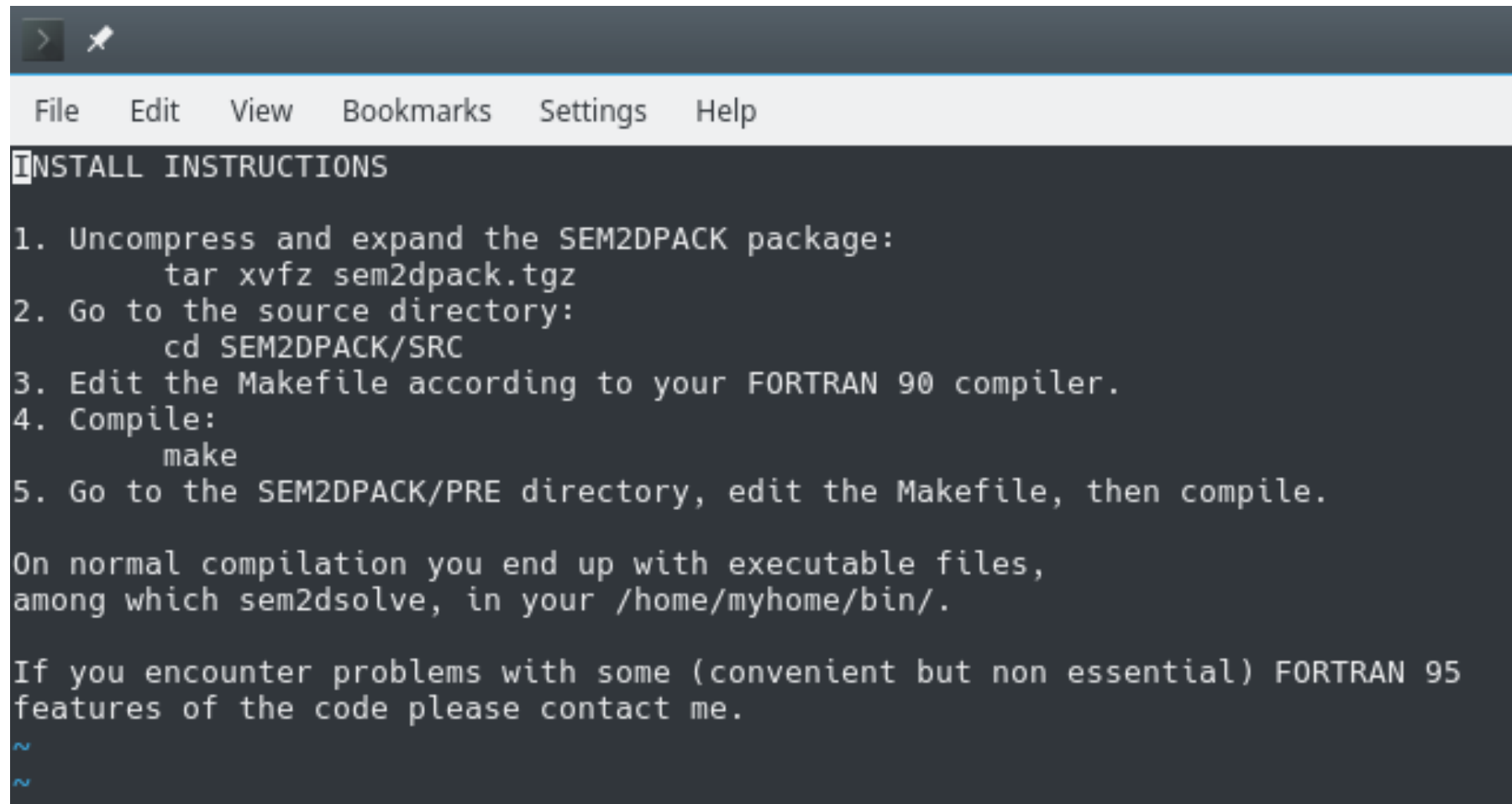
## ➤ model 1 vs. model 5



Tutorial of SEMLAB

Tutorial of sem2dpack

# Installation of sem2dpack



```
> ✂
File Edit View Bookmarks Settings Help
INSTALL INSTRUCTIONS

1. Uncompress and expand the SEM2DPACK package:
   tar xvfz sem2dpack.tgz
2. Go to the source directory:
   cd SEM2DPACK/SRC
3. Edit the Makefile according to your FORTRAN 90 compiler.
4. Compile:
   make
5. Go to the SEM2DPACK/PRE directory, edit the Makefile, then compile.

On normal compilation you end up with executable files,
among which sem2dsolve, in your /home/myhome/bin/.

If you encounter problems with some (convenient but non essential) FORTRAN 95
features of the code please contact me.
~
~
```

# How to run sem2dpack?

```
cd ./sem2dpack-  
25D/EXAMPLES/2.5D_inplane
```

open this file by vim or other method:

```
vi Par.inp
```

# Parameter setup in sem2dpack

```
> ✎
File Edit View Bookmarks Settings Help

#----- Some general parameters -----
&GENERAL iexec=1, ngll=5, fmax=3.d0 , W=10d3, ndof=2 ,
  title = '2.5D elastic in-plane model', verbose='1111' , ItInfo = 400/

#----- Build the mesh -----
&MESH_DEF method = 'CARTESIAN'/
&MESH_CART xlim=0d3,100d3, zlim=0d3,50d3, nelem=160,80/

#---- Material parameters -----
&MATERIAL tag=1, kind='ELAST' /
&MAT_ELASTIC rho=2705.d0, cp=5770.d0, cs=3330.d0 /

#----- Boundary conditions -----
&BC_DEF tag = 1, kind = 'DYNFLT' /
&BC_DYNFLT friction='SWF','TWF', Tn=-50d6,Tt=30.5d6 /
&BC_DYNFLT_SWF Dc=0.4d0, MuS=0.63d0, MuD=0.54d0 /
&BC_DYNFLT_TWF kind=1, MuS=0.63d0, MuD=0.54d0, Mu0=0.63d0,
  X=0.d0, Z=0.d0, V=0.333d3, L=0.1665d3, T=60d0 /

&BC_DEF tag = 2 , kind = 'ABSORB' /
&BC_DEF tag = 3 , kind = 'ABSORB' /
&BC_DEF tag = 4 , kind = 'DIRNEU' /
&BC_DIRNEU h='N', v='D' /

#---- Time scheme settings -----
&TIME kind='leapfrog', TotalTime=30 /

#----- Receivers -----
&REC_LINE number = 10 , first = 0d3,10d3, last = 50.d3,10d3, isamp=20, AtNode=F /

#----- Plots settings -----
&SNAP_DEF itd=100, fields = 'DVS',bin=T,ps=F /
&SNAP_PS vectors=F, interpol=T, DisplayPts=6, ScaleField=0d0 /
~
~
```

# Parameter setup in sem2dpack

```
> ✎
File Edit View Bookmarks Settings Help
#----- Some general parameters -----
&GENERAL iexec=1, ngll=5, fmax=3.d0 , W=10d3, ndof=2 ,
  title = '2.5D elastic in-plane model', verbose='1111' , ItInfo = 400/

#----- Build the mesh -----
&MESH_DEF method = 'CARTESIAN' /
&MESH_CART xlim=0d3,10d3, zlim=0d3,50d3, nelem=160,80/

#---- Material parameters -----
&MATERIAL tag=1, kind='ELAST' /
&MAT_ELASTIC rho=2705.d0, cp=5770.d0, cs=3330.d0 /

#----- Boundary conditions -----
&BC_DEF tag = 1, kind = 'DYNFLT' /
&BC_DYNFLT friction='SWF', 'TWF', Tn=-50d6, Tt=30.5d6 /
&BC_DYNFLT_SWF Dc=0.4d0, MuS=0.63d0, MuD=0.54d0 /
&BC_DYNFLT_TWF kind=1, MuS=0.63d0, MuD=0.54d0, Mu0=0.63d0,
  X=0.d0, Z=0.d0, V=0.333d3, L=0.1665d3, T=60d0 /

&BC_DEF tag = 2 , kind = 'ABSORB' /
&BC_DEF tag = 3 , kind = 'ABSORB' /
&BC_DEF tag = 4 , kind = 'DIRNEU' /
&BC_DIRNEU h='N', v='D' /

#---- Time scheme settings -----
&TIME kind='leapfrog', TotalTime=30 /

#----- Receivers -----
&REC_LINE number = 10 , first = 0d3,10d3, last = 50.d3,10d3, isamp=20, AtNode=F /

#----- Plots settings -----
&SNAP_DEF itd=100, fields = 'DVS', bin=T, ps=F /
&SNAP_PS vectors=F, interpol=T, DisplayPts=6, ScaleField=0d0 /
```

# How to present results?

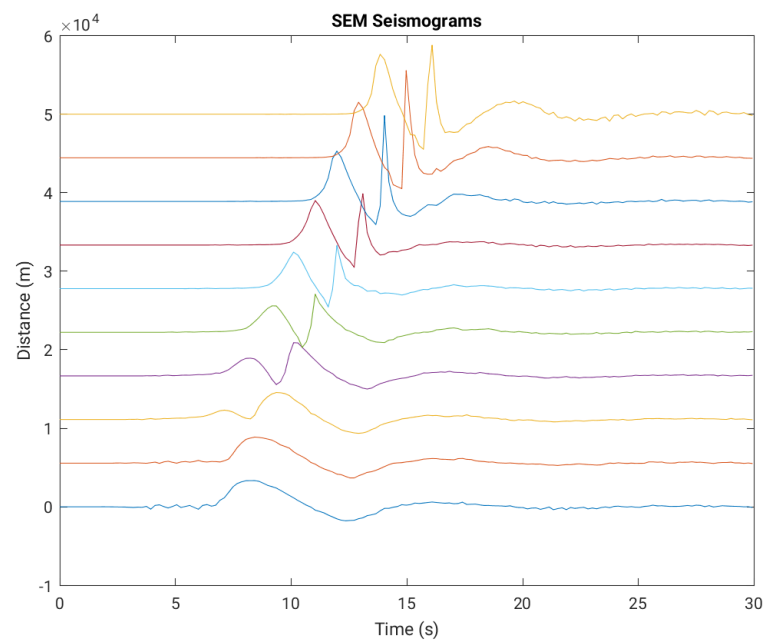
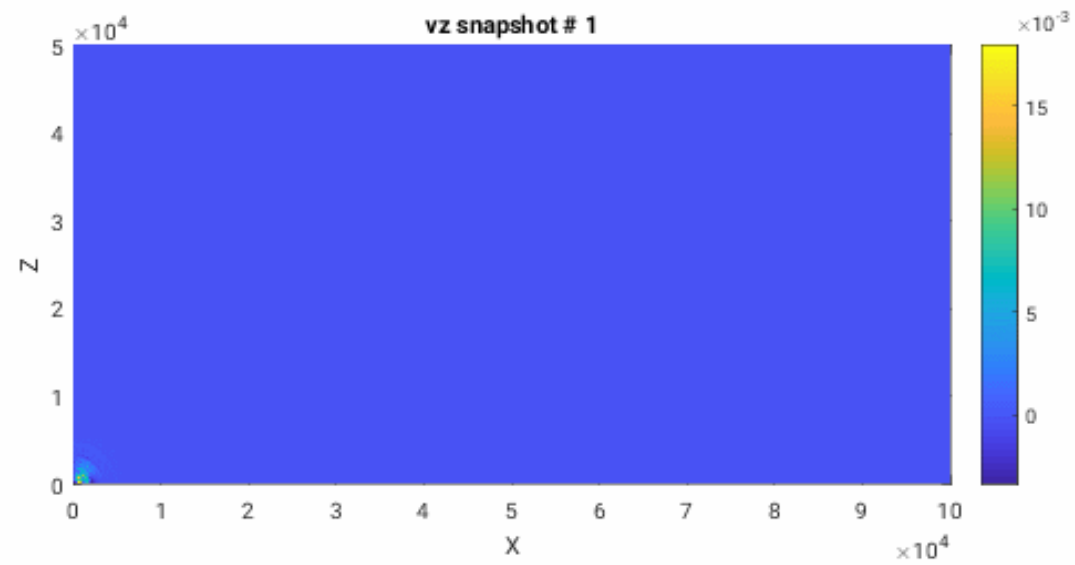
```
cd ${work_dir}/sem2dpack-25D/POST
```

open the script by MATLAB:

`create_movie.m`

run it by MATLAB





# Discussion

- Find seismic phases, such as P wave front, S wave, Rayleigh wave, etc.
- What parameters may control the formation of supershear rupture?





