

SEAS Benchmark Problems BP5-QD and BP5-FD

Junle Jiang and Brittany Erickson

September 22, 2020

Benchmark problem BP5 (-QD: quasi-dynamic; -FD: fully dynamic) is for a three-dimensional (3D) problem in a half-space, modified from the whole-space problem in BP4. Some model parameters are changed to reduce the computational demand. The model size, resolution, initial and boundary conditions, and output are designed specifically for 3D problems.

1 3D Problem Setup

The medium is assumed to be a homogeneous, isotropic, linear elastic half-space defined by

$$\mathbf{x} = (x_1, x_2, x_3) \in (-\infty, \infty) \times (-\infty, \infty) \times (0, \infty),$$

with a free surface at $x_3 = 0$ and x_3 as positive downward. A vertical, strike-slip fault is embedded at $x_1 = 0$, see Figure 1. We use the notation “+” and “-” to refer to the side of the fault with x_1 positive, and x_1 negative, respectively. We assume 3D motion, letting $u_i = u_i(\mathbf{x}, t)$, $i = 1, 2, 3$ denote the displacement in the i -direction. For BP5-FD, motion is governed by momentum balance

$$\rho \frac{\partial^2 \mathbf{u}}{\partial t^2} = \nabla \cdot \boldsymbol{\sigma} \quad (1)$$

in \mathbb{R}^3 , where ρ is the material density. For BP5-QD, inertia is neglected and we consider the equilibrium equation. Hooke’s law relates stresses to strains by

$$\sigma_{ij} = K \epsilon_{kk} \delta_{ij} + 2\mu \left(\epsilon_{ij} - \frac{1}{3} \epsilon_{kk} \delta_{ij} \right) \quad (2)$$

for bulk modulus K and shear modulus μ . The strain-displacement relations are given by

$$\epsilon_{ij} = \frac{1}{2} \left[\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right]. \quad (3)$$

2 Boundary and Interface Conditions

At $x_1 = 0$, the fault defines the interface and we supplement equations (1)–(3) with six interface conditions. A free surface lies at $x_3 = 0$, where all components of the traction vector equal 0. Mathematically, this generates the following condition:

$$\sigma_{j3}(x_1, x_2, 0, t) = 0, \quad j = 1, 2, 3. \quad (4)$$

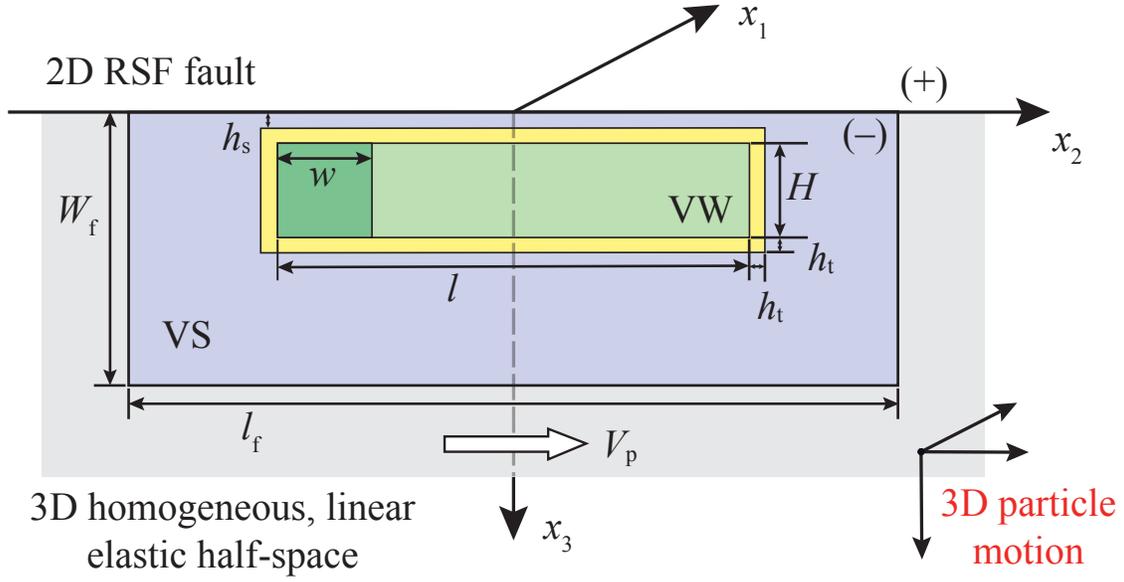


Figure 1: This benchmark considers 3D motion with a planar fault embedded vertically in a homogeneous, linear elastic half-space. The fault is governed by rate-and-state friction in the region $0 \leq x_3 \leq W_f$ and $|x_2| \leq l_f/2$, outside of which it creeps at an imposed constant horizontal rate V_p (gray). The velocity-weakening region (the rectangle in light and dark green; $h_s + h_t \leq x_3 \leq h_s + h_t + H$ and $|x_2| \leq l/2$) is surrounded by a transition zone (yellow) of width h_t to velocity-strengthening regions (blue). A favorable nucleation zone (dark green square with width w) is located at one end of the velocity-weakening patch.

We assume a “no-opening condition” on the fault, namely that

$$u_1(0^+, x_2, x_3, t) = u_1(0^-, x_2, x_3, t), \quad (5)$$

and define the slip vector

$$s_j(x_2, x_3, t) = u_j(0^+, x_2, x_3, t) - u_j(0^-, x_2, x_3, t), \quad j = 2, 3, \quad (6)$$

i.e. the jump in horizontal and vertical displacements across the fault, with right-lateral motion yielding positive values of s_2 . Positive values of s_3 occur when the + side of fault moves in the positive x_3 -direction and the – side moves in the negative x_3 -direction.

We require that components of the traction vector be equal and opposite across the fault, which yields the three conditions

$$-\sigma_{11}(0^+, x_2, x_3, t) = -\sigma_{11}(0^-, x_2, x_3, t), \quad (7a)$$

$$\sigma_{21}(0^+, x_2, x_3, t) = \sigma_{21}(0^-, x_2, x_3, t), \quad (7b)$$

$$\sigma_{31}(0^+, x_2, x_3, t) = \sigma_{31}(0^-, x_2, x_3, t), \quad (7c)$$

and denote the common values by σ (positive in compression), τ and τ_z (respectively), i.e. the normal traction and two components of shear traction. Note that positive values of τ indicate stress that tends to cause right-lateral faulting and positive values of τ_z indicates stress that tends to cause the + side of the fault to move downward (in the positive x_3 direction) and the – side to move in the negative x_3 -direction.

In addition to conditions (5) and (7), the last two interface conditions are domain dependent. We define the slip velocity vector \mathbf{V} in terms of the components

$$V_j = \dot{s}_j, \quad j = 2, 3, \quad (8)$$

letting $V = \|\mathbf{V}\|$ denote the norm of the vector. The shear stress vector is given by

$$\boldsymbol{\tau} = \begin{bmatrix} \tau \\ \tau_z \end{bmatrix}. \quad (9)$$

Within the domain $(x_2, x_3) \in \Omega_f = (-l_f/2, l_f/2) \times (0, W_f)$ we impose rate-and-state friction where shear stress on the fault is equal to fault strength \mathbf{F} , namely

$$\boldsymbol{\tau} = \mathbf{F}(\mathbf{V}, \theta); \quad (10)$$

For BP5-QD, $\boldsymbol{\tau} = \boldsymbol{\tau}^0 + \boldsymbol{\Delta}\tau - \eta\mathbf{V}$ is the sum of the prestress, the shear stress transfer due to (quasi-static) deformation, and the radiation damping approximation to inertia, where $\eta = \mu/2c_s$ is half the shear-wave impedance for shear wave speed $c_s = \sqrt{\mu/\rho}$ and density ρ . For BP5-FD, $\boldsymbol{\tau} = \boldsymbol{\tau}^0 + \boldsymbol{\Delta}\tau$, where $\boldsymbol{\Delta}\tau$ includes all stress transfers due to prior slip over the fault.

The fault strength

$$\mathbf{F} = \bar{\sigma}_n f(V, \theta) \frac{\mathbf{V}}{V}, \quad (11)$$

where θ is the state variable and $\bar{\sigma}_n = \sigma_n - p$ (the effective normal stress on the fault) for pore-pressure p . θ evolves according to the aging law

$$\frac{d\theta}{dt} = 1 - \frac{V\theta}{L}, \quad (12)$$

where L (denoted D_c in BP1 and BP2) is the critical slip distance. The friction coefficient f is given by a regularized formulation

$$f(V, \theta) = a \sinh^{-1} \left[\frac{V}{2V_0} \exp \left(\frac{f_0 + b \ln(V_0\theta/L)}{a} \right) \right] \quad (13)$$

for reference friction coefficient f_0 , reference slip rate V_0 , and rate-and-state parameters a and b . For this benchmark, b is constant as b_0 and a varies throughout Ω_f in order to define the velocity-weakening/strengthening regions (see Figure 1) as follows:

$$a(x_2, x_3) = \begin{cases} a_0, & (h_s + h_t \leq x_3 \leq h_s + h_t + H) \cap (|x_2| \leq l/2) \\ a_{\max}, & (0 \leq x_3 \leq h_s) \cup (h_s + 2h_t + H \leq x_3 \leq W_f) \\ & \cup (l/2 + h_t \leq |x_2| \leq l_f/2) \\ a_0 + r(a_{\max} - a_0), & \text{other regions} \end{cases} \quad (14)$$

where $r = \max(|x_3 - h_s - h_t - H/2| - H/2, |x_2| - l/2)/h_t$.

Outside the domain Ω_f (i.e. $|x_3| > W_f$ or $|x_2| > l_f/2$) the fault creeps horizontally at an imposed constant rate, given by the interface conditions

$$V_2(x_2, x_3, t) = V_p, \quad (15a)$$

$$V_3(x_2, x_3, t) = 0, \quad (15b)$$

where V_p is the plate rate.

3 Initial Conditions and Simulation Time

Since slip on a fault separating identical materials does not alter the normal traction, σ_n remains constant. The initial state and pre-stress on the fault is chosen so that the model can start with a uniform fault slip rate, given by

$$\mathbf{V} = \begin{bmatrix} V_{\text{init}} \\ V_{\text{zero}} \end{bmatrix}, \quad (16)$$

where V_{zero} is chosen as 10^{-20} m/s to avoid infinite $\log(V_3)$ in data output, and

$$\boldsymbol{\tau}^0 = \boldsymbol{\tau}^0 \cdot \mathbf{V}/V. \quad (17)$$

The initial state variable is chosen as the steady state at slip rate V_{init} over the entire fault, namely

$$\theta(x_2, x_3, 0) = L/V_{\text{init}}. \quad (18)$$

For **BP5-QD**, we must specify an initial value for slip, which we take to be zero, namely

$$s_j(x_2, x_3, t) = 0, \quad j = 2, 3. \quad (19)$$

The scalar pre-stress τ^0 is chosen as the steady-state stress:

$$\tau^0 = \bar{\sigma}_n a \sinh^{-1} \left[\frac{V_{\text{init}}}{2V_0} \exp \left(\frac{f_0 + b \ln(V_0/V_{\text{init}})}{a} \right) \right] + \eta V_{\text{init}}, \quad (20)$$

For **BP5-FD**, initial values for displacements and velocities in the medium must be specified. We assume these are initially zero everywhere in the domain (i.e. we assume displacements are measured with respect to the prestressed equilibrium configuration), namely,

$$u_j(x_1, x_2, x_3, 0) = \dot{u}_j(x_1, x_2, x_3, 0) = 0, \quad j = 1, 2, 3. \quad (21)$$

The scalar pre-stress τ^0 is chosen as the steady-state stress:

$$\tau^0 = \bar{\sigma}_n a \sinh^{-1} \left[\frac{V_{\text{init}}}{2V_0} \exp \left(\frac{f_0 + b \ln(V_0/V_{\text{init}})}{a} \right) \right], \quad (22)$$

To break the symmetry of the problem and facilitate comparisons of different simulations, we choose a square region with a width, w , at one end of the VW region, as a favorable location for nucleation of the first and subsequent seismic events. For this purpose, we assign a smaller critical slip distance ($L = 0.13$ m) and impose a higher initial slip rate along the x_2 -direction ($V_i = 0.01$ m/s) within this square region while keeping the initial state variable $\theta(x_2, x_3, 0)$ unchanged, which means that a higher pre-stress along the x_2 -direction is required:

$$\tau_i^0 = \bar{\sigma}_n a \sinh^{-1} \left[\frac{V_i}{2V_0} \exp \left(\frac{f_0 + b \ln(V_0/V_{\text{init}})}{a} \right) \right] + \delta\tau, \quad (23)$$

where $\delta\tau = \eta V_i$ for BP5-QD and $\delta\tau = 0$ for BP5-FD. This initial condition should lead to an immediate initiation of the first seismic event.

Equations (1)–(3), along with interface conditions (5), (7), (10), (15), and initial conditions (19 or 21), (16), (18), (20 or 22) are solved over the time period $0 \leq t \leq t_f$, where t_f is a

Table 1: Parameter values used in this benchmark problem

Parameter	Definition	Value, Units
ρ	density	2670 kg/m ³
c_s	shear wave speed	3.464 km/s
ν	Poisson's ratio	0.25
a_0	rate-and-state parameter	0.004
a_{\max}	rate-and-state parameter	0.04
b_0	rate-and-state parameter	0.03
$\bar{\sigma}_n$	effective normal stress	25 MPa
L	critical slip distance	0.14 m/0.13 m [†]
V_p	plate rate	10 ⁻⁹ m/s
V_{init}	initial slip rate	10 ⁻⁹ m/s
V_0	reference slip rate	10 ⁻⁶ m/s
f_0	reference friction coefficient	0.6
h_s	width of shallow VS zone	2 km
h_t	width of VW-VS transition zone	2 km
H	width of uniform VW region	12 km
l	length of uniform VW region	60 km
W_f	width of rate-and-state fault	40 km
l_f	length of rate-and-state fault	100 km
w	width of favorable nucleation zone	12 km
Δz	suggested cell size	1000 m
t_f	final simulation time	1800 years

[†] the value of L in the favorable nucleation zone.

specified final simulation time. All necessary parameter values for this benchmark problem are given in Table 1.

Because computational efficiency for 3D problems demands a large cell size, we have changed some model parameters from BP1 in order to resolve relevant physical length scales. At a rupture speed of 0^+ , the quasi-static process zone, Λ_0 , is expressed as:

$$\Lambda_0 = C \frac{\mu L}{b \bar{\sigma}_n}, \quad (24)$$

where C is a constant on the order of 1. Another important length scale, the nucleation zone size, h , is expressed as:

$$h = \frac{\pi}{2} \frac{\mu b L}{(b - a)^2 \bar{\sigma}_n}. \quad (25)$$

With the provided model values, the process zone Λ_0 and h are uniform over most of the VW region, with a size of ≈ 6.0 km and ≈ 12.5 km, respectively. Within the favorable nucleation zone, Λ_0 and h are ≈ 5.6 km and ≈ 11.6 km.

We suggest using a cell size of $\Delta z = 1000$ m for the simulations; results from simulations using smaller, if feasible, or larger cell sizes are welcome. For a cell size of 1000 m, Λ_0 is resolved by ~ 6 grid points and h by ~ 12 grid points. For methods that use multiple degrees of freedom along cell edges/faces, please take $\Delta z = \text{edge length} / \text{number of unique degrees of freedom}$. For instance, for a high-order finite element method, if Δh is the edge length and N the polynomial order then $\Delta z = \Delta h / N$.

4 Benchmark Output

We request five types of data output, if available, for this benchmark:

- (1) On-fault time series (section 4.1)
- (2) Off-fault time series (section 4.2)
- (3) Source parameter time series (section 4.3)
- (4) Earthquake catalog (section 4.4)
- (5) Slip and stress evolution profile (section 4.5)
- (6) Rupture time contour for first event (section 4.6)

The location information relevant to these outputs are shown in Fig. 2. Data files for (1), (2), (3) and (6) are uploaded to the SCEC code verification web server (section 5). Information on how to share output (4) and (5) is detailed in sections 4.4 and 4.5.

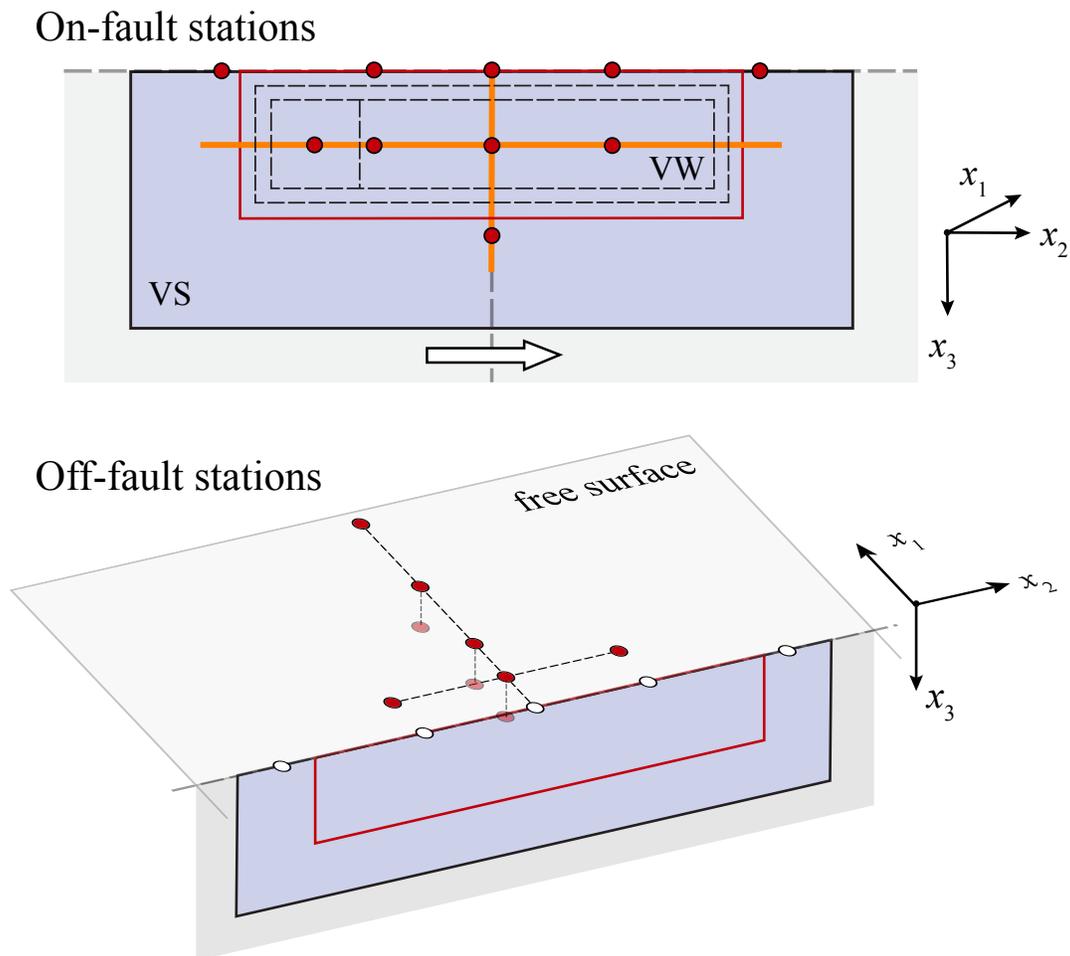


Figure 2: Observation points, lines, and region for model outputs. Local time series is output at (top) on-fault and (bottom) off-fault points (red). Slip and stress evolution are output along two cross-section lines (orange). The region outlined in red is used for estimating source parameter time series and rupture time contour. Dashed rectangles indicate fault areas with different frictional properties, shown in Figure 1.

4.1 On-fault Time Series Output

You need to upload on-fault ($x_1 = 0$) time series files, which give slip components s_2 and s_3 , base 10 log of the components of slip rate V_2 and V_3 , base 10 log of the state variable (i.e. $\log_{10}(\theta)$), and shear stress components τ and τ_z , for each on-fault station at representative time steps. We define the simulation periods as either aseismic (when $\max(V) < 10^{-3}$ m/s, where $\max(V)$ is the maximum of the norm of the slip velocity vector over the entire fault) or seismic (when $\max(V) \geq 10^{-3}$ m/s). When outputting modeling results, use larger time intervals (e.g., ~ 0.1 yr) during aseismic periods and smaller time intervals (e.g., ~ 0.1 s) during seismic periods. More variable time steps are OK. Please keep the total number of time steps in the data file on the order of 10^4 – 10^5 .

Time series data is supplied as ASCII files, one file for each station. There are 10 observational points on the fault (including 5 at the surface fault trace), as follows:

1. fltst_strk-36dp+00: $x_2 = -36$ km, $x_3 = 0$ km;
2. fltst_strk-16dp+00: $x_2 = -16$ km, $x_3 = 0$ km;
3. fltst_strk+00dp+00: $x_2 = 0$ km, $x_3 = 0$ km;
4. fltst_strk+16dp+00: $x_2 = 16$ km, $x_3 = 0$ km;
5. fltst_strk+36dp+00: $x_2 = 36$ km, $x_3 = 0$ km;
6. fltst_strk-24dp+10: $x_2 = -24$ km, $x_3 = 10$ km;
7. fltst_strk-16dp+10: $x_2 = -16$ km, $x_3 = 10$ km;
8. fltst_strk+00dp+10: $x_2 = 0$ km, $x_3 = 10$ km;
9. fltst_strk+16dp+10: $x_2 = 16$ km, $x_3 = 10$ km;
10. fltst_strk+00dp+22: $x_2 = 0$ km, $x_3 = 22$ km.

Each time series has 8 data fields, as follows.

Field Name	Description, Units and Sign Convention
t	Time (s)
slip_2	Horizontal component of slip (m). Positive for right-lateral motion.
slip_3	Vertical component of slip (m). Positive for + side moving downward.
slip_rate_2	\log_{10} of the amplitude of the horizontal component of slip-rate (\log_{10} m/s), which is positive for right-lateral motion.
slip_rate_3	\log_{10} of the amplitude of the vertical component of slip-rate (\log_{10} m/s), which is positive for + side moving downward.
shear_stress_2	Horizontal component of shear stress (MPa), which is positive for shear stress that tends to cause right-lateral motion.
shear_stress_3	Vertical component of shear stress (MPa), which is positive for shear stress that tends to cause + side to move downward.
state	\log_{10} of state variable (\log_{10} s).

The on-fault time series file consists of three sections, as follows:

File Section	Description
File Header	<p>A series of lines, each beginning with a # symbol, that give the following information:</p> <ul style="list-style-type: none"> • Benchmark problem (BP5-QD) • Code name • Code version (optional) • Modeler • Date • Node spacing or element size • Station location • Minimum time step (optional) • Maximum time step (optional) • Number of time steps in file (optional) • Anything else you think is relevant (optional) • Descriptions of data columns (8 lines) • Anything else you think is relevant
Field List	<p>A single line, which lists the names of the 8 data fields, in column order, separated by spaces. It should be:</p> <pre>t slip_2 slip_3 slip_rate_2 slip_rate_3 shear_stress_2 shear_stress_3 state (all on one line).</pre> <p>The server examines this line to check that your file contains the correct data fields.</p>
Time History	<p>A series of lines. Each line contains 8 numbers, which give the data values for a single time step. The lines must appear in order of increasing time.</p> <p>Make sure to use double-precision when saving all fields.</p> <p>C/C++ users: We recommend using 21.13E or 21.13e floating-point format for the time field and 14.6E or 14.6e format for all other data fields.</p> <p>Fortran users: We recommend using E22.14 or 1PE22.13 floating-point format for the time field and E15.7 or 1PE15.6 format for other data fields. The server accepts most common numeric formats. If the server cannot understand your file, you will see an error message when you attempt to upload the file.</p>

Here is an example of an on-fault time-series file, with invented data.

```
# This is the file header:
# problem=SEAS Benchmark BP5-QD
# code=MYcode
# version=1.0
# modeler=A. Modeler
# date=2019/12/01
# element_size=1000 m
# location= on fault, 0km along strike, 10km depth
# minimum_time_step=0.1
# maximum_time_step=3.157e6
# num_time_steps=2400
# Column #1 = Time (s)
# Column #2 = Slip_2 (m)
# Column #3 = Slip_3 (m)
# Column #4 = Slip_rate_2 (log10 m/s)
# Column #5 = Slip_rate_3 (log10 m/s)
# Column #6 = Shear_stress_2 (MPa)
```

```

# Column #7 = Shear_stress_3 (MPa)
# Column #8 = State (log10 s)
# The line below lists the names of the data fields
t slip_2 slip_3 slip_rate_2 slip_rate_3 shear_stress_2 shear_stress_3 state
# Here is the time-series data.
0.000000E+00 0.000000E+00 -9.000000E+00 3.000000E+01 7.000000E+00 ...
5.000000E-02 0.000000E+00 -9.000000E+00 3.000000E+01 7.000000E+00 ...
1.000000E-01 0.000000E+00 -9.000000E+00 3.000000E+01 7.000000E+00 ...
1.500000E-01 0.000000E+00 -9.000000E+00 3.000000E+01 7.000000E+00 ...
# ... and so on.

```

4.2 Off-fault Time Series Output

You need to upload time series files for off-fault stations, which give three components of displacement u_1 , u_2 , and u_3 , and of velocity v_1 , v_2 , and v_3 (not base 10 log), for each station at representative time steps. Please use the same time steps for outputting the on-fault and off-fault time series.

Time series data is supplied as ASCII files, one file for each station. There are six off-fault observational points on the surface ($x_3 = 0$ km), as follows:

1. bl kst_strk-16fn+08dp+00: $x_2 = -16$ km, $x_1 = 8$ km;
2. bl kst_strk+00fn+08dp+00: $x_2 = 0$ km, $x_1 = 8$ km;
3. bl kst_strk+16fn+08dp+00: $x_2 = 16$ km, $x_1 = 8$ km;
4. bl kst_strk+00fn+16dp+00: $x_2 = 0$ km, $x_1 = 16$ km;
5. bl kst_strk+00fn+32dp+00: $x_2 = 0$ km, $x_1 = 32$ km;
6. bl kst_strk+00fn+48dp+00: $x_2 = 0$ km, $x_1 = 48$ km.

There are three additional off-fault stations at depth ($x_2 = 0$ km), as follows:

7. bl kst_strk+00fn+08dp+10: $x_1 = 8$ km, $x_3 = 10$ km;
8. bl kst_strk+00fn+16dp+10: $x_1 = 16$ km, $x_3 = 10$ km;
9. bl kst_strk+00fn+32dp+10: $x_1 = 32$ km, $x_3 = 10$ km.

Each time series has 7 data fields, as follows.

Field Name	Description, Units and Sign Convention
t	Time (s)
di sp_1	Fault-perpendicular (x_1) component of displacement (m). Positive for extensional motion.
di sp_2	Fault-parallel (x_2) component of displacement (m). Positive for right-lateral motion.
di sp_3	Vertical (x_3) component of displacement (m). Positive for + side moving downward.
vel _1	x_1 component of velocity (m/s), which is positive for extensional motion.
vel _2	x_2 component of velocity (m/s), which is positive for right-lateral motion.
vel _3	x_3 component of velocity (m/s), which is positive for + side moving downward.

The off-fault time series file consists of three sections, as follows:

File Section	Description
File Header	<p>A series of lines, each beginning with a # symbol, that give the following information:</p> <ul style="list-style-type: none"> • Benchmark problem (BP5-QD) • Code name • Code version (optional) • Modeler • Date • Node spacing or element size • Station location • Minimum time step (optional) • Maximum time step (optional) • Number of time steps in file (optional) • Anything else you think is relevant (optional) • Descriptions of data columns (7 lines) • Anything else you think is relevant
Field List	<p>A single line, which lists the names of the 7 data fields, in column order, separated by spaces. It should be: t di sp_1 di sp_2 di sp_3 vel _1 vel _2 vel _3 (all on one line). The server examines this line to check that your file contains the correct data fields.</p>
Time History	<p>A series of lines. Each line contains 8 numbers, which give the data values for a single time step. The lines must appear in order of increasing time. Make sure to use double-precision when saving all fields. C/C++ users: We recommend using 21.13E or 21.13e floating-point format for the time and velocity fields, and 14.6E or 14.6e format for all other data fields. Fortran users: We recommend using E22.14 or 1PE22.13 floating-point format for the time and velocity fields, and E15.7 or 1PE15.6 format for other data fields. The server accepts most common numeric formats. If the server cannot understand your file, you will see an error message when you attempt to upload the file.</p>

Here is an example of an on-fault time-series file, with invented data.

```
# This is the file header:
# problem=SEAS Benchmark BP5-QD
# code=MYcode
# version=1.0
# modeler=A. Modeler
# date=2019/12/01
# element_size=1000 m
# location= off fault, 0km along strike, 8km away from the fault, 0km depth
# minimum_time_step=0.1
# maximum_time_step=3.157e6
# num_time_steps=2400
# Column #1 = Time (s)
# Column #2 = Displacement_1 (m)
# Column #3 = Displacement_2 (m)
# Column #4 = Displacement_3 (m)
# Column #5 = Velocity_1 (m/s)
```

```

# Column #6 = Velocity_2 (m/s)
# Column #7 = Velocity_3 (m/s)
# The line below lists the names of the data fields
t disp_1 disp_2 disp_3 vel_1 vel_2 vel_3
# Here is the time-series data.
0.000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 1.000000E-20 1.000000E-9 1.000000E-20
5.000000E-02 0.000000E+00 0.000000E+00 0.000000E+00 1.000000E-20 1.000000E-9 1.000000E-20
1.000000E-01 0.000000E+00 0.000000E+00 0.000000E+00 1.000000E-20 1.000000E-9 1.000000E-20
# ... and so on.

```

4.3 Source Parameter Time Series Output

You need to upload a file named `global.dat`, which includes time series of two global source variables, maximum amplitude of slip rates

$$V_{\max} = \max_{(x_2, x_3) \in 2A} V$$

and moment rates

$$M_t = \int_A \mu V dA$$

for the domain A that surrounds the velocity-weakening patch plus twice the transition zone width in every direction, i.e. $(|x_2| \leq l/2 + 2h_t) \cap (0 \leq x_3 \leq H + h_s + 3h_t)$ (shown in Fig. 2). Upload data corresponding to the same time steps you used for section 4.1.

Here is an example of a source parameter time-series file, with invented data.

```

# This is the file header:
# problem=SEAS Benchmark BP5-QD
# code=MYcode
# version=1.0
# modeler=A.Modeler
# date=2019/12/01
# element_size=1000 m
# location= VW patch + transition zone
# minimum_time_step=0.1
# maximum_time_step=3.157e6
# num_time_steps=2400
# Column #1 = Time (s)
# Column #2 = Max_slip_rate (log10 m/s)
# Column #3 = Moment_rate (N-m/s)
# The line below lists the names of the data fields
t max_slip_rate moment_rate
# Here is the time-series data.
0.000000E+00 0.000000E+00 -9.000000E+00
5.000000E-02 0.000000E+00 -9.000000E+00
1.000000E-01 0.000000E+00 -9.000000E+00

```

```
1.500000E-01 0.000000E+00 -9.000000E+00
# ... and so on.
```

4.4 Earthquake Catalog Output

The earthquake catalog output file with a name `catalog.dat` is a single ASCII file that includes characteristics of all seismic events in the simulation. We define an event to start when the maximum slip rate exceeds 10^{-3} m/s, and to be over when maximum slip rate dips below 10^{-3} m/s for a time duration of at least 10 seconds (to avoid inaccurate estimate of event termination in the presence of slip rate fluctuation at the end of rupture). For each earthquake in the entire simulation period, the file contains the event number, the event initiation time, event termination time, the total rupture area R , average stress in the rupture area τ_{avg} at the beginning and end of the event (so stress drops can be computed from the two values), and the average coseismic slip in the rupture area s_{avg} .

We consider the rupture domain Ω_R as those locations on the fault where slip rates have exceeded 10^{-3} m/s at any time during the event, with $R = |\Omega_R|$ being the total rupture area. We define τ_{avg} as the amplitude of spatially averaged stress vector $\boldsymbol{\tau}$ within the rupture area:

$$\tau_{\text{avg}} = \frac{1}{R} \left| \int_{\Omega_R} \boldsymbol{\tau} d\Omega_R \right|.$$

Similarly, we define s_{avg} as the amplitude of spatially averaged slip vector \mathbf{s} within the rupture domain:

$$s_{\text{avg}} = \frac{1}{R} \left| \int_{\Omega_R} \mathbf{s} d\Omega_R \right|.$$

NOTE: Please upload this data to a Dropbox folder that will be shared with you (or send request to bae@uoregon.edu). Please ensure that the file is ~ 10 s of MBs or less.

The data file has 7 data fields, as follows:

Field Name	Description, Units and Sign Convention
<code>event_no</code>	Event number.
<code>t_start</code>	Time event starts (s).
<code>t_end</code>	Time event ends (s).
<code>rup_area</code>	Rupture area (m^2).
<code>avg_stress_start</code>	Spatial average of stress within rupture area at <code>t_start</code> (MPa).
<code>avg_stress_end</code>	Spatial average of stress within rupture area at <code>t_end</code> (MPa).
<code>avg_slip</code>	Average slip in rupture area (m).

The data output consists of three sections, as follows:

File Section	Description
File Header	A series of lines, each beginning with a # symbol, that give the following information: <ul style="list-style-type: none"> • Benchmark problem (BP5-QD) • Modeler • Date • Code • Code version (if desired) • Node spacing or element size • Descriptions of data fields (1 line) • Anything else you think is relevant (e.g. computational domain size)
Field List	One line listing the 7 data fields on one line, separated by spaces. It should be: event_no t_start t_end rup_area avg_stress_start avg_stress_end avg_slip (all on one line).
Catalog	A series of lines. Make sure to use double-precision when saving all fields. C/C++ users: We recommend using 21.13E or 21.13e floating-point format for the time field and 14.6E or 14.6e format for all other data fields. Fortran users: We recommend using E22.14 or 1PE22.13 floating-point format for the time field and E15.7 or 1PE15.6 format for other data fields.

Here is an example of a catalog file, with invented data.

```
# This is the file header:
# problem=SEAS Benchmark BP5-QD
# author=A. Modeler
# date=2019/12/01
# code=MyCode
# code_version=3.7
# element_size=1000 m
# Column #1 = Event number
# Column #2 = Event start time (s)
# Column #3 = Event end time (s)
# Column #6 = Rupture area (m^2)
# Column #7 = Stress avg at start time (MPa)
# Column #8 = Stress avg at end time (MPa)
# Column #9 = Avg slip in rupture area (m)
# Computational domain size: depth 100 km, distance off fault 100 km
# The line below lists the names of the data fields
event_no t_start t_end rup_area avg_stress_start avg_stress_end avg_slip
# Here are the data
1 3.15500000E+07 3.15500200E+07 1.800000E+09 ... 5.000000E+00
2 4.70000000E+09 4.70000002E+09 1.800000E+09 ... 4.800000E+00
3 9.45000000E+09 9.45000002E+09 1.800000E+09 ... 4.820000E+00
# ... and so on.
```

4.5 Slip and Stress Evolution Output

The slip and stress evolution output files with the names

```
slip_2_depth.dat
slip_2_strike.dat
stress_2_depth.dat
stress_2_strike.dat
slip_3_depth.dat
slip_3_strike.dat
stress_3_depth.dat
stress_3_strike.dat
```

are 8 ASCII files that record the spatial distribution of slip and stress (both horizontal and vertical components) on a subset of fault nodes at one-dimensional cross sections (either a specified distance along strike OR along depth), at representative time steps during the aseismic and seismic phases of the simulation. Data can be saved using representative time intervals of ~ 5 yr and ~ 1 s for results in aseismic and seismic phases, respectively, or with variable time steps. Either way, data will be interpolated to plot slip every 5 yr during the aseismic phase, and every 1 s during the coseismic phase.

The data should include nodes with a spacing of ~ 1000 m (or larger for simulations with larger cells) along depth ($0 \leq x_3 \leq W_f$) or along strike ($-l/2 - 8 \text{ km} \leq x_2 \leq l/2 + 8 \text{ km}$). The files should also contain the time series of maximum slip rate amplitude (taken over the entire fault), so that we can precisely differentiate aseismic and seismic phases. We request a total of 8 data files containing slip and stress data at two one-dimensional cross sections: one along-dip profile at $x_2 = 0 \text{ km}$ and one along-strike profile at $x_3 = 10 \text{ km}$ (see the orange lines in Figure 2).

NOTE: Please upload this data to a Dropbox folder that will be shared with you (or send a request to bae@uoregon.edu).

Each data file has 4 data fields, as follows:

Field Name	Description, Units and Sign Convention
x2 OR x3	Strike (m) at ~ 1000 m increments from -38 km to 38 km OR Depth (m) at ~ 1000 m increments from 0 km to 40 km
t	Time (s). Nonuniform time steps.
max_slip_rate	The \log_{10} of maximum amplitude of slip-rate (taken over the entire fault) ($\log_{10} \text{ m/s}$).
slip_2 OR slip_3 OR stress_2 OR stress_3	Horizontal OR vertical component of slip (m) (positive for right-lateral motion OR for + side moving downward, respectively) OR horizontal OR vertical component of stress (in MPa).

The data output consists of three sections, as follows:

File Section	Description
File Header	<p>A series of lines, each beginning with a # symbol, that give the following information:</p> <ul style="list-style-type: none"> • Benchmark problem (BP5-QD) • Modeler • Date • Code • Code version (if desired) • Node spacing or element size • Descriptions of data fields (4 lines) • Anything else you think is relevant (e.g. computational domain size)
Field List	<p>Four lines. The first line lists either x_2 OR x_3. The next two lines lists the time steps and max slip rate (respectively). The last line lists which component of slip or stress. It should be:</p> <pre>x2 OR x3 t max_slip_rate slip_2 OR slip_3 OR stress_2 OR stress_3</pre>
Slip History	<p>A series of lines that form a 2-dimensional array of rows and columns. The first row/line lists the numbers 0, 0 (to maintain a consistent array size), followed by the spatial nodes with increasing distance along strike OR depth as you go across the row. Starting from the second row/line, each row/line contains time, maximum slip rate, and slip OR stress at all nodes at the time. These lines appear in order of increasing time (from top to bottom) and slip OR stress is recorded with increasing distance along strike or depth (from left to right).</p> <p>Make sure to use double-precision when saving all fields.</p> <p>C/C++ users: We recommend using 21.13E or 21.13e floating-point format for the time field and 14.6E or 14.6e format for all other data fields.</p> <p>Fortran users: We recommend using E22.14 or 1PE22.13 floating-point format for the time field and E15.7 or 1PE15.6 format for other data fields.</p>

Note that x_2 or x_3 should appear in the first row, preceded by two zero numbers, for nodes with a spacing of ~ 1000 m. Time and maximum slip rate should appear as two single columns that start on the second row, with time increasing as you go down. Slip or stress history (the remaining block) is represented by a two-dimensional array with time increasing as you go down the rows/lines, and either x_2 or x_3 increasing as you go across the columns (~ 77 and 41 columns, respectively). For example, the output in `slip_2_strike.dat` is a two-dimensional matrix of the form:

$$\begin{bmatrix} 0 & 0 & x_2 \\ T & \max(V) & \text{slip} \end{bmatrix}$$

The matrix should be of size $(N_t + 1, \sim 79)$, where N_t is the total number of time steps. This means that you output slip at selected nodes at one time step and move on to the next time step. (To keep the file on the order of 10s of MB, N_t should be on the order of 10,000).

Here is an example of a slip-evolution file for `slip_2_strike.dat`, with invented data.

```
# This is the file header:
# problem=SEAS Benchmark BP5-QD
```

```

# author=A. Model er
# date=2019/12/01
# code=MyCode
# code_versi on=3. 7
# el ement_si ze=1000 m
# Row #1 = Strike (m) with two zeros first
# Column #1 = Time (s)
# Column #2 = Max slip rate (log10 m/s)
# Columns #3-83 = Horizontal slip along depth (Slip_2) (m)
# Computational domain size: -100km < x1 < 100km, -100km < x2 < 100km, -100km < x3 < 100km
# The line below lists the names of the data fields
x2
t
max_slip_rate
slip_2
# Here are the data
0.000000E+00 0.000000E+00 -3.80000E+04 -3.70000E+04 ... 3.600000E+04
0.000000E+00 -9.000000E+00 0.000000E+00 0.000000E+00 ... 0.000000E+00
3.140000E+05 -9.000000E+00 1.340000E-05 1.340000E-05 ... 3.140000E-05
1.227000E+07 -9.000000E+00 1.560000E-05 1.560000E-05 ... 1.220000E-02
4.690000E+07 -9.000000E+00 1.580000E-05 1.580000E-05 ... 4.680000E-02
...
9.467078E+10 -4.500000E-01 9.050000E+01 9.050000E+01 ... 9.461000E+01

```

4.6 Rupture Time Contour Output

The rupture time contour output with a name `rupture.dat` is a single ASCII files that record the spatial distribution of arrival time of coseismic rupture front for the 1st earthquake in the simulation. We request the rupture time contour within a larger region that includes the VW zone, $(|x_2| \leq l/2 + 2h_t) \cap (0 \leq x_3 \leq H + h_s + 3h_t)$ (shown in Fig. 2). At each node, the rupture time value is chosen as the first time instance when local slip rate amplitude reaches 1 mm/s. This data needs to be uploaded to the web server.

Each data file has 3 data fields, as follows:

Field Name	Description, Units and Sign Convention
x2	Distance along strike (m). Positive means a location to the right of the origin. The values range from -34 km to 34 km.
x3	Distance down-dip (m). Positive means a location below the origin. The values range from 0 km to 20 km.
t	Rupture time (s). This is the time at which slip rate first reaches a value greater than 1 mm/s. If this node never ruptures, use the value $1.0E+09$.

The data output consists of three sections, as follows:

File Section	Description
File Header	<p>A series of lines, each beginning with a # symbol, that give the following information:</p> <ul style="list-style-type: none"> • Benchmark problem (BP5-QD) • Modeler • Date • Code • Code version (if desired) • Node spacing or element size • Descriptions of data fields (1 line) • Anything else you think is relevant (e.g. computational domain size)
Field List	<p>A single line, which lists the names of the 3 data fields on one line, separated by spaces. It should be: x2 x3 t (on one line).</p>
Rupture History	<p>A series of lines. Each line contains three numbers, which give the (x2,x3) coordinates of a node on the fault surface, and the time t at which that node ruptures.</p> <p>Make sure to use double-precision when saving all fields.</p> <p>C/C++ users: We recommend using 21.13E or 21.13e floating-point format.</p> <p>Fortran users: We recommend using E22.14 or 1PE22.13 floating-point format.</p> <p>If a node never ruptures, the time should be given as 1.0E+09.</p>

Note: The nodes may appear in any order. The nodes do not have to form a rectangular grid, or any other regular pattern.

Note: When you upload a file, the server constructs the Delaunay triangulation of your nodes. Then, it uses the Delaunay triangulation to interpolate the rupture times over the entire fault surface. Finally, it uses the interpolated rupture times to draw a series of contour curves at intervals of 0.5 seconds.

Here is an example of a rupture time file, with invented data.

```
# This is the file header:
# problem=SEAS Benchmark BP5-QD
# author=A. Modeler
# date=2019/12/01
# code=MyCode
# code_version=3.7
# element_size=1000 m
# Column #1 = x2 (m)
# Column #2 = x3 (m)
# Column #3 = time (s)
# Computational domain size: depth 100 km, distance off fault 100 km
# The line below lists the names of the data fields
x2 x3 t
# Here are the data
-3.40000E+04 0.000000E+04 0.000000E+00
```

```
-3.30000E+04 0.10000E+04 3.14000E+05  
3.00000E+04 4.20000E+04 6.44000E+06  
3.00000E+04 4.00000E+04 3.14000E+06  
4.00000E+04 5.10000E+04 2.14000E+06  
4.00000E+04 5.20000E+04 1.40000E+06  
# ... and so on.
```

5 Using the Web Server

The web server lets you upload your modeling results (section 4). Once uploaded, you and other modelers can view the data in various ways.

5.1 Logging in and Selecting a Problem

To log in, start your web browser and go to the home page at:

```
https://strike.scec.org/cvws/cgi-bin/seas.cgi
```

Click on “Upload Files,” and then log in using your user name and password. Remember that passwords are case-sensitive. You are then presented with a list of code validation problems. Find the problem you want, and click the “Select” button. You will then see a list of files for the problem.

5.2 Navigating the Site

You navigate through the site by clicking buttons on the web pages. Avoid using your browser’s Back or Forward buttons. If you use the Back or Forward buttons, you may get error messages from your browser.

5.3 Uploading Files

To upload a file, do the following steps:

- Find the file you want to upload, and click “Select.” The server displays a page for you to upload the file.
- Select the data file on your computer. The exact method for file selection varies depending on operating system and web browser.
- Click on “Click Once to Upload.” The file you selected is uploaded to the server.

When you upload a file, the web server immediately checks for correct formatting. There are three possible results:

- If the file is correctly formatted, the server displays a page noting the successful upload.
- If the file contains errors, the server displays an error log. The error log lists the errors that were detected in the file, each identified as specifically as possible.

- If the file is correctly formatted, but is questionable in some way (for example, a missing time step), then the server displays a warning log, which describes the problem.

When uploading time series files, the website may issue a warning that the time series cannot be filtered. Modelers should ignore this warning. After uploading a file, the file list shows the date and time that you uploaded the file. Remember that any file you upload will be visible to anyone who has access to the web site.

Additional help is available by clicking the “Help” link in the upper right corner of the webpage. Modelers who want to upload multiple versions of the benchmark (for example, using different element sizes), can do so using the “Change Version” feature of the website, which is described in the help screens. Direct further questions to Michael Barall.

5.4 Graphing, Viewing, and Deleting Files

After uploading a file, additional functions become available. These functions let you graph, view, or delete the uploaded file.

Graphing: To graph a file, find the file you want and click “Graph.” For a time-series file, the server displays graphs of all the data fields in the file. At the bottom of each graph page, there is a box you can use to adjust graphing preferences. Graphing a file is a good way to check that the server is interpreting your data as you intended.

Viewing: To view the text of a file, find the file you want and click “View.”

Deleting: To delete a file from the server, find the file you want and click “Delete.” The server displays a page asking you to confirm the deletion.

6 Benchmark Tips

Numerical boundary conditions (to truncate the half-space in x_1 , x_2 , and x_3 directions when defining the computational domain) will most likely change results at least quantitatively, or even qualitatively. We suggest extending these boundaries until you see results appear independent of the computational domain size. We prefer participants to use the cell size suggested in Table 1 and welcome results for different spatial resolutions. Each person can submit (at most) results from two different spatial resolutions and two different computational domain sizes.