SEAS Benchmark Problems BP7-QD/FD-A/S

Valère Lambert, Junle Jiang and Brittany Erickson

November, 8 2021

Benchmark problems BP7 (-QD: quasi-dynamic; -FD: fully dynamic; -A: aging law; -S: slip law) are a set of three-dimensional (3D) problems in a whole-space with a single circular asperity governed by velocity-weakening friction and either the aging or slip evolution law. 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, whole-space defined by:

$$\mathbf{x} = (x_1, x_2, x_3) \in \mathbb{R}^3.,$$

A planar 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. The medium deforms with respect to a prestressed reference configuration at time $t = 0$; displacements and strains are measured with respect to this reference configuration. The prestress tensor is $\boldsymbol{\sigma}^0$, which satisfies the equilibrium equation.

We assume 3D motion, letting $u_i = u_i(\mathbf{x}, t), i = 1, 2, 3$ denote the displacement in the i-direction. For fully dynamic problems **BP7-FD-A/S**, motion is governed by the momentum balance equation:

$$\rho \frac{\partial^2 \mathbf{u}}{\partial t^2} = \nabla \cdot \boldsymbol{\sigma}, \tag{1}$$

where ρ is the material density. For quasi-dynamic problems **BP7-QD-A/S**, inertia is neglected and motion is governed by the equilibrium equation:

$$0 = \nabla \cdot \boldsymbol{\sigma}. \tag{2}$$

The solution to (1,2) can be expressed as the superposition of a rigid body translation with zero associated stress changes, denoted $\mathbf{u}^{\mathrm{rigid}}$, and the displacements, strains and stress changes due to fault slip differing from steady slip provided by the rigid body translation. The rigid body translation is different for the two sub-domains, and the discontinuity in that translational motion across the fault is the long-term slip.

The stresses are given by $\sigma_{ij} = \sigma_{ij}^0 + \Delta\sigma_{ij}$, the sum of the initial stress and the elastic stress changes. Deformation with respect to the reference configuration is assumed to be linear elastic. Hooke's law relates stresses to strains by:

$$\sigma_{ij} = \sigma_{ij}^0 + \lambda \epsilon_{kk} \delta_{ij} + 2\mu \epsilon_{ij}, \tag{3}$$

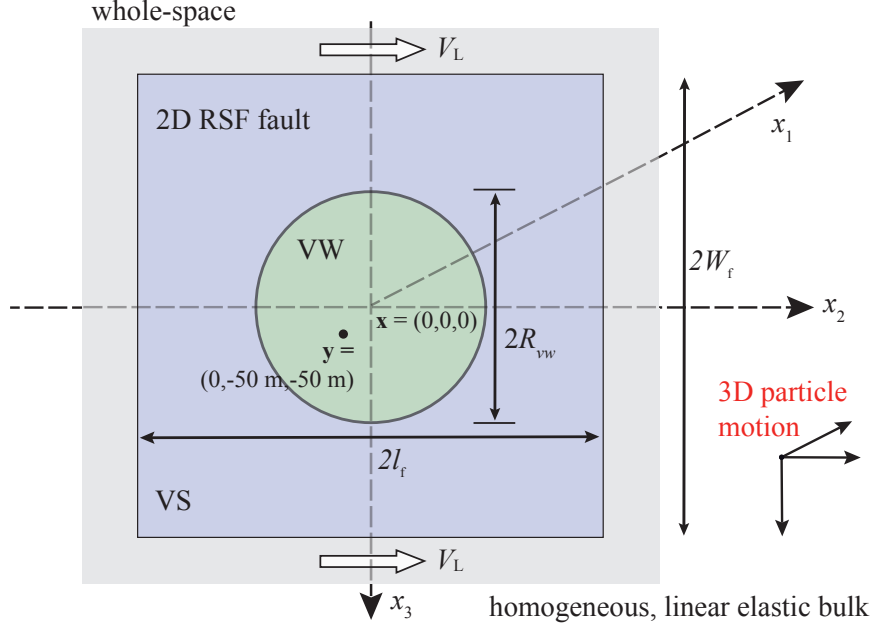for shear modulus μ and Lamé's first parameter λ.

Figure 1: This benchmark considers 3D motion with a planar fault embedded in a homogeneous, linear elastic whole-space. The fault is governed by rate-and-state friction in the region $|x_3| \leq W_f$ and $|x_2| \leq l_f$, outside of which it creeps at an imposed constant horizontal rate V_L (gray) which is constant with the rigid body plate rate V_p. The velocity-weakening region (the circle in light green; $\sqrt{x_2^2 + x_3^2} \leq R$) is surrounded by velocity-strengthening regions (blue). The first rupture is nucleated in a smooth manner about the hypocenter y.

2   Boundary and Interface Conditions

The infinite domain problem requires conditions at infinity. For the quasi-dynamic problem **BP7-QD** we require that stress changes $\Delta\sigma_{ij}$ and displacement changes (from rigid body translation), $\mathbf{u} - \mathbf{u}^{\text{rigid}}$ vanish at infinity ($x_1 \to \pm\infty$, $x_2 \to \pm\infty$, $x_3 \to \pm\infty$). For the fully dynamic problem **BP7-FD**, we permit only outgoing waves through the Sommerfield radiation condition at infinity (i.e. only solutions that scatter energy to infinity are permitted, no energy may be radiated into the source region from infinity) [1, 2]. The rigid body translation we specify as:

$$u_1^{\pm,\text{rigid}}(t) = 0 \tag{4a}$$

$$u_2^{\pm,\text{rigid}}(t) = \pm V_p t/2, \tag{4b}$$

$$u_3^{\pm,\text{rigid}}(t) = 0 \tag{4c}$$

where V_p is the plate rate.

At $x_1 = 0$, the fault defines the interface and we supplement equations (1)-(3) with six interface conditions. We assume a "no-opening condition" on the fault, namely that:

$$u_1(0^+, x_2, x_3, t) = u_1(0^-, x_2, x_3, t), \tag{5}$$

and define the slip vector:

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

2

i.e. the jump in displacements in the x_2- and x_3-directions 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 define the slip velocity vector $\boldsymbol{V}$ in terms of the components:

$$V_j = \partial s_j / \partial t, \quad j = 2, 3, \tag{7}$$

letting $V = ||\boldsymbol{V}||$ denote the norm of the vector.

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

$$\sigma_{11}(0^+, x_2, x_3, t) = \sigma_{11}(0^-, x_2, x_3, t), \tag{8a}$$
$$\sigma_{21}(0^+, x_2, x_3, t) = \sigma_{21}(0^-, x_2, x_3, t), \tag{8b}$$
$$\sigma_{31}(0^+, x_2, x_3, t) = \sigma_{31}(0^-, x_2, x_3, t), \tag{8c}$$

The total normal stress is given by the prestress and does not change in time. We denote this as $\sigma = -\sigma_{11}^0$, which is positive in compression and spatially uniform.

The fault shear stress vector is given by:

$$\boldsymbol{\tau} = \begin{bmatrix} \tau_y \\ \tau_z \end{bmatrix}. \tag{9}$$

For **BP7-FD-A/S**, the fault shear stress $\boldsymbol{\tau}$ is the sum of the reference shear tractions $\boldsymbol{\tau}^0$, with components $\tau_y^0 = \sigma_{21}^0$ and $\tau_z^0 = \sigma_{31}^0$, a shear stress perturbation to nucleate the first rupture $\Delta\boldsymbol{\tau}^0$ (discussed futher in section 3.2), and the shear stress change $\Delta\boldsymbol{\tau}$ due to elastic deformation:

$$\boldsymbol{\tau} = \boldsymbol{\tau}^0 + \Delta\boldsymbol{\tau}^0 + \Delta\boldsymbol{\tau}, \quad \text{(BP7-FD)}. \tag{10}$$

For **BP7-QD-A/S**, the fault shear stress is the sum of the reference shear tractions $\boldsymbol{\tau}^0$, the shear stress perturbation to nucleate the first rupture $\Delta\boldsymbol{\tau}^0$, the shear stress change $\Delta\boldsymbol{\tau}$ due to (quasi-static) deformation, and the radiation damping approximation to inertia:

$$\boldsymbol{\tau} = \boldsymbol{\tau}^0 + \Delta\boldsymbol{\tau}^0 + \Delta\boldsymbol{\tau} - \eta\boldsymbol{V}, \quad \text{(BP7-QD)} \tag{11}$$

where $\eta = \mu/2c_\mathrm{s}$ is half the shear-wave impedance for shear wave speed $c_\mathrm{s} = \sqrt{\mu/\rho}$ and density ρ. Note that positive values of τ_y 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 in the positive x_3 direction and the $-$ side to move in the negative x_3-direction.

Within the frictional domain $(x_2, x_3) \in \Omega_f = (-l_\mathrm{f}, l_\mathrm{f}) \times (-W_\mathrm{f}, W_\mathrm{f})$ we impose rate-and-state friction where shear stress on the fault is equal to fault shear resistance $\boldsymbol{F}$, namely:

$$\boldsymbol{\tau} = \boldsymbol{F}(\boldsymbol{V}, \theta); \tag{12}$$

The fault shear resistance is defined as:

$$\boldsymbol{F} = \bar{\sigma} f(V, \theta) \frac{\mathbf{V}}{V}, \tag{13}$$

where θ is the state variable and $\bar{\sigma} = \sigma - p$ is the effective normal stress on the fault for pore-pressure p. For **BP7-QD/FD-A**, θ evolves according to the aging law:

$$\frac{d\theta}{dt} = 1 - \frac{V\theta}{D_\mathrm{RS}}. \tag{14}$$

3

For **BP7-QD/FD-S**, θ evolves according to the slip law:

$$\frac{d\theta}{dt} = -\frac{V\theta}{D_{\text{RS}}} \ln\left(\frac{V\theta}{D_{\text{RS}}}\right), \tag{15}$$

where D_{RS} is the characteristic slip distance. The friction coefficient f is given by a regularized formulation:

$$f(V, \theta) = a \sinh^{-1}\left[\frac{V}{2V_*} \exp\left(\frac{f_* + b\ln(V_*\theta/D_{\text{RS}})}{a}\right)\right], \tag{16}$$

with the reference friction coefficient f_* at a reference slip rate V_*, and rate-and-state direct effect and evolution parameters a and b, respectively.

For this benchmark, b is fixed as b_0 and a varies throughout Ω_f in order to define the velocity-weakening/strengthening regions (see Figure 1) as follows:

$$a(x_2, x_3) = \begin{cases} a_0, & (r \leq R_{\text{vw}}) \\ a_{\text{max}}, & (r > R_{\text{vw}}) \end{cases} \tag{17}$$

where $r = \sqrt{x_2^2 + x_3^2}$.

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

$$V_2(x_2, x_3, t) = V_L, \tag{18a}$$
$$V_3(x_2, x_3, t) = 0, \tag{18b}$$

where V_L is the imposed fault slip rate (equal to the plate rate V_p for this problem).

3   Initial Conditions and Simulation Time

The initial state and reference shear traction on the fault is chosen so that the model can start with a uniform fault slip rate, given by:

$$\mathbf{V} = \begin{bmatrix} V_{\text{init}} \\ V_{\text{zero}} \end{bmatrix}, \tag{19}$$

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

$$\boldsymbol{\tau}^0 = \tau^0 \cdot \mathbf{V}/V. \tag{20}$$

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

$$\theta(x_2, x_3, 0) = L/V_{\text{init}}. \tag{21}$$

For **BP7-QD-A/S**, we must specify an initial value for slip, which we take to be zero, namely:

$$s_j(x_2, x_3, t) = 0, \quad j = 2, 3. \tag{22}$$

The scalar reference traction τ^0 is chosen with respect to the steady-state stress:

$$\tau^0 = \bar{\sigma} a \sinh^{-1}\left[\frac{V_{\text{init}}}{2V_*} \exp\left(\frac{f_* + b\ln(V_*/V_{\text{init}})}{a}\right)\right] + \eta V_{\text{init}} \tag{23}$$

such that the actual shear stress on the fault is initially consistent with the steady-state shear resistance at the desired initial slip rate V_{init}.

For **BP7-FD-A/S**, 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,

$$u_j(x_1, x_2, x_3, 0) = \dot{u}_j(x_1, x_2, x_3, 0) = 0, \quad j = 1, 2, 3. \tag{24}$$

The scalar reference shear traction τ^0 is chosen as the steady-state stress:

$$\tau^0 = \bar{\sigma} a \sinh^{-1}\left[\frac{V_{\text{init}}}{2V_*} \exp\left(\frac{f_* + b\ln(V_*/V_{\text{init}})}{a}\right)\right], \tag{25}$$

3.1 Nucleation procedure

In order to nucleate the first rupture, a horizontal shear traction perturbation (i.e. a perturbation to τ_y) is imposed starting at $t = 0$:

$$\Delta\boldsymbol{\tau}^0 = \begin{bmatrix} \Delta\tau^0 \\ 0 \end{bmatrix}. \tag{26}$$

The perturbation smoothly grows in space and time from zero to its maximum amplitude $\Delta\tau_0$ over a finite time interval T and is confined to a finite region of the fault of radius R_{nuc}. The perturbation is radially symmetric, with radial distance away from the hypocenter (y_2, y_3) along the fault given by $r = \sqrt{(x_2 - y_2)^2 + (x_3 - y_3)^2}$. The perturbation is mathematically smooth in time and space (i.e., the function and all derivatives are continuous). Specifically the perturbation is:

$$\Delta\tau^0(x_2, x_3, t) = \Delta\tau_0 G_1(\sqrt{(x_2 - y_2)^2 + (x_3 - y_3)^2})G_2(t), \tag{27}$$

where,

$$G_1(r) = \begin{cases} \exp\left(\frac{r^2}{r^2 - R_{\text{nuc}}^2}\right), & r < R_{\text{nuc}} \\ 0, & r \geq R_{\text{nuc}} \end{cases} \tag{28}$$

and,

$$G_2(t) = \begin{cases} \exp\left(\frac{(t-T)^2}{t(t-2T)}\right), & 0 < t < T \\ 1, & t \geq T \end{cases} \tag{29}$$

3.2 Simulation parameters

The problem stated above is solved over the time period $0 \leq t \leq t_{\text{f}}$, where t_{f} is a specified final simulation time. All necessary parameter values for this benchmark problem are given in Table 1.

We have chosen model parameters in order to balance the need to resolve relevant physical length scales and computational efficiency. At a rupture speed of 0^+, the quasi-static process zone, Λ_0, is expressed as:

$$\Lambda_0 = C\frac{\mu D_{\text{RS}}}{b\bar{\sigma}_{\text{n}}}, \tag{30}$$

5

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
μ	shear modulus	32.04 GPa
ν	Poisson's ratio	0.25
a_0	rate-and-state direct effect parameter	0.004
a_max	rate-and-state direct effect parameter	0.016
b_0	rate-and-state evolution effect parameter	0.01
$\bar{\sigma}$	effective normal stress	25 MPa
D_RS	characteristic slip distance	0.50/0.53 mm
V_p	plate rate	10^{-9} m/s
V_L	imposed fault slip rate	10^{-9} m/s
V_init	initial slip rate	10^{-9} m/s
V_*	reference slip rate	10^{-6} m/s
f_*	reference friction coefficient	0.6
R_vw	radius of uniform VW region	200 m
W_f	half-width of rate-and-state fault	400 m
l_f	half-length of rate-and-state fault	400 m
Δz	suggested cell size	10 m
t_f	final simulation time	10 years
$\Delta\tau_0$	max amplitude of nucleation stress perturbation	2 MPa
R_nuc	radius of the nucleation stress perturbation	150 m
T	duration of the nucleation stress perturbation	1 s
(y_2, y_3)	hypocenter for nucleation stress perturbation	$(-50$ m, -50 m$)$

where C is a constant on the order of 1. Another important length scale is the nucleation scale. An estimate of the length of the nucleation region h^* in 3D using the aging law is:

$$h^* = \frac{\pi}{2} \frac{\mu' b D_\mathrm{RS}}{(b-a)^2 \bar{\sigma}_\mathrm{n}}, \tag{31}$$

where $\mu' = \mu$ for mode III and $\mu' = \mu/(1-\nu)$ for mode II. With the provided parameter values, the process zone Λ_0 and h^* are uniform over the VW region. The smallest value for the estimated nucleation size given the benchmark parameters is $h^* \approx 262$ m in mode III, and the smallest value for the estimated quasi-static process zone is $\Lambda_0 \approx 88$ m.

We suggest using a cell size of $\Delta z = 10$ m for the simulations; results from simulations using smaller, if feasible, or larger cell sizes are welcome. For methods that use multiple degrees of freedom along cell edges/faces, please take $\Delta z = $ edge length / 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 $\Delta z = \Delta h/N$.

In order to facilitate comparison among modeling groups and the need to balance domain size with reasonable computational cost, we suggest submitting results for the following simulated domain:

$$\mathbf{x} = (x_1, x_2, x_3) \in (-500\mathrm{m}, \, 500\mathrm{m}) \times (-500\mathrm{m}, \, 500\mathrm{m}) \times (-500\mathrm{m}, \, 500\mathrm{m}).$$

Modeling groups may upload results for additional domain sizes as well.

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.
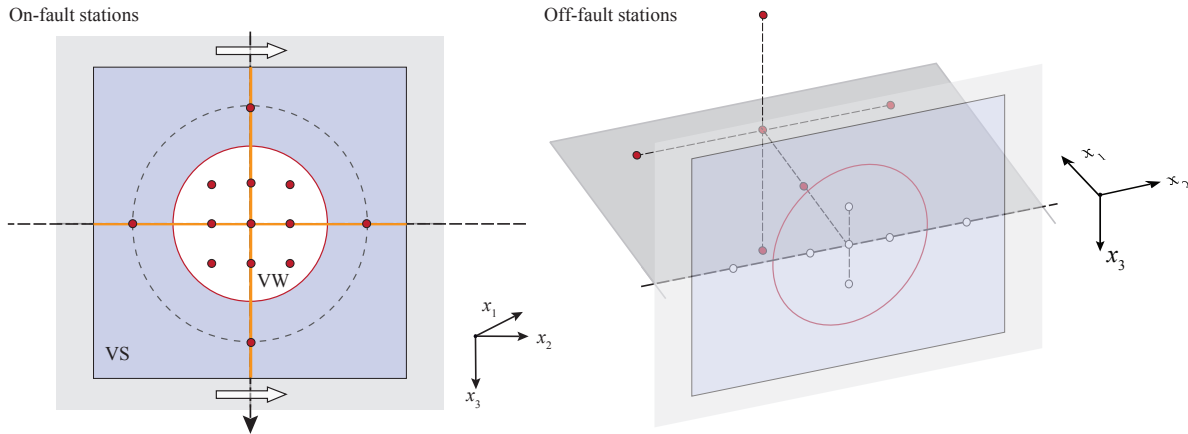

Figure 2: Observation points, lines, and region for model outputs. Local time series is output at (top) on-fault and (bottom) off-fault points (red). Slip and stress evolution are output along two cross-section lines (orange). The region outlined in red is used for estimating source parameter time series 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., ~1 week) during aseismic periods and smaller time intervals (e.g., ~0.05 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 13 observational points on the fault (including 5 at the surface fault trace), as follows:

1. `fltst_strk+000dp+000`: $x_2 = 0$ m, $x_3 = 0$ m;
2. `fltst_strk-100dp+000`: $x_2 = -100$ m, $x_3 = 0$ m;
3. `fltst_strk+000dp+100`: $x_2 = 0$ m, $x_3 = 100$ m;
4. `fltst_strk+100dp+000`: $x_2 = 100$ m, $x_3 = 0$ m;
5. `fltst_strk+000dp-100`: $x_2 = 0$ m, $x_3 = -100$ m;
6. `fltst_strk-100dp-100`: $x_2 = -100$ m, $x_3 = -100$ m;
7. `fltst_strk-100dp+100`: $x_2 = -100$ m, $x_3 = 100$ m;
8. `fltst_strk+100dp-100`: $x_2 = 100$ m, $x_3 = -100$ m;
9. `fltst_strk+100dp+100`: $x_2 = 100$ m, $x_3 = 100$ m;
10. `fltst_strk-300dp+000`: $x_2 = -300$ m, $x_3 = 0$ m;
11. `fltst_strk+000dp+300`: $x_2 = 0$ m, $x_3 = 300$ m;
12. `fltst_strk+300dp+000`: $x_2 = 300$ m, $x_3 = 0$ m;
13. `fltst_strk+000dp-300`: $x_2 = 0$ m, $x_3 = -300$ m.

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	A series of lines, each beginning with a # symbol, that give the following information: • Benchmark problem (e.g. BP7-QD-A) • 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	A single line, which lists the names of the 8 data fields, in column order, separated by spaces. It should be: `t slip_2 slip_3 slip_rate_2 slip_rate_3 shear_stress_2 shear_stress_3 state` (all on one line). The server examines this line to check that your file contains the correct data fields.
Time History	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 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. 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.

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

```
# This is the file header:
# problem=SEAS Benchmark BP7-QD-A
# code=MYcode
# version=1.0
# modeler=A.Modeler
# date=2021/08/01
# element_size=10 m
# location= on fault, 0km along strike, 0km depth
# minimum_time_step=9.6225e-04
# maximum_time_step=5.0636e+04
# num_time_steps=45521
# 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 0.0000000E+00 0.0000000E+00 -2.0000000E+00 -2.0000000E+01
9.622786E-04 9.5383231E-06 2.3908243E-10 -2.0072593E+00 -6.2945862E+00 ...
2.886836E-03 2.6726014E-05 3.1342133E-09 -2.0862801E+00 -5.5839683E+00 ...
4.811393E-03 4.1027878E-05 9.2580681E-09 -2.1704691E+00 -5.4211285E+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, as follows:

1. blkst_strk+000fn+200dp+000: $x_2 = \quad 0$ m, $x_1 = \ 200$ m, $x_3 = \quad 0$ m;
2. blkst_strk+000fn+400dp+000: $x_2 = \quad 0$ m, $x_1 = \ 400$ m, $x_3 = \quad 0$ m;
3. blkst_strk-300fn+400dp+000: $x_2 = -300$ m, $x_1 = \ 400$ m, $x_3 = \quad 0$ m;
4. blkst_strk+000fn+400dp+300: $x_2 = \quad 0$ m, $x_1 = \ 400$ m, $x_3 = \ 300$ m;
5. blkst_strk+300fn+400dp+000: $x_2 = \ 300$ m, $x_1 = \ 400$ m, $x_3 = \quad 0$ m;
6. blkst_strk+000fn+400dp-300: $x_2 = \quad 0$ m, $x_1 = \ 400$ m, $x_3 = -300$ m;

Each time series has 7 data fields, as follows.

Field Name	Description, Units and Sign Convention
t	Time (s)
disp_1	Fault-perpendicular (x_1) component of displacement (m). Positive for extensional motion.
disp_2	Fault-parallel (x_2) component of displacement (m). Positive for right-lateral motion.
disp_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	A series of lines, each beginning with a # symbol, that give the following information: • Benchmark problem (BP7-QD-A) • 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	A single line, which lists the names of the 7 data fields, in column order, separated by spaces. It should be: `t disp_1 disp_2 disp_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.
Time History	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.

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

```
# This is the file header:
# problem=SEAS Benchmark BP7-QD-A
# code=MYcode
# version=1.0
# modeler=A.Modeler
# date=2021/08/01
# element_size=10 m
# location= off fault, x_2 = x_3 = 0 m, x_1 = 400 m
# minimum_time_step=9.6225e-04
# maximum_time_step=5.0636e+04
# num_time_steps=45521
# 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
9.622786E-04 0.000000E+00 0.000000E+00 0.000000E+00 1.000000E-20 1.000000E-9 1.000000E-20
2.886836E-03 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 three global source variables, including the maximum amplitude of slip rates

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

the moment rate

$$\dot{M} = \int_A \mu V dA,$$

for the domain A that surrounds the velocity-weakening patch plus half of the radius of the velocity-weakening patch in every direction, i.e. $\sqrt{x_2^2 + x_3^2} \leq 1.5 R_{\text{vw}}$ (shown in Fig. 2), as well as the moment rate for the velocity-weakening patch

$$\dot{M}_{\text{vw}} = \int_{A_{\text{vw}}} \mu V dA_{\text{vw}},$$

where A_{vw} is the domain $\sqrt{x_2^2 + x_3^2} \leq R_{\text{vw}}$. 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 BP7-QD-A
# code=MYcode
# version=1.0
# modeler=A.Modeler
# date=2021/08/01
# element_size=10 m
# location= 1.5 times VW patch radius
# minimum_time_step=9.6225e-04
# maximum_time_step=5.0636e+04
# num_time_steps=45521
# Column #1 = Time (s)
# Column #2 = Max_slip_rate (log10 m/s)
# Column #3 = Moment_rate (N-m/s)
# Column #4 = Moment_rate of VW patch (N-m/s)
# The line below lists the names of the data fields
```

```
t max_slip_rate moment_rate moment_rate_vw
# Here is the time-series data.
0.000000000E+00 -2.0000000E+00 1.0124054E+13 1.0124054E+13
9.622786759E-04 -1.9829841E+00 1.0381976E+13 1.0381976E+13
2.886836027E-03 -1.9527641E+00 1.0731650E+13 1.0731650E+13
4.811393379E-03 -1.9285083E+00 1.1022296E+13 1.1022296E+13
# ...  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 $R = \left| \Omega_R \right|$ being the total rupture area. We define τ_{avg} as the amplitude of spatially averaged stress vector $\boldsymbol{\tau}$ within the rupture area:

$$\tau_{\mathrm{avg}} = \frac{1}{R} \Big| \int_{\Omega_R} \boldsymbol{\tau} \, d\Omega_R \Big|.$$

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

$$s_{\mathrm{avg}} = \frac{1}{R} \Big| \int_{\Omega_R} \boldsymbol{s} \, d\Omega_R \Big|.$$

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

The data file has 7 data fields, as follows:

Field Name	Description, Units and Sign Convention
event_no	Event number.
t_start	Time event starts (s).
t_end	Time event ends (s).
rup_area	Rupture area (m^2).
avg_stress_start	Spatial average of stress within rupture area at t_start (MPa).
avg_stress_end	Spatial average of stress within rupture area at t_end (MPa).
avg_slip	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: • Benchmark problem (BP7-QD-A) • 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 BP7-QD-A
# author=A.Modeler
# date=2020/08/01
# code=MyCode
# code_version=1.0
# element_size=10 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:  1000m x 1000 m
# 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 9.62278675904542E-04 7.91955350269437E-01 1.4370000E+05 ...  1.4290000E-02
2 2.82816197959969E+07 2.82816211499232E+07 1.3540000E+05 ...  7.9080000E-03
3 5.62723530302155E+07 5.62723543802928E+07 1.3520000E+05 ...  8.0210000E-03
# ...  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 ~ 1 week and ~ 0.05 s for results in aseismic and seismic phases, respectively, or with variable time steps. Either way, data will be interpolated to plot slip every 1 week during the aseismic phase, and every 0.05 s during the coseismic phase.

The data should include nodes with a spacing of ~ 10 m (or larger for simulations with larger cells) along depth ($-400\text{m} \leq x_3 \leq 400\text{m}$) or along strike ($-400\,\text{m} \leq x_2 \leq 400\,\text{m}$). 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 $x_2 = 0\,\text{m}$ and one along-strike profile at $x_3 = 0\,\text{m}$ (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 valerelambert@ucsc.edu).

Each data file has 4 data fields, as follows:

Field Name	Description, Units and Sign Convention
x2 OR x3	Strike (m) at ~ 10 m increments from $-400\,\text{m}$ to $400\,\text{m}$ OR Depth (m) at ~ 10 m increments from $-400\,\text{m}$ to $400\,\text{m}$
t	Time (s). Nonuniform time steps.
max_slip_rate	The $\log_{10}$ of maximum amplitude of slip-rate (taken over the entire fault) ($\log_{10}$ 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	A series of lines, each beginning with a # symbol, that give the following information: • Benchmark problem (e.g. BP7-QD-A) • 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	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:
	`x2 OR x3`
	`t`
	`max_slip_rate`
	`slip_2 OR slip_3 OR stress_2 OR stress_3`

Slip History	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). **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.

Note that x_2 or x_3 should appear in the first row, preceded by two zero numbers, for nodes with a spacing of ~10 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 (~81 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 $(N_t + 1, \sim 83)$, 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 BP7-QD-A
# author=A.Modeler
```

```
# date=2019/12/01
# code=MyCode
# code_version=1.0
# element_size=10 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:  -400m < x2 < 400m, -400m < x3 < 400m
# The line below lists the names of the data fields
x2
t
max_slip_rate
slip_2
# Here are the data
0.0000000E+00  0.0000000E+00 -4.00000E+02 -3.900000E+02 ...  4.000000E+02
0.0000000E+00 -2.0000000E+00 0.000000E+00 0.0000000E+00 ...  0.000000E+00
9.6227867E-04 -1.9829841E+00 9.622786E-13 9.6227873E-13 ...  9.622786E-13
1.1547344E-02 -1.8217367E+00 1.154730E-11 1.1547382E-11 ...  1.154730E-11
2.1170130E-02 -1.6756341E+00 2.117003E-11 2.1170211E-11 ...  2.117003E-11
...
```

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 frictional domain, $(|x_2| \leq l/2 \cap (|x_3| \leq W_f/2$)(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 -400 m to 400 m.
x3	Distance down-dip (m). Positive means a location below the origin. The values range from -400 m to 400 m.
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	A series of lines, each beginning with a # symbol, that give he following information: • Benchmark problem (BP7-QD-A) • 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	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).
Rupture History	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. **Make sure to use double-precision when saving all fields**. **C/C++ users:** We recommend using 21.13E or 21.13e floating-point format. **Fortran users:** We recommend using E22.14 or 1PE22.13 floating-point format. If a node never ruptures, the time should be given as 1.0E+09.

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 BP7-QD-A
# author=A.Modeler
# date=2021/08/01
# code=MyCode
# code_version=1.0
# element_size=10 m
# Column #1 = x2 (m)
# Column #2 = x3 (m)
# Column #3 = time (s)
# Computational domain size:  1000 m x 1000 m
# The line below lists the names of the data fields
x2 x3 t
# Here are the data
-4.000000E+02 4.00000E+02 1.000000000E+09
-3.900000E+02 4.00000E+02 1.000000000E+09
```

```
-3.800000E+02 4.00000E+02 1.000000000E+09
-3.700000E+02 4.00000E+02 1.000000000E+09
-3.600000E+02 4.00000E+02 1.000000000E+09
# ...  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 whole-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.

References

[1] A. C. Eringen and E. S. Suhubi. Elastodynamics (vol. II - lineary theory). Academic Press, 1975.

[2] M. Bonnet. Boundary Integral Equation Methods for Solids and Fluids. Wiley, 1999.