SEAS Benchmark Problems BP8-QD-GS/PW

Taeho Kim, Eric Dunham, Pierre Romanet, and Brittany Erickson

August 12, 2026

Benchmark problems BP8-QD (-GS: Gaussian source; -PW: Peaceman well) are a set of quasi-dynamic three-dimensional (3D) problems in a whole-space with a 2D fault governed by velocity-strengthening rate-and-state friction and the aging law subjected to perturbations in effective normal stress due to fluid injection and along-fault pore fluid diffusion. The fluid injection is modeled with either a Gaussian source or a Peaceman well.

1 3D Problem Setup

The medium is assumed to be a homogeneous, isotropic whole-space defined by:

$$(x_1, x_2, x_3) \in (-\infty, \infty) \times (-\infty, \infty) \times (-\infty, \infty).$$

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 quasi-dynamic problems, inertia is neglected and motion is governed by the equilibrium equation:

$$0 = \nabla \cdot \boldsymbol{\sigma}. \tag{1}$$

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:

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

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

2 Boundary and Interface Conditions

At $x_1 = 0$, the fault defines the interface and we supplement equations (1)-(2) with six interface conditions. We assume a "no-opening condition" on the fault, namely that:

$$u_1(0^+, x_2, x_3, t) = u_1(0^-, x_2, x_3, t), \tag{3}$$

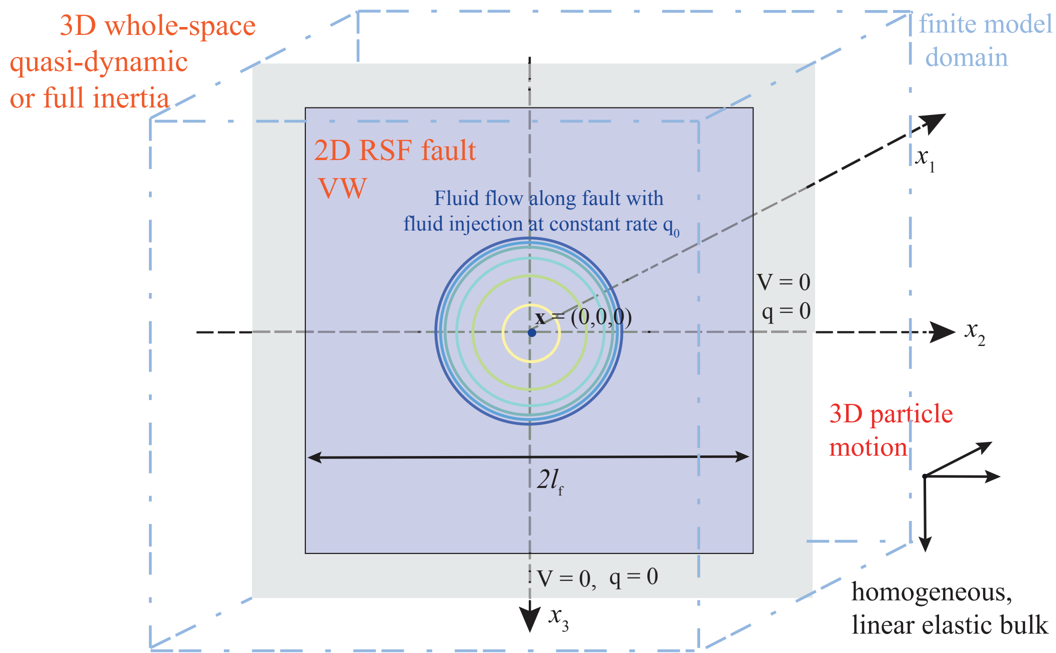

Figure 1: This benchmark considers a 3D antiplane problem for a planar fault embedded in a homogeneous whole-space that responds in a linear elastic manner to slip. The fault ($\{\boldsymbol{x} \mid x_1 = 0, \max(|x_2|, |x_3|) \leq l_\mathrm{f}\}$, blue)is governed by either constant velocity-strengthening rate-and-state friction with the aging-law. Fault slip is induced by pore pressure perturbations due to the injection of fluid in the middle of the fault ($\boldsymbol{x} = (0, 0, 0)$) and the along-fault diffusion of pore fluids.

and define the slip vector:

$$s_j(x_2, x_3, t) = u_j(0^+, x_2, x_3, t) - u_j(0^-, x_2, x_3, t), \quad j = 2, 3, \tag{4}$$

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:

$$V_j = \partial s_j / \partial t, \quad j = 2, 3, \tag{5}$$

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:

$$\sigma_{11}(0^+, x_2, x_3, t) = \sigma_{11}(0^-, x_2, x_3, t), \tag{6a}$$
$$\sigma_{21}(0^+, x_2, x_3, t) = \sigma_{21}(0^-, x_2, x_3, t), \tag{6b}$$
$$\sigma_{31}(0^+, x_2, x_3, t) = \sigma_{31}(0^-, x_2, x_3, t), \tag{6c}$$

The total normal stress on the fault is provided 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 background pore pressure is also spatially uniform. Letting $p(x_2, x_3, t)$ denote the pressure change in response to fluid injection, we have effective normal stress

$$\bar{\sigma}(x_2, x_3, t) = \sigma - p(x_2, x_3, t). \tag{7}$$

The fault shear stress is the sum of the reference shear tractions $\boldsymbol{\tau}^0$, the shear stress change $\Delta\boldsymbol{\tau}$ due to (quasi-static) deformation, and the radiation damping approximation to inertia:

$$\boldsymbol{\tau} = \boldsymbol{\tau}^0 + \Delta\boldsymbol{\tau} - \eta\boldsymbol{V}, \tag{8}$$

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_2 indicate stress that tends to cause right-lateral faulting and positive values of τ_3 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 $(x_2, x_3) \in \Omega_f = (-l_\mathrm{f}, l_\mathrm{f}) \times (-l_\mathrm{f}, l_\mathrm{f})$ we impose rate-and-state friction where shear stress on the fault is equal to fault shear resistance $\boldsymbol{F}$, namely:

$$\boldsymbol{\tau} = \boldsymbol{F}(\boldsymbol{V}, \theta); \tag{9}$$

The fault shear resistance is defined as:

$$\boldsymbol{F} = \bar{\sigma} f(V, \theta)\frac{\mathbf{V}}{V}, \tag{10}$$

where θ is the state variable. The state evolution follows the aging law:

$$\frac{d\theta}{dt} = 1 - \frac{V\theta}{D_\mathrm{RS}}, \tag{11}$$

where D_RS is the characteristic slip distance. The friction coefficient f is given by a regularized formulation:

$$f(V, \theta) = a \sinh^{-1}\left[\frac{V}{2V_*}\exp\left(\frac{f_* + b\ln(V_*\theta/D_\mathrm{RS})}{a}\right)\right], \tag{12}$$

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.

We impose no-slip interface conditions outside of the frictional domain Ω_f (i.e. $|x_2| > l_\mathrm{f}$ or $|x_3| > l_\mathrm{f}$):

$$\boldsymbol{V}(x_2, x_3, t) = 0. \tag{13}$$

The whole-space solution requires conditions at infinity ($x_1 \to \pm\infty$, $x_2 \to \pm\infty$, $x_3 \to \pm\infty$). We require that displacement changes, $\mathbf{u}$, and thus stress changes $\Delta\sigma_{ij}$ vanish at infinity.

2.1 Fluid flow

Fault slip is induced due to perturbations in pore fluid pressure via fluid injection at the center of the fault $(x_2, x_3) = (0, 0)$. Fluid flow is confined to the fault: $x_1 = 0$, $(x_2, x_3) \in \Omega_f$; fault-normal flow is neglected. The continuity of fluid mass is thus:

$$\frac{\partial m}{\partial t} + \nabla.(\rho_f \boldsymbol{q}) = \rho_f q_\mathrm{inj}(t)\delta_D(x_2)\delta_D(x_3) \tag{14a}$$

$$q_\mathrm{inj}(t) = Q_\mathrm{inj}(t)/L_{fwid} \tag{14b}$$

where $m(x_2, x_3, t)$ is the fluid mass per unit volume of rock, ρ_f is the fluid density, $\boldsymbol{q}(x_2, x_3, t)$ is the Darcy velocity along the fault zone (volume flux per unit area of porous solid, having units of m/s). The right-hand-side represents the fluid source term with $Q_{\text{inj}}(t)$ being the total volume injection rate and $q_{\text{inj}}(t)$ being the injection rate per unit thickness of the fault zone where the true fault thickness is defined as L_{fwid}. $\delta_D(x_2), \delta_D(x_3)$ are the Dirac delta functions.

Assuming only elastic changes in rock porosity ϕ and a compressible fluid, then the change in fluid mass is given by:

$$\frac{\partial m}{\partial t} = \rho_f \phi \beta \frac{\partial p_{fault}}{\partial t}, \tag{15}$$

where β is the sum of the fluid and pore compressibilities. In the linearized model used in this benchmark, both ϕ and β in (15) are regarded as constants. The Darcy velocity is given by:

$$q_j = -\frac{k}{\eta}\frac{\partial p}{\partial x_j}, \quad j = 2, 3, \tag{16}$$

where k is the permeability, η is the fluid viscosity, and the effects of gravity are neglected.

Given Equations (14), (15) and (16), and assuming constant permeability, fluid viscosity, porosity, and compressibility, fluid mass conservation can be rewritten to express the evolution of pore fluid pressure along Ω_f as the 2D pressure diffusion equation:

$$\frac{\partial p}{\partial t} = \frac{k}{\phi \beta \eta}\nabla^2 p + \frac{q_{\text{inj}}(t)}{\beta \phi}\delta_D(x_2)\delta_D(x_3). \tag{17}$$

Note that the derivation of Equation 17 from Equations (14), (15) and (16) also requires neglecting the pressure-dependence of ρ_f in transport, while accounting for such a dependence in storage. The hydraulic diffusivity for this diffusion problem is given by $\alpha = k/(\phi\beta\eta)$.

We impose the following boundary condition for fluid flow:

$$\boldsymbol{q}(x_2, x_3, t) \cdot \boldsymbol{n_{\partial\Omega}} = 0, \quad (x_2, x_3) \in \partial\Omega \tag{18}$$

where $\boldsymbol{n_{\partial\Omega}}$ is the normal vector along the boundary, $\partial\Omega$, of the frictional domain Ω_f, indicative of zero fluid flux.

A point source injection in a 2D domain as in (14) results in a singularity of the pressure field at the injection point. Smearing the point source over the volume of the cell that contains the source renders the numerical results dependent on discretization. In order to regularize the singularity and eliminate mesh-dependence, the fluid source is implemented in two different ways: A. through a Gaussian distribution of the fluid source (GS) and B. with a Peaceman well model.

2.1.1 Gaussian injection source (GS)

The injected fluid volume is smeared over the entire computational region through a Gaussian distribution with a characteristic size, L_{gauss}. In other words, (14) is replaced with:

$$\frac{\partial m}{\partial t} + \nabla.(\rho_f \boldsymbol{q}) = \frac{\rho_f q_{\text{inj}}(t)}{2\pi L_{\text{gauss}}^2}\exp\left(-\frac{x_2^2 + x_3^2}{2L_{\text{gauss}}^2}\right) \tag{19}$$

Fluids is injected at a constant injection rate $q_0 = Q_0/L_{fwid}$ for time $0 \leq t < t_{\text{off}}$, after which fluid injection is turned off:

$$q_{\text{inj}}(t) = \begin{cases} q_0, & 0 \leq t < t_{\text{off}} \\ 0, & \text{otherwise.} \end{cases} \tag{20}$$

Note again that Q_0 is the volumetric injection rate, where q_0 is the injection rate per unit thickness of the fault zone. For timescales sufficiently shorter than the diffusion time across l_f (i.e. $t \ll l_f^2/(4\alpha) \approx 220$ hours), the evolution of pore pressure approximately follows the analytic solution:

$$p(x_2, x_3, t) = \begin{cases} \frac{q_o}{4\pi\alpha\beta\phi} \ln\left(\frac{2L_{\text{gauss}}^2 + 4\alpha t}{2L_{\text{gauss}}^2}\right), & (x_2, x_3) = (0, 0) \\ \frac{q_o}{4\pi\alpha\beta\phi}\left[E_1\left(\frac{x_2^2 + x_3^2}{2L_{\text{gauss}}^2 + 4\alpha t}\right) - E_1\left(\frac{x_2^2 + x_3^2}{2L_{\text{gauss}}^2}\right)\right], & \text{otherwise,} \end{cases} \tag{21}$$

where E_1 is the exponential integral function.

2.1.2 Peaceman well model (PW)

In real injection settings, the singularity is regularized in part by the distribution of the injection source over the surface of a well with a finite radius, r_{well}. Typical well radius ranges between 1-4 inches which is substantially smaller than the typical size of a numerical fault cell. The Peaceman well model accounts for the pressure drop between the injection pressure at the well surface to a representative location along the fault cell that includes the well (refered hereafter as the 'well cell'). The model boils down to the following relationships between the injection rate, q_{inj}, well pressure, p_{well}, and the pressure in the well cell, p_e:

$$\begin{aligned} S_{well}\frac{dp_{\text{well}}}{dt} &= -Q_{well\to fault} + Q_{\text{inj}}(t) \\ &= -WI \cdot (p_{\text{well}}(t) - p_e(t)) + Q_{\text{inj}}(t) \end{aligned} \tag{22}$$

$$\begin{aligned} S_e\frac{dp_e}{dt} &= V_e \nabla_h \cdot \boldsymbol{q_e} + Q_{well\to fault} \\ &= V_e \nabla_h \cdot \boldsymbol{q_e} + WI \cdot (p_{\text{well}}(t) - p_e(t)) \end{aligned} \tag{23}$$

where S_{well} and S_e are the volumetric storage of the well and the well cell, V_e is the volume of the well cell, $Q_{well\to fault} \equiv WI \cdot (p_{\text{well}} - p_e)$ is the fluid transfer from the well into the fault, $WI = \frac{2\pi k L_{fwid}}{\eta \ln(r_e/r_{\text{well}})}$ is the 'well index,' and r_e is the 'equivalent radius.' ∇_h is the discretized divergence operator applied to the flux of the well cell $\boldsymbol{q_e}$ for a given discretization scheme. The relationship comes from 1. assuming that flow between the injection source and the fault cells adjacent to the well cell can be approximated by steady-state radial flow, and 2. by enforcing conservation of mass between the injection and the numerical flux out of the well cell. The latter condition makes the determination of r_e discretization-dependent (and consequently, the model output mesh-independent). For example, the equivalent radius for the centered-approximation finite difference scheme (five-point stencil) along square meshes is $r_e = 0.198\Delta x$. For finite volume methods along the same mesh, $r_e = 0.208\sqrt{A}$, where A is the area of the cell. Results for other discretizations may be found in the literature.

Fluids is injected at a constant injection rate q_0 for time $0 \le t < t_{\text{off}}$, after which fluid injection is turned off:

$$q_{\text{inj}}(t) = \begin{cases} q_0, & 0 \le t < t_{\text{off}} \\ 0, & \text{otherwise.} \end{cases} \tag{24}$$

For timescales sufficiently shorter than the diffusion time across l_f (i.e. $t \ll l_f^2/(4\alpha) \approx$ 220 hours), the evolution of pore pressure approximately follows the analytic solution for a point-source injection in 2D:

$$p(x_2, x_3, t) = \frac{q_o}{4\pi\alpha\beta\phi} E_1 \left(\frac{x_2^2 + x_3^2}{4\alpha t} \right) \tag{25}$$

although, discrepancies occur due to the distribution of the fluid source around the finite wellbore radius and numerical errors from the approximation of the equivalent radius, r_e.

Note that we must now define the initial state for the well pressure, $p_{\text{well},init}$. We simply set the initial value to the ambient pore pressure, assuming the injection begins with a full column of water that is at equilibrium with the fluid in the fault.

3 Initial Conditions and Simulation Time

Initial conditions are required for slip, pore pressure, and state. By definition, the initial pore pressure change is zero: $p(x_2, x_3, 0) = 0$. Slip is also initially zero,

$$s_j(x_2, x_3, 0) = 0, \quad j = 2, 3 \tag{26}$$

which together with the requirement that displacements vanish at infinity, implies that displacements everywhere are initially zero:

$$u_j(x_1, x_2, x_3, 0) = 0 \quad j = 1, 2, 3 \tag{27}$$

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:

$$\mathbf{V} = \begin{bmatrix} V_{\text{init}} \\ V_{\text{zero}} \end{bmatrix}, \tag{28}$$

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

$$\boldsymbol{\tau}^0 = \tau_{init} \cdot \mathbf{V}/V. \tag{29}$$

For the given intial slip rate and shear traction, the initial state variable is defined uniquely by Equation 12.

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

Table 1: Parameter values

Parameter	**Definition**	**Value, Units**
μ	shear modulus	32.04 GPa
ν	Poisson's ratio	0.25
ρ	density	2670 kg/m^3
c_s	shear wave speed	3.464 km/s
$\bar{\sigma}_0$	initial effective normal stress	25.0 MPa
τ_{init}	initial shear stress	14.6 MPa
Q_0	fluid injection rate	0.003 m^3/s
S_{well}	volumetric well storage (PW)	1e-7 m^3/Pa
r_{well}	true well radius (PW)	5 cm
L_{gauss}	characteristic distance of gaussian source	50 m
L_{fwid}	fault thickness	1 m
β	pore and fluid compressibility	10^{-8} Pa^{-1}
ϕ	porosity	0.1
k	permeability	5e-14 m^2
η	fluid viscosity	1e-3 Pa·s
α	hydraulic diffusivity	0.05 m^2/s
l_f	half-length of rate-and-state fault	400 m
Δz	cell size	10 m
t_{off}	injection turn-off time	100 hours
t_{f}	Simulation time	30 days
a	rate-and-state direct effect parameter	0.016
b	rate-and-state evolution effect parameter	0.010
D_{RS}	state evolution distance	0.5 mm
V_*	reference slip rate	10^{-6} m/s
f_*	reference friction coefficient	0.6
V_{init}	initial slip rate	10^{-12} m/s

4 Benchmark Output

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

(1) Time series (section 4.1)

(2) Source parameter time series (section 4.2)

(3) Slip, pore pressure, and stress evolution profiles (section 4.3)

All data files are uploaded to the CRESCENT code verification web server (section 5). Information on how to share output (3) is detailed in section 5.

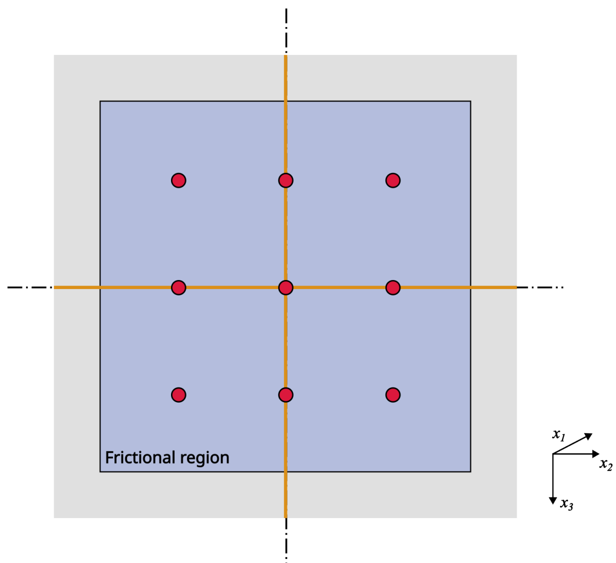

Figure 2: Observation points, lines, and region for model outputs. Local time series is output at select points (red circles). Slip, pore pressure, and stress evolution are output along two cross-section lines (orange lines). The region outlined in black illustrates the frictional region from which the source parameter time series is calculated.

4.1 Time Series Output

You need to upload time series files, which give the components of slip s_2 and s_3, base 10 log of the components of the slip rate V_2 and V_3, base 10 log of the state variable (i.e. $\log_{10}(\theta)$), shear stress components τ_2 and τ_3, pore pressure change p, and components of the Darcy velocity q_2 and q_3 for each station at representative time steps. When outputting modeling results, consider time intervals of 1-hour increments. 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 9 observational points on the fault, as follows:

1. `fltst_strk+000dp+000`: $x_2 = 0$ m, $x_3 = 0$ m;
2. `fltst_strk-200dp+000`: $x_2 = -200$ m, $x_3 = 0$ m;
3. `fltst_strk+000dp+200`: $x_2 = 0$ m, $x_3 = 200$ m;
4. `fltst_strk+200dp+000`: $x_2 = 200$ m, $x_3 = 0$ m;

5. `fltst_strk+000dp-200`: $x_2 = \quad 0$ m, $x_3 = -200$ m;
6. `fltst_strk-200dp-200`: $x_2 = -200$ m, $x_3 = -200$ m;
7. `fltst_strk-200dp+200`: $x_2 = -200$ m, $x_3 = \quad 200$ m;
8. `fltst_strk+200dp-200`: $x_2 = \quad 200$ m, $x_3 = -200$ m;
9. `fltst_strk+200dp+200`: $x_2 = \quad 200$ m, $x_3 = \quad 200$ m;

Each time series has 11 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.
`pore_pressure`	Pore fluid pressure change (MPa).
`darcy_vel_2`	Horizontal component of Darcy velocity (m/s)
`darcy_vel_3`	Vertical component of Darcy velocity (m/s)
`state`	$\log_{10}$ of state variable ($\log_{10}$ s).

The 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. BP8-QD-GS) • 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 11 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 pore_pressure darcy_vel_2 darcy_vel_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 11 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 time-series file, with invented data.

```
# This is the file header:
# problem=SEAS Benchmark BP8-QD-GS
# code=MYcode
# version=1.0
# modeler=A.Modeler
# date=2026/02/15
# element_size=10 m
# location= on fault, z = 0 km
# minimum_time_step=9.622E-04
# maximum_time_step=5.000E+02
# num_time_steps=42356
# Column #1 = Time (s)
# Column #2 = Slip_2 (m)
# Column #3 = Slip_3 (m)
# Column #4 = Slip_rate_2 (m/s)
# Column #5 = Slip_rate_3 (m/s)
# Column #6 = Shear_stress_2 (MPa)
```

```
# Column #7 = Shear_stress_3 (MPa)
# Column #8 = Pore_pressure (MPa)
# Column #9 = Darcy_velocity_2 (m/s)
# Column #10 = Darcy_velocity_3 (m/s)
# Column #11 = 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 pore_pressure darcy_vel_2
darcy_vel_3 state
# Here is the time-series data.
0.00000000E+00 0.0000000E+00 -1.2000000E+01 2.9200000E+01  0.0000000E+00 ...
6.73595073E-03 6.7359534E-15 -1.1999999E+01 2.9200000E+01 -8.4199100E-07 ...
6.06235565E-02 6.0623808E-14 -1.1999995E+01 2.9200000E+01 -7.5776430E-06 ...
4.91724403E-01 4.9174118E-13 -1.1999955E+01 2.9200000E+01 -6.1445444E-05 ...
# ...  and so on.
```

4.2 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

$$V_{\max}(t) = \max_{(x_2,x_3)\in\Omega_\mathrm{f}} V(x_2, x_3, t)$$

and moment rates

$$M_t(t) = \int_{\Omega_\mathrm{f}} \mu\, V(x_2, x_3, t)\; dx_2 dx_3$$

for the frictional domain $(x_2, x_3) \in \Omega_f = (-l_\mathrm{f}, l_\mathrm{f}) \times (-l_\mathrm{f}, l_\mathrm{f})$. 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 BP8-QD-GS
# code=MYcode
# version=1.0
# modeler=A.Modeler
# date=2026/02/15
# element_size=10 m
# location= frictional domain
# minimum_time_step=9.622E-04
# maximum_time_step=5.000E+02
# num_time_steps=42356
# 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.
0.00000000E+00 -1.2000000E+01 1.2000000E+03
6.73595073E-03 -1.2000000E+01 1.2177690E+03
6.06235565E-02 -1.1999997E+01 1.2177690E+03
4.91724403E-01 -1.1999978E+01 1.2177690E+03
# ...  and so on.
```

4.3 Slip, Shear Stress and Pore Pressure Evolution Output

The slip, stress and pore pressure evolution output files with the names

`slip_2_depth.dat`
`slip_2_strike.dat`
`slip_3_depth.dat`
`slip_3_strike.dat`
`shear_stress_2_depth.dat`
`shear_stress_2_strike.dat`
`shear_stress_3_depth.dat`
`shear_stress_3_strike.dat`
`pore_pressure_depth.dat`
`pore_pressure_strike.dat`

are the ASCII files that record the spatial distribution of slip, shear, (both horizontal and vertical components) and pore pressure on a subset of fault nodes at representative time steps throughout the simulation. Data can be saved using representative time intervals 1 hour or with variable time steps. Either way, data will be interpolated to plot slip every 1 hour.

The data should include nodes with a spacing of 10 m (exactly) along depth ($x_2 = 0$, $-400\text{m} \leq x_3 \leq 400\text{m}$) or along strike ($-400\,\text{m} \leq x_2 \leq 400\,\text{m}$, $x_3 = 0$). The files should also contain the time series of maximum slip rate amplitude (taken over the entire fault).

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 `shear_stress_2` OR `shear_stress_3` OR `pore_pressure`	Horizontal OR vertical component of Slip (m) OR horizontal OR vertical component of shear stress (MPa) OR pore pressure (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. BP8-QD-GS) • 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 `shear_stress_2` OR `shear_stress_3` OR `pore_pressure`

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 OR pore pressure 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 columns). 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 ($N_\mathrm{t} + 1$, ∼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 1,000).

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 BP8-QD-GS
# author=A.Modeler
# date=2026/02/15
# 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
...
```

5 Using the CRESCENT DET Benchmark Uploader

The CRESCENT DET benchmark uploader 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://det-uploader.cascadiaquakes.org/`

You will be prompted to request access by filling out your name, email, and affiliation. Once the request is approved, you will receive an email to activate the account and access the data sharing platform.

5.2 Uploading Files

Once logged in, you will see the main homepage for the data sharing platform. To upload files, scroll along the drop-down menu for "Select Benchmark ID" and choose either "BP8-QD-GS" for the Gaussian source or "BP8-QD-PW" for the Peaceman well model. Compress all results (section 4) into a single zip folder with the naming convention of nameOfModeler_version.zip. After reviewing the terms and conditions, accept and upload the zip folder. The platform will appropriately organize the contents of the zip folder based on the names of the files (thus, it is important that the file names are according to the conventions set by section 4). If there are errors in the filenames, no data will be visualized.

5.3 Graphing and Deleting Files

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

Graphing: To graph a file, find the drop-down menu for "View Benchmark Results" at the bottom of the page. Click "Go to Result" which will take you to the data visualization page. Under "Dataset choice," you may check the boxes for the datasets you would like to visualize.

For time-series files (section 4.1 and 4.2), the different datasets will be plotted on the same graph with different colors. A legend to the right of the graph will indicate which colors correspond to which datasets. You may use the mouse scroll to zoom in and out of the plot or click on the plot to draw a specific region to zoom-in. You can always click the home button at the top right to return to the default plotting window.

For the line profiles (section 4.3), contour plots in the space-time domain for the different datasets will be plotted separately. Under the "Graph control" tab, you can change the variable that is plotted under the drop-down menu for "variable selection". At the bottom of all contour plots, snapshots at a specified time will be plotted on the same graph for all datasets. This time can be changed with the cross-section slider at the bottom of the "Graph control" tab. In the "Graph control" tab, you can also change the min and max values of the plotted variable.

Deleting: To delete a dataset that you uploaded, first scroll along the "Select Benchmark ID" drop-down menu under the "File Management" tab. Once you select a particular benchmark problem, you should see all files you have uploaded with the option to click "Delete" for each dataset.

6 Benchmark Tips

Numerical boundary conditions (to truncate the whole-space in x and z 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.

There are multiple options for implementing the Peaceman well model. Three examples include 1. operator-splitting to update well pressure explicitly, 2. Fully implicit time discretization with the linear system increasing in size due to one additional unknown (the well pressure), and 3. formally same as 2, but taking one step in Gaussian elimination to eliminate well pressure as an unknown, which effectively adds the well storage to the specific storage of well cell. The modelers are encouraged to check the numerical solutions against the analytical solution for the point source to avoid significant discrepancies.