TPV18 through TPV21
Branched-Fault Benchmarks with Plasticity

November 24, 2011

These 3D benchmarks use a branching fault with off-fault plastic yielding. We are doing four benchmarks, as shown in the following table.

Benchmarks			
Benchmark	*Dimension*	*Rupture Type*	*Material Properties*
TPV18	3D	Right-lateral, releasing branch.	Linear elastic.
TPV19	3D	Right-lateral, releasing branch.	Drucker-Prager plastic.
TPV20	3D	Left-lateral, restraining branch.	Linear elastic.
TPV21	3D	Left-lateral, restraining branch.	Drucker-Prager plastic.

The material properties are the same as in benchmarks TPV12 and TPV13.

The fault geometry, station locations, and file formats are the same as in benchmarks TPV14 and TPV15.

The nucleation technique is the same as in benchmarks TPV16 and TPV17.

Contents

Part 1: Considerations for Plastic Benchmarks

Benchmarks with plasticity are more complicated to set up than benchmarks with linear elastic material properties. The following list highlights some special considerations to keep in mind when implementing a benchmark with off-fault plastic yielding.

- **Benchmarks TPV19 and TPV21 use non-associative Drucker-Prager plasticity with yielding in shear.** Benchmarks TPV18 and TPV20 use linear elasticity. We include a detailed description of Drucker-Prager plasticity. The material properties are the same as in benchmarks TPV12 and TPV13.

- **There are two test problems you can use to check your implementation of plasticity.** We *strongly* urge you to run the two test problems before you attempt TPV19 and TPV21. Analytic solutions to the test problems are provided. The test problems are the same ones that accompanied benchmark TPV13, so if you did the test problems for TPV13 there is no need for you to repeat them now.

- **The initial stress tensor is specified everywhere throughout the entire model volume.** Benchmarks with linear elasticity only need to specify initial shear and normal stresses on the fault surfaces. In TPV18 through TPV21, the initial shear and normal stresses on the faults are implied by resolving the initial stress tensor onto the fault surfaces.

- **Gravity must be accounted for.** The initial stress increases with depth, so gravitational acceleration is required to balance the gradient in the initial stress tensor.

- **You may need to apply traction forces to the boundary of the model volume.** The initial stresses exert traction forces on the boundary of the model volume. Depending on your implementation, you may need to balance them with externally-applied tractions.

- **Fluid pressure is included in the model.** Fluid pressure is hydrostatic, with water table at the earth's surface. The fluid pressure enters into the plastic constitutive law, and the friction law.

- **Nucleation is done using forced rupture.** Benchmarks with linear elasticity can simply impose higher shear stress near the hypocenter, but that is not possible with plasticity because it would be inconsistent with the initial stress tensor. In TPV18 through TPV21, we are using the new two-stage nucleation method that is being introduced with benchmarks TPV16 and TPV17. It works by having a small zone of forced rupture surrounding the hypocenter, and a larger zone of reduced fracture energy. A detailed description is provided.

We continue to use a linear slip-weakening friction law. As with TPV14 and TPV15, **we use vertical faults that branch along-strike, with a 30-degree branch angle, in a uniform half-space**.

Part 2: Description of the 3D Benchmarks
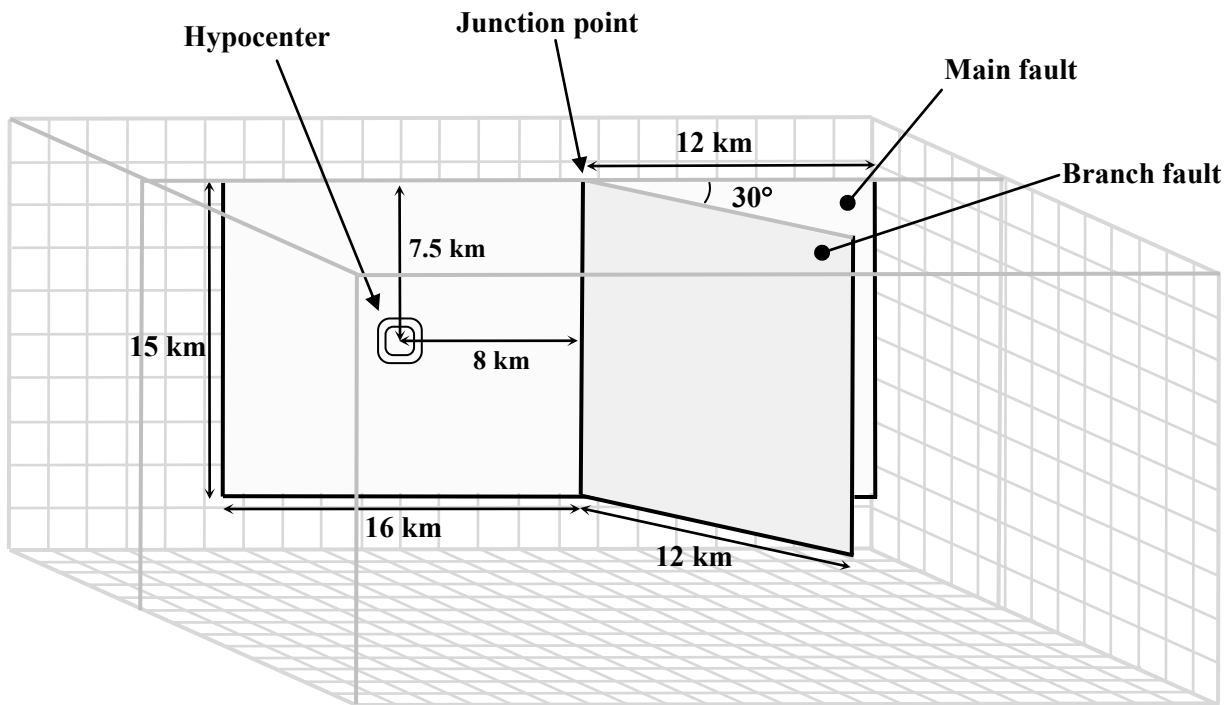
Fault Geometry

The model volume is a half-space.

There are two faults, called the **main fault** and the **branch fault**. The two faults are vertical, planar, strike-slip faults. The faults reach the earth's surface. In TPV18 and TPV19, they are right-lateral faults. In TPV20 and TPV21, they are left-lateral faults.

The main fault is a rectangle measuring 28000 m along-strike and 15000 m deep. The branch fault is a rectangle measuring 12000 m along-strike and 15000 m deep. There is a junction point. It is located 12000 m from the right edge of the main fault, and the main fault passes through it. The branch fault makes an angle of 30 degrees to the main fault, and it ends near, but does not fully reach, the junction point.

The hypocenter is centered in the left side of the main fault. That is, the hypocenter is 8000 m from the junction point, and 7500 m deep.

A node which lies precisely on the border of the main or branch fault is considered to be inside the fault, and so should be permitted to slip. These are the same rules we have used in previous benchmarks.

The main fault runs continuously through the junction point. On the branch fault, the slip vector is required to go to zero at the junction point. Therefore, the branch fault does not connect with the junction point. Instead, the branch fault starts one element away from the junction point. So, a rupture beginning at the hypocenter can propagate freely into the right side of the main fault, but the rupture must jump a short distance to propagate into the branch fault. This is discussed in detail later.

Material Properties

In TPV18 and TPV20, the entire model volume is a linear elastic material, with the following parameters.

Density $\rho = 2700 \text{ kg/m}^3$

Shear-wave velocity $V_s = 3300 \text{ m/s}$

Pressure-wave velocity $V_p = 5716 \text{ m/s}$

In TPV19 and TPV21, the entire model volume is a non-associative Drucker-Prager plastic material that yields in shear, with the following parameters. Drucker-Prager plasticity is described later, in *Part 6*.

Density $\rho = 2700 \text{ kg/m}^3$

Shear-wave velocity $V_s = 3300 \text{ m/s}$

Pressure-wave velocity $V_p = 5716 \text{ m/s}$

Cohesion $c = 5 \text{ MPa}$

Bulk friction $v = 0.85$

Note that these are the same material properties used in benchmarks TPV12 and TPV13.

The material properties are the only difference between the elastic benchmarks (TPV18, TPV20), and the plastic benchmarks (TPV19, TPV21).

Initial Stress Tensor

In an inelastic calculation, the initial stress tensor needs to be specified throughout the medium. The total (time-varying) stress, which equals the initial stress plus the stress change, is used in the plastic yield calculation. The (time-varying) normal and shear stresses acting on the fault surface are implied by the stress tensor in the adjacent rock.

For TPV18 through TPV21, the initial stress tensor varies only with depth. The components of the stress tensor are defined in the following table.

Stress Tensor Components	
Component	*Definition*
σ_{11}	Compressive stress in the vertical direction. Negative values denote compression.
σ_{22}	Compressive stress in the direction parallel to the main fault. Negative values denote compression.
σ_{33}	Compressive stress in the direction perpendicular to the main fault. Negative values denote compression. This equals the negative of the total normal stress on the main fault.
σ_{23}	Shear stress in the horizontal plane. Positive values denote right-lateral shear stress on the main fault. This equals the shear stress on the main fault.
σ_{13}	Shear stress in a vertical plane perpendicular to the main fault.
σ_{12}	Shear stress in a vertical plane parallel to the main fault.

The stress tensor components can vary with time during the simulation.

Fluid pressure P_f is hydrostatic with water table at the surface. It varies with depth, but is constant in time.

For the 3D benchmarks, the fluid pressure and the initial values of the stress tensor are:

$$P_f = (1000 \text{ kg/m}^3)(9.8 \text{ m/s}^2)(\text{depth in meters})$$

$$\sigma_{11} = -(2700 \text{ kg/m}^3)(9.8 \text{ m/s}^2)(\text{depth in meters})$$

$$\sigma_{22} = \begin{cases} b_{22}\left(\sigma_{11} + P_f\right) - P_f, & \text{if depth} \leq 15000 \text{ m} \\ \sigma_{11}, & \text{if depth} > 15000 \text{ m} \end{cases}$$

$$\sigma_{33} = \begin{cases} b_{33}\left(\sigma_{11} + P_f\right) - P_f, & \text{if depth} \leq 15000 \text{ m} \\ \sigma_{11}, & \text{if depth} > 15000 \text{ m} \end{cases}$$

$$\sigma_{23} = \begin{cases} b_{23}\left(\sigma_{11} + P_f\right), & \text{if depth} \leq 15000 \text{ m} \\ 0, & \text{if depth} > 15000 \text{ m} \end{cases}$$

$$\sigma_{13} = 0$$

$$\sigma_{12} = 0$$

The coefficients b_{22}, b_{33}, and b_{23} are given in the following table.

Initial Stress Tensor Coefficients		
Coefficient	*Value for TPV18, TPV19*	*Value for TPV20, TPV21*
b_{22}	0.44327040	0.67738619
b_{33}	0.50911055	0.27499476
b_{23}	−0.15487679	0.09812971

These coefficient values are the only difference between the right-lateral benchmarks (TPV18, TPV19) and the left-lateral benchmarks (TPV20, TPV21).

Note that we have chosen the gravitational acceleration g to be *exactly* 9.8 m/s^2. Do not substitute a "more accurate" value of g.

For all benchmarks, the initial axis of maximum compressive stress is vertical, and its initial value is lithostatic. For the right-lateral benchmarks, the initial axis of minimum compressive stress forms an angle of 39 degrees to the main fault. For the left-lateral benchmarks, the angle is 77 degrees.

Friction Parameters and Nucleation

We use a linear slip-weakening friction law, which has the following five parameters.

Friction Parameters		
Symbol	*Parameter*	*Unit*
μ_s	Static coefficient of friction.	Dimensionless
μ_d	Dynamic coefficient of friction.	Dimensionless
D_c	Slip-weakening critical distance.	Meter
C_0	Frictional cohesion.	Pascal
T	Time of forced rupture.	Second

The operation of the slip-weakening friction law is described in detail later, in *Part 4*.

The friction parameter values are as follows, where r denotes distance to the hypocenter:

$$\mu_s = 0.60$$

$$\mu_d = 0.12$$

$$D_c = \begin{cases} 0.04 \text{ m}, & \text{if } r \leq 360 \text{ m} \\ r/9000, & \text{if } 360 \text{ m} \leq r \leq 3600 \text{ m} \\ 0.40 \text{ m}, & \text{if } r \geq 3600 \text{ m} \end{cases}$$

$$C_0 = \begin{cases} 0.20 \text{ MPa} + (0.0006 \text{ MPa/m})(3000 \text{ m} - \text{depth}), & \text{if depth} \leq 3000 \text{ m} \\ 0.20 \text{ MPa}, & \text{if depth} \geq 3000 \text{ m} \end{cases}$$

$$T = \begin{cases} r/(0.7 \, V_S), & \text{if } r \leq 720 \text{ m} \\ (720 \text{ m})/(0.7 \, V_S) + (r - 720 \text{ m})/(0.35 \, V_S), & \text{if } 720 \text{ m} \leq r \leq 900 \text{ m} \\ 1.0\text{E}+9, & \text{if } r > 900 \text{ m} \end{cases}$$

The variation in the values of D_c and T near the hypocenter causes the rupture to nucleate, in a manner that is described later, in *Part 5*.

Note that the frictional cohesion C_0 is 2.00 MPa at the earth's surface. It is 0.20 MPa at depths greater than 3000 m, and its value is linearly tapered in the uppermost 3000 m.

Running Time, Node Spacing, and Results

Run the model for times from **0.0 to 12.0 seconds after nucleation**.

We recommend using **100 m node spacing** on the fault planes.

The requested output files are:

- **On-fault time-series files**, which give slips, slip rates, and stresses for each on-fault station at each time step. These files are described in *Part 8*.

- **Off-fault time-series files**, which give displacements and velocities for each off-fault station at each time step. These files are described in *Part 9*.

- **Contour-plot files** which, for each node on one of the two faults, gives the time at which the slip rate first changes from 0 to greater than 0.001 m/s. There are two contour-plot files, one for the main fault and one for the branch fault. These files are described in *Part 10*.

Part 3: Junction Point Behavior
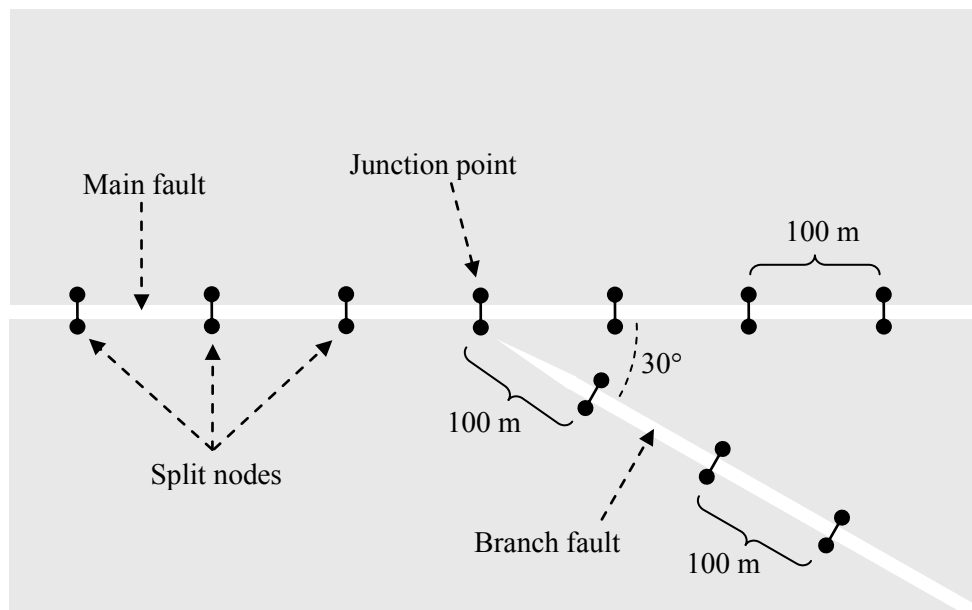
The physical behavior of the junction point is defined as follows: **On the branch fault, the slip vector goes to zero at the junction point.**

The figure illustrates how this behavior can be implemented. It shows the main and branch faults near the junction point. The faults are implemented using split nodes, which in the figure are spaced at intervals of 100 m along the faults. Each split node consists of a pair of nodes on opposite sides of the fault. Fault slip is achieved by allowing the two halves of a split node to move relative to each other.

The main fault runs continuously through the junction point. At the junction point, the main fault has an ordinary split node. So, **a rupture that begins on the left side of the main fault can propagate freely onto the right side of the main fault**.

The branch fault does not meet the junction point. The first split node on the branch fault is located 100 m from the junction point, measured parallel to the strike of the branch fault as shown in the figure. (So, the first split node on the branch fault is located at 50 m perpendicular distance from the main fault.) If the branch fault were extended toward the main fault, it would intersect the main fault at the junction point. The branch fault does not have a split node at the junction point, and so it does not connect to the main fault. As a result, **in order for a rupture on the main fault to propagate onto the branch fault, the rupture must jump a short distance from the main fault to the branch fault**.

9

Part 4: Linear Slip-Weakening Friction

Benchmarks TPV18 through TPV21 use linear slip-weakening friction, with frictional cohesion and forced rupture. This friction law has the following parameters and variables:

Friction Parameters		
Symbol	*Parameter*	*Unit*
μ_s	Static coefficient of friction.	Dimensionless
μ_d	Dynamic coefficient of friction.	Dimensionless
D_c	Slip-weakening critical distance.	Meter
C_0	Frictional cohesion.	Pascal
T	Time of forced rupture.	Second

Friction Variables		
Symbol	*Parameter*	*Unit*
σ_n	Total normal stress acting on the fault, taken to be positive in compression.	Pascal
τ	Shear stress acting on the fault.	Pascal
P_f	Fluid pressure (which is assumed to be constant in time).	Pascal
$\sigma_n - P_f$	Effective normal stress acting on the fault, which is positive in compression.	Pascal

When the fault is sliding, the shear stress τ at a given point on the fault is given by:

$$\tau = C_0 + \mu \times \max(0, \ \sigma_n - P_f)$$

The time-varying coefficient of friction μ is given by:

$$\mu = \begin{cases} \mu_s + (\mu_d - \mu_s) \times D/D_c \,, & \text{if } \ D < D_c \ \text{ and } \ t < T \\ \\ \mu_d \,, & \text{if } \ D \geq D_c \ \text{ or } \ t \geq T \end{cases}$$

where D is the total distance the node has slipped and t is the time since the start of the earthquake.

The distance D that the node has slipped is path-integrated. For example, if the node slips 0.4 m in one direction and then 0.1 m in the opposite direction, the value of D is 0.5 m (and not 0.3 m).

The time of forced rupture T is used to nucleate the rupture, as described in the next section. At time $t = T$, the node is forced to rupture by setting the coefficient of friction μ equal to the dynamic coefficient of friction μ_d. If a given node does not undergo forced rupture, then the value of T is 1.0E9.

Tension on the fault: If you encounter tension on the fault, you should **treat tension on the fault the same as if the effective normal stress equals zero**. This is shown in the above formulas by the expression $\max(0, \sigma_n - P_f)$.

You should **constrain the motion of the node so that the fault cannot open (that is, only permit sliding parallel to the fault), even when the fault is in tension**. During the time the fault is in tension, continue to accumulate the slip distance D as usual.

In your time-series files, where you are asked to report "normal stress," you should report the effective normal stress which is defined to be $\sigma_n - P_f$.

Part 5: Nucleation

The nucleation method is built in to the friction coefficients, so you don't need to do anything special to nucleate the rupture.

For TPV18-21, we are using a new two-stage nucleation method. Nucleation is achieved by adjusting the friction parameters T and D_c in the vicinity of the hypocenter.

The first stage is a circular zone of forced rupture which surrounds the hypocenter. Its radius is 0.9 km. At the hypocenter, $T = 0$. The value of T then increases with distance from the hypocenter, which creates an expanding circular region of forced rupture. The forced rupture expands at a speed of $0.7 * V_S$ for 80% of the way, and then $0.35 * V_S$ for the remaining 20% of the way to the edge of the zone. Outside the zone of forced rupture, T is equal to 1.0E9, which means that forced rupture does not occur outside the zone.

The second stage is a circular zone of reduced D_c which surrounds the hypocenter. Its radius is 3.6 km. In the innermost 10% of the zone, D_c equals 0.04 m. The value of D_c then increases linearly with distance from the hypocenter, and reaches its final value of 0.4 m at the edge of the zone. Outside the zone, D_c equals 0.4 m. The effect is to create a circular region of reduced fracture energy surrounding the hypocenter, which helps the rupture to expand during the early part of the simulation.

Part 6: Drucker-Prager Plasticity

Constitutive Law

For TPV19 and TPV21 we are using non-associative Drucker-Prager plasticity with yielding in shear. **This is exactly the same plasticity, with the same constitutive parameters, used in benchmark TPV13.** The material is characterized by six constitutive parameters:

Density $\rho = 2700 \text{ kg/m}^3$

Shear-wave velocity $V_s = 3300 \text{ m/s}$

Pressure-wave velocity $V_p = 5716 \text{ m/s}$

Cohesion $c = 5 \text{ MPa}$

Bulk friction $v = 0.85$

Fluid pressure $P_f = (1000 \text{ kg/m}^3)(9.8 \text{ m/s}^2)(\text{depth in meters})$

Except for the fluid pressure, all the constitutive parameters are uniform. The "cohesion" listed here is not to be confused with the "frictional cohesion" that appears in the fault friction law. From these six constitutive parameters, we can derive several additional ones:

Shear modulus $\mu = \rho V_s^2$

Lame volumetric parameter $\lambda = \rho(V_p^2 - 2V_s^2)$

Bulk modulus $K = \lambda + \frac{2}{3}\mu$

Angle of friction $\phi = \tan^{-1} v$

Let σ_{ij} denote the stress tensor. Define:

Mean stress $\sigma_m = \frac{1}{3}(\sigma_{11} + \sigma_{22} + \sigma_{33})$

Stress deviator $s_{ij} = \sigma_{ij} - \sigma_m \delta_{ij}$

Second invariant of the stress deviator $J_2(\sigma) = \frac{1}{2}\Sigma_{i,j} s_{ij}s_{ji}$

Drucker-Prager yield stress $Y(\sigma) = \max(0, \ c\cos\phi - (\sigma_m + P_f)\sin\phi)$

Drucker-Prager yield function $F(\sigma) = \sqrt{J_2(\sigma)} - Y(\sigma)$

The Drucker-Prager material is required to satisfy the *yield equation*: $F(\sigma) \le 0$

When $F(\sigma) < 0$, the material behaves like a linear elastic material, with Lame parameters λ and μ.

When $F(\sigma) = 0$, if the material is subjected to a strain that tends to cause an increase in $F(\sigma)$, then the material *yields*. For TPV19 and TPV21, we assume that the material yields in shear. *Yielding in shear* means that when the material yields, the stress tensor σ_{ij} changes by an amount proportional to the stress deviator s_{ij}, so as to preserve the condition $F(\sigma) = 0$ with no change in mean stress σ_m. This is explained further in the section on the return map algorithm, below.

Note for the mathematically inclined: Yielding in shear corresponds to using a plastic flow potential function $Q(\sigma) = \sqrt{J_2(\sigma)}$.

Return Map Algorithm

The return map algorithm is a popular algorithm for implementing plasticity, so we describe it here.

Given:

σ_{ij}^0 = Stress tensor at the start of a time step.

ϵ_{ij} = Strain tensor increment applied during the time step.

We want to find:

σ_{ij} = Stress tensor at the end of the time step.

Step 1. Calculate a *trial stress tensor* by assuming that the material behaves elastically:

$$\sigma_{ij}^{\text{trial}} = \sigma_{ij}^0 + \lambda(\epsilon_{11} + \epsilon_{22} + \epsilon_{33})\delta_{ij} + 2\mu\epsilon_{ij}$$

Step 2. Calculate the following:

Trial mean stress $\sigma_m^{\text{trial}} = \frac{1}{3}\left(\sigma_{11}^{\text{trial}} + \sigma_{22}^{\text{trial}} + \sigma_{33}^{\text{trial}}\right)$

Trial stress deviator $s_{ij}^{\text{trial}} = \sigma_{ij}^{\text{trial}} - \sigma_m^{\text{trial}}\delta_{ij}$

Second invariant of the trial stress deviator $J_2(\sigma^{\text{trial}}) = \frac{1}{2}\sum_{i,j} s_{ij}^{\text{trial}} s_{ji}^{\text{trial}}$

Trial Drucker-Prager yield stress $Y(\sigma^{\text{trial}}) = \max\left(0, \ c\cos\phi - \left(\sigma_m^{\text{trial}} + P_f\right)\sin\phi\right)$

Trial Drucker-Prager yield function $F(\sigma^{\text{trial}}) = \sqrt{J_2(\sigma^{\text{trial}})} - Y(\sigma^{\text{trial}})$

Step 3. If $F(\sigma^{\text{trial}}) \leq 0$, then this is an *elastic step*. In this case, we take:

$$\sigma_{ij} = \sigma_{ij}^{\text{trial}}$$

Step 4. If $F(\sigma^{\text{trial}}) > 0$, then this is a *plastic step*. In this case, we take:

$$r = Y(\sigma^{\text{trial}}) / \sqrt{J_2(\sigma^{\text{trial}})}$$

$$\sigma_{ij} = \sigma_m^{\text{trial}} \delta_{ij} + r s_{ij}^{\text{trial}}$$

It is easy to check that in this last case, we obtain $F(\sigma) = 0$ and also $\sigma_m = \sigma_m^{\text{trial}}$.

Sample Fortran Code

Courtesy of Joe Andrews, here is sample Fortran code for the return map algorithm.

```
c   ---[ Constitutive parameters ]---
c       Density
        density = 2700.0
c       Vs
        vs = 3300.0
c       Vp
        vp = 5716.0
c       Shear modulus
        mu = density*(vs**2)
c       Lame volumetric parameter
        lambda = density*(vp**2 - 2.0*(vs**2))
c       Cohesion is 5 MPa
        cohes = 5.0e6
c       Bulk friction
        blkfric = 0.85
c       Angle of friction
        angfric = atan(blkfric)
c       Fluid pressre
        fluidpresh = 0.0

c   ---[ Calculate trial stress tensor ]---
c       Stress tensor components: sxx,syy,szz,sxy,syz,sxz
c       Strain tensor increment components: exx,eyy,ezz,exy,eyz,exz
        etrace = exx + eyy + ezz
        sxx = sxx + lambda*etrace + 2.0*mu*exx
        syy = syy + lambda*etrace + 2.0*mu*eyy
        szz = szz + lambda*etrace + 2.0*mu*ezz
        sxy = sxy + 2.0*mu*exy
        syz = syz + 2.0*mu*eyz
        sxz = sxz + 2.0*mu*exz
```

```
c  ---[ Calculate trial yield stress ]---
c     Mean stress
      sm = (sxx + syy + szz)/3.0
c     Find stress deviator components
      sdxx = sxx - sm
      sdyy = syy - sm
      sdzz = szz - sm
      sdxy = sxy
      sdyz = syz
      sdxz = sxz
c     Second invariant of stress deviator
      secinv = 0.5*(sdxx**2+sdyy**2+sdzz**2)+sdxy**2+sdyz**2+sdxz**2
c     Scalar measure of shear stress
      tau = sqrt(secinv)
c     Yield stress
      taulim = cohes*cos(angfric) - (sm + fluidpresh)*sin(angfric)
      taulim = max(0.0, taulim)

c  ---[ Yielding in shear ]---
c     Stress deviators are adjusted, no change to mean stress
      if (tau .gt. taulim) then
        yldfac = taulim/tau
        sxx = sdxx*yldfac + sm
        syy = sdyy*yldfac + sm
        szz = sdzz*yldfac + sm
        sxy = sdxy*yldfac
        syz = sdyz*yldfac
        sxz = sdxz*yldfac
      end if
```

Test Problem #1 (S-Wave Single-Element Test)

We are providing two test problems, that you can use to test your implementation of non-associative Drucker-Prager plasticity. The first problem tests the effect of an S-wave on a single element, and the second problem tests the effect of a P-wave on a single element. Both problems have analytic solutions.

This test is performed on a single element.

- For codes that use cubical elements, the single element is a cube oriented with edges parallel to the Cartesian axes. One face of the cube is at $x = 0$ and the opposite face of the cube is at $x = 1$ m.

- For codes that use tetrahedral elements, the single element is a tetrahedron. One face of the tetrahedron is at $x = 0$, and the remaining node is at $x = 1$ m.

Velocity and displacement of all nodes are prescribed as boundary conditions.

- For nodes located at $x = 0$, all components of displacement and velocity remain at zero.

- For nodes located at $x = 1$ m, displacement is initially zero. The y component of velocity is constant at 1 m/s. All other components of velocity are zero.

All six components of the stress tensor in the element are initially zero.

Calculate all six components of the stress tensor as a function of time.

We recommend using a time step of $5.0e{-}6$ seconds, and calculating for times t from 0 to $5.0e{-}4$ seconds.

Use the material properties for TPV19:

Density $\rho = 2700$ kg/m^3

Shear-wave velocity $V_s = 3300$ m/s

Pressure-wave velocity $V_p = 5716$ m/s

Cohesion $c = 5$ MPa

Bulk friction $v = 0.85$

Fluid pressure $P_f = 0$

Analytic Solution

For $t \leq 1.2956829537e{-}4$ seconds, the element behaves elastically, and the stress tensor components are:

$$\sigma_{xy} = 2.9403000000e{+}10 * t$$
$$\sigma_{xx} = \sigma_{xz} = \sigma_{yy} = \sigma_{yz} = \sigma_{zz} = 0$$

For $t > 1.2956829537e{-}4$ seconds, the element behaves plastically, and the stress components are:

$$\sigma_{xy} = 3.8096965888e{+}6$$
$$\sigma_{xx} = \sigma_{xz} = \sigma_{yy} = \sigma_{yz} = \sigma_{zz} = 0$$

Test Problem #2 (P-Wave Single-Element Test)

This problem tests the effect of a P-wave on a single element. It is exactly the same as the first problem, above, except for the velocity that is applied to nodes at $x = 1$ m:

- For nodes located at $x = 1$ m, displacement is initially zero. The x component of velocity is constant at -1 m/s. All other components of velocity are zero.

We recommend using a time step of $5.0e{-}5$ seconds, and calculating for times t from 0 to $5.0e{-}3$ seconds.

Analytic Solution

For $t \leq 1.7246213245e{-}3$ seconds, the element behaves elastically, and the stress tensor components are:

$$\sigma_{xx} = -8.8216171200e{+}10 * t$$
$$\sigma_{yy} = \sigma_{zz} = -2.9410171200e{+}10 * t$$
$$\sigma_{xz} = \sigma_{yz} = \sigma_{zz} = 0$$

For $t > 1.7246213245e{-}3$ seconds, the element behaves plastically, and the stress components are:

$$\sigma_{xx} = -8.5665432299e{+}10 * t - 4.3990587021e{+}6$$
$$\sigma_{yy} = \sigma_{zz} = -3.0685540651e{+}10 * t + 2.1995293511e{+}6$$
$$\sigma_{xz} = \sigma_{yz} = \sigma_{zz} = 0$$

Part 7: Methods for Implementing Gravity and Boundary Tractions

In an inelastic calculation, the initial stress needs to be specified throughout the medium. The total stress, which equals initial stress plus stress change, is used in the yield calculation. This creates two challenges.

- Because the initial stress varies with depth, a gravitational force is required to balance the gradient of the initial stress tensor.

- The initial stress tensor exerts traction forces on the boundary of the mesh. If the code permits the mesh boundaries to move, then external tractions must be applied to balance these forces.

There are two basic methods for dealing with gravity and boundary tractions, described below. You may choose either method.

Method 1 — Using Stress Change

The stress tensor that is calculated and stored in the code is the change of stress from some initial state. The initial state is assumed to be in static equilibrium. With this method, the gravity and boundary tractions do not appear explicitly in the calculation, because as far as the code is concerned, all the initial stresses and forces are equal to zero.

In an elastic calculation, the initial state does not need to be known. In an inelastic calculation, the initial stress tensor must be specified and stored within each element (it varies only with depth in this case). Performing the plastic yield calculation then requires three steps:

1. Add the stress change to the initial stress, obtaining the total stress tensor.
2. Perform the plastic yield calculation (for example, with the return map algorithm), obtaining a new total stress tensor.
3. Subtract the initial stress from the new total stress tensor, obtaining a new stress change.

Because the code starts off with all stresses and forces equal to zero, you must "manually" specify the initial shear stress and initial effective normal stress at each node on the fault surface. To calculate the frictional sliding on the fault, you must (a) calculate the tractions on the fault surface implied by the change in stress, and then (b) add the initial shear stress and initial effective normal stress, to obtain the total shear stress and total effective normal stress which appear in the friction law.

Method 2 — Using Explicit Gravity and Boundary Tractions

The stress tensor that is calculated and stored in the code is the total stress tensor. The stress tensor throughout the medium must be initialized to a state that is in static equilibrium. The equilibrium stress state might come from a static calculation. In this case, because it depends only on depth, the initial stress state is simple enough to specify directly.

The acceleration of gravity must be added explicitly to the equations of motion. The gradient of the initial stress tensor is balanced by gravitational forces. For TPV18 through TPV21, you should set the gravitational acceleration g to be exactly 9.8 m/s².

If your code allows the mesh boundaries to move, then you must also apply external tractions to the mesh boundary, to balance the tractions caused by the initial stress. You need to calculate and store the force exerted on each boundary node by the initial stress conditions (including the gravitational force). Then, during the dynamic calculation, apply the negative of the stored force to each node lying in the boundary, thereby eliminating unwanted motion of the boundary. There are two ways to calculate the forces:

1. Perform a preliminary static calculation.

2. Analytically calculate the forces, in a manner that depends on the details of your code.

Because of approximation errors in the equations of motion, there may be small velocities throughout the mesh before real signals arrive. They are caused by small residual forces that exist because the nodal forces calculated from the initial stress tensor may not precisely cancel out the calculated gravitational forces. This effect is generally small enough to be ignored.

(Remark: If you do a preliminary static calculation to obtain boundary tractions, you can at the same time calculate the residual forces. Then, during the dynamic calculation, you can cancel out both the boundary tractions and the residual forces, thereby eliminating the small velocities. Just be careful that you don't also cancel out the traction forces acting on the fault surface!)

The plastic yield calculation has the total stress tensor available, so that the initial stress tensor does not need to be stored.

The code automatically calculates the total shear stress and total normal stress acting on the fault surface. So, it is not necessary to "manually" specify initial stresses acting on the fault surface.

However, the frictional sliding calculation also requires the fluid pressure, so that effective normal stress can be computed. There are two ways you could obtain the fluid pressure at each node on the fault surface. You may use either technique.

1. Calculate the fluid pressure by applying the formula

$$P_f = (1000 \text{ kg/m}^3)(9.8 \text{ m/s}^2)(\text{depth in meters})$$

    This should work well for fault nodes not located on the earth's surface. However, special considerations apply to a fault node located at the earth's surface. For most codes it is incorrect to assign a fluid pressure of zero. We recommend that the fluid pressure be set as follows: (a) If your code has an initial stress tensor that is uniform within each element, then a fault node at the earth's surface should be assigned a fluid pressure corresponding to a depth of 1/2 element. (b) If

your code has an initial stress tensor that varies linearly with depth inside each element, then a fault node at the earth's surface should be assigned a fluid pressure corresponding to a depth of 1/3 element.

2. Perform a preliminary static calculation to obtain the traction exerted on the fault surface by fluid pressure. Set up the static calculation to use density and initial stress as

$$\rho = 1000 \text{ kg/m}^3$$
$$\sigma_{11} = \sigma_{22} = \sigma_{33} = -(1000 \text{ kg/m}^3)(9.8 \text{ m/s}^2)(\text{depth in meters})$$

The static calculation should also include the gravitational force. You can numerically calculate the normal stress exerted at each node on the fault surface by fluid pressure, and store the calculated normal stress. Then, during the dynamic calculation, you use the stored value as the fluid pressure.

(Because of numerical approximation errors, the static calculation may also yield a small shear stress acting on the fault surface. If you wish, you may store the calculated shear stress, and deduct it from the total shear stress (using vector subtraction) during the dynamic calculation.)

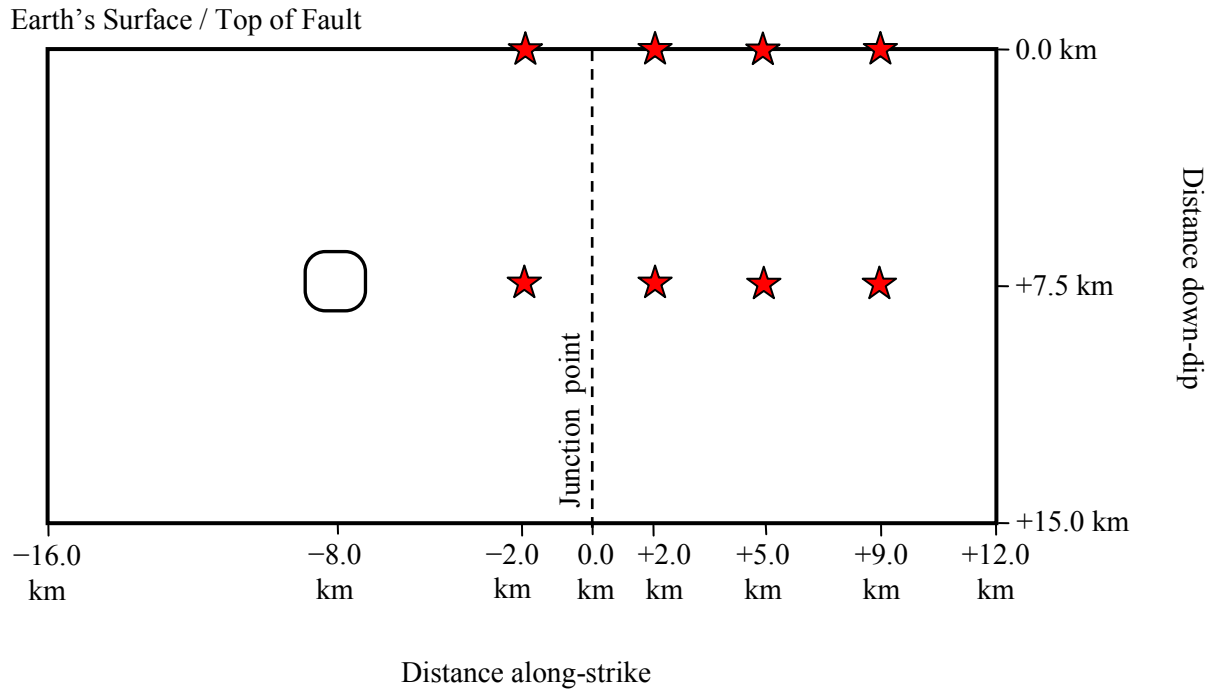## Part 8: On-Fault Stations, and Time-Series File Format

The 3D benchmarks use 8 stations on the main fault and 6 stations on the branch fault, which are listed below. Diagrams of station locations are given following the table. You need to supply one time-series file for each station. **All station locations and file formats are exactly the same as for benchmarks TPV14 and TPV15.**

On-Fault Stations, for TPV18 through TPV21	
Station Name	**Location**
faultst-020dp000	On main fault, -2.0 km along strike, 0 km down-dip.
faultst020dp000	On main fault, 2.0 km along strike, 0 km down-dip.
faultst050dp000	On main fault, 5.0 km along strike, 0 km down-dip.
faultst090dp000	On main fault, 9.0 km along strike, 0 km down-dip.
faultst-020dp075	On main fault, -2.0 km along strike, 7.5 km down-dip.
faultst020dp075	On main fault, 2.0 km along strike, 7.5 km down-dip.
faultst050dp075	On main fault, 5.0 km along strike, 7.5 km down-dip.
faultst090dp075	On main fault, 9.0 km along strike, 7.5 km down-dip.
branchst020dp000	On branch fault, 2.0 km along strike, 0 km down-dip.
branchst050dp000	On branch fault, 5.0 km along strike, 0 km down-dip.
branchst090dp000	On branch fault, 9.0 km along strike, 0 km down-dip.
branchst020dp075	On branch fault, 2.0 km along strike, 7.5 km down-dip.
branchst050dp075	On branch fault, 5.0 km along strike, 7.5 km down-dip.
branchst090dp075	On branch fault, 9.0 km along strike, 7.5 km down-dip.

If you do not have a node at the location of a station, there are two options: (1) you can move the station to the nearest node, or (2) you can interpolate the data values from nodes near the station location.

Note: Location along-strike is measured relative to the junction point. Positive locations are to the right of the junction point. On the branch fault, distance is measured from the junction point parallel to the strike of the branch fault (so the first split node on the branch fault has an along-strike location of 100 m, assuming 100 m node spacing).

Station Locations on the Main Fault

Earth's Surface / Top of Fault

Junction point

Distance down-dip

0.0 km

+7.5 km

+15.0 km

−16.0 km   −8.0 km   −2.0 km   0.0 km   +2.0 km   +5.0 km   +9.0 km   +12.0 km

Distance along-strike

There are 4 stations at the earth's surface:

- −2.0 km, +2.0 km, +5.0 km, +9.0 km along-strike, and 0 km down-dip distance.
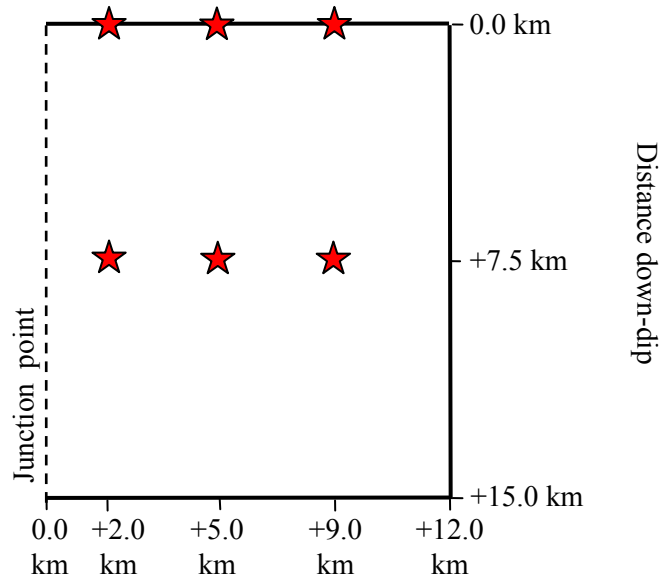
There are 4 deeper stations:

- −2.0 km, +2.0 km, +5.0 km, +9.0 km along-strike, and 7.5 km down-dip distance.

Note that location along-strike is measured relative to the junction point. Positive locations are to the right of the junction point.

Station Locations on the Branch Fault

Earth's Surface / Top of Fault

There are 3 stations at the earth's surface:

- +2.0 km, +5.0 km, +9.0 km along-strike, and 0 km down-dip distance.

There are 3 deeper stations:

- +2.0 km, +5.0 km, +9.0 km along-strike, and 7.5 km down-dip distance.

Note that location along-strike is measured relative to the junction point. Positive locations are to the right of the junction point.

Each time series file is an ASCII file that contains 8 data fields, as follows.

On-Fault Time Series Data Fields for TPV18 through TPV21	
Field Name	**Description, Units, and Sign Convention**
t	Time (s).
h-slip	Horizontal slip along-strike (m). Sign convention: Positive means **right lateral** slip.
h-slip-rate	Horizontal slip rate along-strike (m/s). Sign convention: Positive means **right lateral** motion.
h-shear-stress	Horizontal shear stress along-strike (MPa). Sign convention: Positive means shear stress that tends to cause **right-lateral** slip.
v-slip	Vertical along-dip slip (m). Sign convention: Positive means **downward slip** (that is, the far side of the fault moving downward relative to the near side of the fault).
v-slip-rate	Vertical along-dip slip rate (m/s). Sign convention: Positive means **downward motion** (that is, the far side of the fault moving downward relative to the near side of the fault).
v-shear-stress	Vertical along-dip shear stress (MPa). Sign convention: Positive means shear stress that tends to cause **downward slip** (that is, the far side of the fault moving downward relative to the near side of the fault).
n-stress	Normal stress (MPa). Note: You should report **effective normal stress**, which is defined to be total normal stress minus fluid pressure. Sign convention: Positive means **extension**.

The **near side** of a fault is in the front of the diagram.
The **far side** of a fault is in the back of the diagram.

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

On-Fault Time Series File Format for TPV18 through TPV21	
File Section	**Description**
File Header	A series of lines, each beginning with a # symbol, that gives the following information: • Benchmark problem (TPV18 or TPV19) • Author • Date • Code • Code version (if desired) • Node spacing or element size • Time step • Number of time steps in file • Station location • Descriptions of data columns (7 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  h-slip  h-slip-rate  h-shear-stress  v-slip  v-slip-rate v-shear-stress  n-stress` (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. **C/C++ users:** For all data fields except the time, we recommend using 14.6E or 14.6e floating-point format. For the time field, we recommend using 20.12E or 20.12e format (but see the note on the next page). **Fortran users:** For all data fields except the time, we recommend using E15.7 floating-point format. For the time field, we recommend using E21.13 format (but see the note on the next page). 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.

Note: We recommend higher precision for the time field so the server can tell that your time steps are all equal. (If the server thinks your time steps are not all equal, it will refuse to apply digital filters to your data.) If you use a "simple" time step value like 0.01 seconds or 0.005 seconds, then there is no need for higher precision, and you can write the time using the same precision as all the other data fields. When you upload a file, the server will warn you if it thinks your time steps are not all equal.

Here is an example of an on-fault time-series file. This is an invented file, not real modeling data.

```
# Example on-fault time-series file.
#
# This is the file header:
# problem=TPV18
# author=A.Modeler
# date=2011/01/23
# code=MyCode
# code_version=3.7
# element_size=100 m
# time_step=0.005
# num_time_steps=1600
# location= on main fault, 9 km along strike, 7.5km down-dip
# Column #1 = Time (s)
# Column #2 = horizontal slip (m)
# Column #3 = horizontal slip rate (m/s)
# Column #4 = horizontal shear stress (MPa)
# Column #5 = vertical slip (m)
# Column #6 = vertical slip rate (m/s)
# Column #7 = vertical shear stress (MPa)
# Column #8 = normal stress (MPa)
#
# The line below lists the names of the data fields:
t h-slip h-slip-rate h-shear-stress v-slip v-slip-rate v-shear-stress n-stress
#
# Here is the time-series data.
# There should be 8 numbers on each line, but this page is not wide enough
#  to show 8 numbers on a line, so we only show the first five.
0.000000E+00   0.000000E+00   0.000000E+00   7.000000E+01   0.000000E+00  ...
5.000000E-03   0.000000E+00   0.000000E+00   7.104040E+01   0.000000E+00  ...
1.000000E-02   0.000000E+00   0.000000E+00   7.239080E+01   0.000000E+00  ...
1.500000E-02   0.000000E+00   0.000000E+00   7.349000E+01   0.000000E+00  ...
2.000000E-02   0.000000E+00   0.000000E+00   7.440870E+01   0.000000E+00  ...
2.500000E-02   0.000000E+00   0.000000E+00   7.598240E+01   0.000000E+00  ...
# ... and so on.
```

Part 9: Off-Fault Stations, and Time-Series File Format

The 3D benchmarks use the 11 off-fault stations listed below. All stations are at the earth's surface.

Refer to the next page for a diagram of station locations. You need to supply one time-series file for each station.

All station locations and file formats are exactly the same as for benchmarks TPV14 and TPV15.

Off-Fault Stations for TPV18 through TPV21	
Station Name	**Location**
`body030st-020dp000`	3.0 km off main fault (far side), -2.0 km along strike, 0 km depth.
`body-030st-020dp000`	-3.0 km off main fault (near side), -2.0 km along strike, 0 km depth.
`body030st020dp000`	3.0 km off main fault (far side), 2.0 km along strike, 0 km depth.
`body-006st020dp000`	-0.6 km off main fault (near side), 2.0 km along strike, 0 km depth.
`body-042st020dp000`	-4.2 km off main fault (near side), 2.0 km along strike, 0 km depth.
`body030st050dp000`	3.0 km off main fault (far side), 5.0 km along strike, 0 km depth.
`body-014st050dp000`	-1.4 km off main fault (near side), 5.0 km along strike, 0 km depth.
`body-059st050dp000`	-5.9 km off main fault (near side), 5.0 km along strike, 0 km depth.
`body030st080dp000`	3.0 km off main fault (far side), 8.0 km along strike, 0 km depth.
`body-023st080dp000`	-2.3 km off main fault (near side), 8.0 km along strike, 0 km depth.
`body-076st080dp000`	-7.6 km off main fault (near side), 8.0 km along strike, 0 km depth.

In the station names, the first number is the horizontal perpendicular distance from the station to the main fault. A positive number means that the station is located on the **far side** of the main fault.

If you do not have a node at the location of a station, there are two options: (1) you can move the station to the nearest node, or (2) you can interpolate the data values from nodes near the station location.

Off-Fault Station Locations
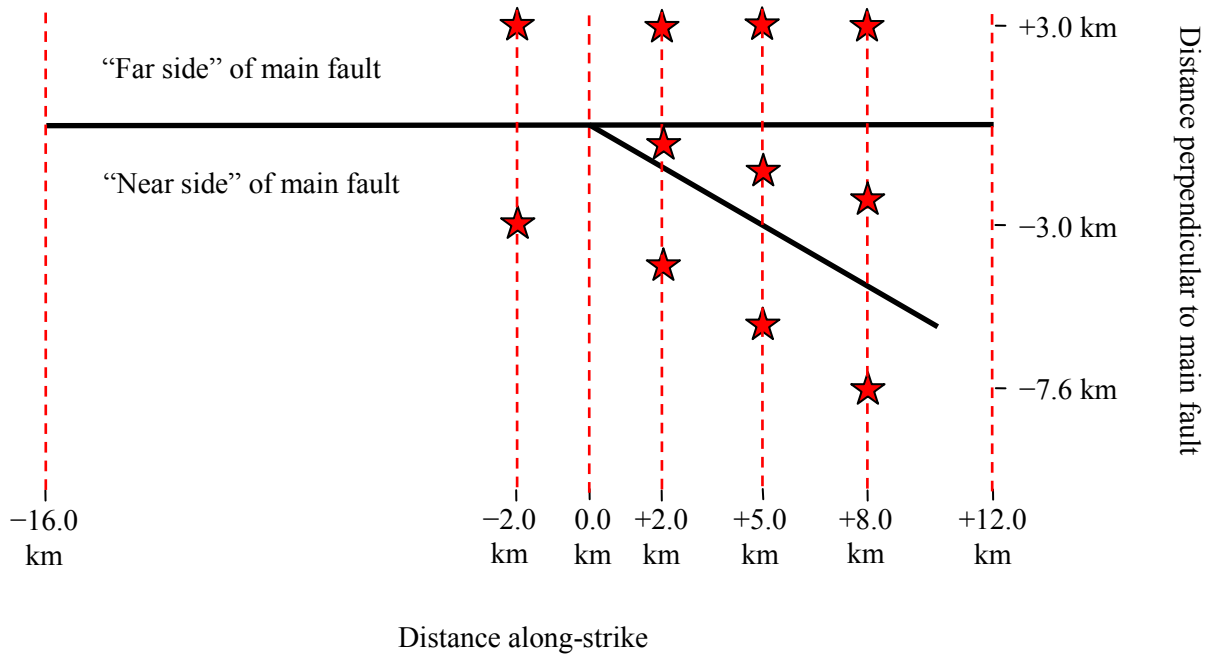
The diagram shows the earth's surface, looking downwards.

There are 11 stations at the earth's surface:

- −2.0 km along strike, 0 km depth, and ±3.0 km perpendicular distance from the main fault trace.
- +2.0 km along strike, 0 km depth, and +3.0 km, −0.6 km, −4.2 km perpendicular distance from the main fault trace.
- +5.0 km along strike, 0 km depth, and +3.0 km, −1.4 km, −5.9 km perpendicular distance from the main fault trace.
- +8.0 km along strike, 0 km depth, and +3.0 km, −2.3 km, −7.6 km perpendicular distance from the main fault trace.

The **near side** of the main fault is in the front of the diagram.
The **far side** of the main fault is in the back of the diagram.
Positive perpendicular distance from the main fault means that the station is on the **far side**.

Each time series file is an ASCII file that contains 7 data fields, as follows.

Off-Fault Time Series Data Fields for TPV18 through TPV21	
Field Name	**Description, Units, and Sign Convention**
t	Time (s).
h-disp	Horizontal displacement, parallel to the fault strike (m). Sign convention: Positive means displacement **to the right** relative to the station's initial position.
h-vel	Horizontal velocity, parallel to the fault strike (m/s). Sign convention: Positive means motion **to the right**.
v-disp	Vertical displacement (m). Sign convention: Positive means displacement **downward** relative to the station's initial position.
v-vel	Vertical velocity (m/s). Sign convention: Positive means motion **downward**.
n-disp	Horizontal displacement, perpendicular to the fault strike (m). Sign convention: Positive means displacement **away from the viewer, into the paper** (that is, away from near side of the main fault and toward the far side of the fault) relative to the station's initial position.
n-vel	Horizontal velocity, perpendicular to the fault strike (m/s). Sign convention: Positive means motion **away from the viewer, into the paper** (that is, away from near side of the main fault and toward the far side of the fault).

The **near side** of the main fault is in the front of the diagram.
The **far side** of the main fault is in the back of the diagram.

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

Off-Fault Time Series File Format for TPV18 through TPV21	
File Section	**Description**
File Header	A series of lines, each beginning with a # symbol, that gives the following information: • Benchmark problem (TPV18 or TPV19) • Author • Date • Code • Code version (if desired) • Node spacing or element size • Time step • Number of time steps in file • Station location • 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  h-disp  h-vel  v-disp  v-vel  n-disp  n-vel` (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 7 numbers, which give the data values for a single time step. The lines must appear in order of increasing time. **C/C++ users:** For all data fields except the time, we recommend using 14.6E or 14.6e floating-point format. For the time field, we recommend using 20.12E or 20.12e format (but see the note on the next page). **Fortran users:** For all data fields except the time, we recommend using E15.7 floating-point format. For the time field, we recommend using E21.13 format (but see the note on the next page). 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.

Note: We recommend higher precision for the time field so the server can tell that your time steps are all equal. (If the server thinks your time steps are not all equal, it will refuse to apply digital filters to your data.) If you use a "simple" time step value like 0.01 seconds or 0.005 seconds, then there is no need for higher precision, and you can write the time using the same precision as all the other data fields. When you upload a file, the server will warn you if it thinks your time steps are not all equal.

Here is an example of an off-fault time-series file. This is an invented file, not real modeling data.

```
# Example off-fault time-series file.
#
# This is the file header:
# problem=TPV18
# author=A.Modeler
# date=2011/01/23
# code=MyCode
# code_version=3.7
# element_size=100 m
# time_step=0.005
# num_time_steps=1600
# location= 3.0 km off fault, 5 km along strike, 0.0km depth
# Column #1 = Time (s)
# Column #2 = horizontal displacement (m)
# Column #3 = horizontal velocity (m/s)
# Column #4 = vertical displacement (m)
# Column #5 = vertical velocity (m/s)
# Column #6 = normal displacement (m)
# Column #7 = normal velocity (m/s)
#
# The line below lists the names of the data fields:
t  h-disp  h-vel  v-disp  v-vel  n-disp  n-vel
#
# Here is the time-series data.
# There should be 7 numbers on each line, but this page is not wide enough
#  to show 7 numbers on a line, so we only show the first five.
0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00   0.000000E+00  ...
5.000000E-03  -2.077270E-85  -2.575055E-83  -2.922774E-86  -3.623018E-84  ...
1.000000E-02  -1.622118E-82  -2.005817E-80  -1.387778E-83  -1.713249E-81  ...
1.500000E-02  -9.020043E-80  -1.114231E-77  -4.402893E-81  -5.424313E-79  ...
2.000000E-02  -1.201684E-77  -1.467704E-75  -4.549845E-79  -5.533119E-77  ...
2.500000E-02  -1.528953E-75  -1.866265E-73  -4.126064E-77  -5.004886E-75  ...
# ... and so on.
```

Part 10: Contour-Plot File Format

The contour-plot files are exactly the same as for benchmarks TPV14 and TPV15.

There are two contour-plot files, one for the main fault and one for the branch fault, as shown here:

Contour-plot files for TPV18 through TPV21	
File Name	**Description**
cplot_main	Rupture times for the main fault.
cplot_branch	Rupture times for the branch fault

The contour plot file lists the locations of all the nodes on the fault surface, and the time at which each node ruptures.

The contour plot file is an ASCII file that contains three data fields, as follows.

Contour Plot Data Fields for TPV18 through TPV21	
Field Name	**Description, Units, and Sign Convention**
j	Distance along strike (m). Sign convention: Positive means a location **to the right** of the junction point. For TPV18 through TPV21, the value of j can range from -16000 to 12000 on the main fault, and from 0 to 12000 on the branch fault. (On the branch fault, j is the distance from the junction point, measured parallel to the strike of the branch fault. Since the branch fault starts one element away from the main fault, in actuality the values on the branch fault can range from 100 to 12000, assuming 100 m node spacing.)
k	Distance down-dip (m). Sign convention: Zero is the earth's surface, and positive means **underground**. For TPV18 through TPV21, the value of k can range from 0 to 15000.
t	Rupture time (s). This is the time at which fault slip-rate first changes from 0 to greater than 0.001 m/s. If this node never ruptures, use the value 1.0E+09.

A pair of numbers (j,k) denotes a point on the fault surface.

The contour plot file consists of three sections, as follows.

Contour Plot File Format for TPV18 through TPV21	
File Section	**Description**

File Header	A series of lines, each beginning with a # symbol, that gives the following information:
	    • Benchmark problem (TPV18 or TPV19)
	    • Author
	    • Date
	    • Code
	    • Code version (if desired)
	    • Node spacing or element size
	    • Descriptions of data columns (7 lines)
	    • Anything else you think is relevant

Field List	A **single line**, which lists the names of the 3 data fields, in column order, separated by spaces. It should be: `j   k   t` (all on one line). The server examines this line to check that your file contains the correct data fields.
Rupture History	A series of lines. Each line contains three numbers, which give the `(j,k)` coordinates of a node on the fault surface, and the time `t` at which that node ruptures. **C/C++ users:** We recommend using 14.6E or 14.6e floating-point format. **Fortran users:** We recommend using E15.7 floating-point format. If a node never ruptures, the time should be given as 1.0E+09. Nodes may be listed in any order.

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 contour-plot file. This is an invented file, not real modeling data.

```
# Example contour-plot file.
#
# This is the file header:
# problem=TPV18
# author=A.Modeler
# date=2011/01/23
# code=MyCode
# code_version=3.7
# element_size=100 m
# Column #1 = horizontal coordinate, distance along strike (m)
# Column #2 = vertical coordinate, distance down-dip (m)
# Column #3 = rupture time (s)
#
# The line below lists the names of the data fields.
# It indicates that the first column contains the horizontal
#  coordinate (j), the second column contains the vertical
#  coordinate (k), and the third column contains the time (t).
j  k  t
#
# Here is the rupture history
-6.000000E+02   7.000000E+03   3.100000E-02
-6.000000E+02   7.100000E+03   4.900000E-02
-6.000000E+02   7.200000E+03   6.700000E-02
-7.000000E+02   7.000000E+03   1.230000E-01
-7.000000E+02   7.100000E+03   1.350000E-01
-7.000000E+02   7.200000E+03   1.470000E-01
# ... and so on.
```