Created in COMSOL Multiphysics 6.3

Phase-Field Modeling of Dynamic Crack Branching

Introduction

This example considers a classical benchmark for phase-field modeling of dynamic fracture, see for example Ref. 1. An instantaneous tensile load is applied to a planar specimen with a pre-existing crack. Initially, the crack propagates perpendicular to the loading direction, after which the crack branches off symmetrically until catastrophic failure occurs. The problem is solved using the AT1 phase-field model.

Model Definition

The AT1 phase-field model is described in terms of the crack surface density function

$$\gamma_l(\phi, \nabla\phi) = \frac{1}{c_w l_{\mathrm{int}}}(w(\phi) + l_{\mathrm{int}}^2 \|\nabla\phi\|^2) \tag{1}$$

where $w(\phi) = \phi$, $c_w = 8/3$ is a normalization constant, and l_{int} is an internal length scale for the phase-field regularization. The fracture energy density is then written as

$$W_{\mathrm{frac}}(\phi, \nabla\phi) = G_{\mathrm{c}}\gamma_l(\phi, \nabla\phi) \tag{2}$$

where G_{c} is the critical energy release rate.

The governing equations for the phase-field damage model include the standard momentum balance

$$\rho\frac{\partial^2\mathbf{u}}{\partial t^2} = \nabla\cdot\sigma \tag{3}$$

together with the phase-field evolution equation

$$\frac{1}{c_w}(w'(\phi) - 2l_{\mathrm{int}}^2(\nabla\cdot\nabla\phi)) = -g'(\phi)\frac{l_{\mathrm{int}}}{G_{\mathrm{c}}}H \tag{4}$$

Herein, $g(\phi)$ is the damage degradation function and $H = \Psi_0^+(\varepsilon)$ is the crack driving force, which is based on a tension–compression split of the strain energy density

$$\Psi(\varepsilon, \phi) = g(\phi)\Psi_0^+(\varepsilon) + \Psi_0^-(\varepsilon) \tag{5}$$

For the AT1 model, Equation 4 does not allow a zero solution for the phase field at zero crack driving force H, which implies that the phase field is unbounded.

A strain energy threshold can be introduced in order to ensure that $\phi = 0$ is a valid solution, so that the phase field is bounded. This threshold can be derived by inserting the zero solution into Equation 4, and solving for the critical value W_{c0} of the crack driving force H.

For the quadratic degradation function $g(\phi) = (1 - \phi)^2$, this results into the condition

$$\frac{3G_c}{8l_{int}} = 2W_{c0} = E\varepsilon_c^2 \tag{6}$$

from which the critical strain $\varepsilon_c = \sqrt{3G_c/8El_{int}}$ can be identified.

The modified crack driving force is then written as

$$H = \max(\Psi_0^+, W_{c0}) \tag{7}$$

This also implies that crack propagation only starts once this threshold is exceeded.

The model geometry and boundary conditions are depicted in Figure 1. Symmetry about the X-axis is used in order to reduce the size of the computational domain.

Plane strain conditions are assumed, and the thickness is arbitrarily set to 1 m.

The pre-existing crack is modeled by prescribing the phase field to a value $\phi = 1$ along the lower-left boundary, while being free to deform.

The instantaneous load is applied over the first time step using a smooth step function.

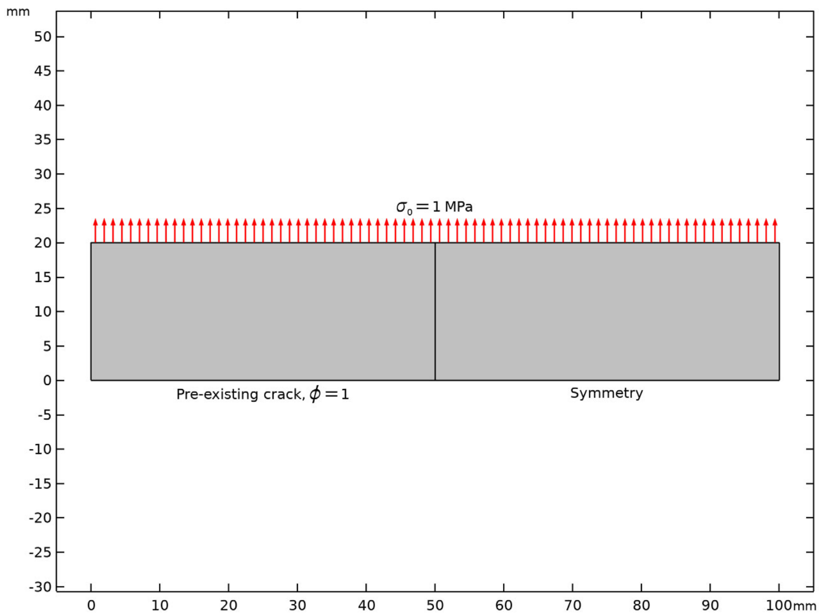

Figure 1: Geometry and boundary conditions for the notched planar tension sample.

The material properties used in this example are inspired by Ref. 1 and listed in Table 1.

TABLE 1: MATERIAL PROPERTIES.

MATERIAL PROPERTY	**VALUE**
Young's modulus	32 GPa
Poisson's ratio	0.2
Density	2450 kg/m^3
Rayleigh wave speed	2125 m/s
Critical energy release rate	3 J/m^2
Internal length scale	0.5 mm

Results and Discussion

The damaged stress is shown in Figure 2 at selected time points. The parts of the domain with damage $d(\phi) = 1 - g(\phi) > 0.95$ have been removed from the plot in order to visualize the crack path.

Crack propagation is initiated after approximately 10 µs. First, the crack propagates perpendicular to the applied load, after which the crack branches off at around 33 µs and spreads quickly throughout the sample, as shown in the snapshots at 45 µs and 75 µs.

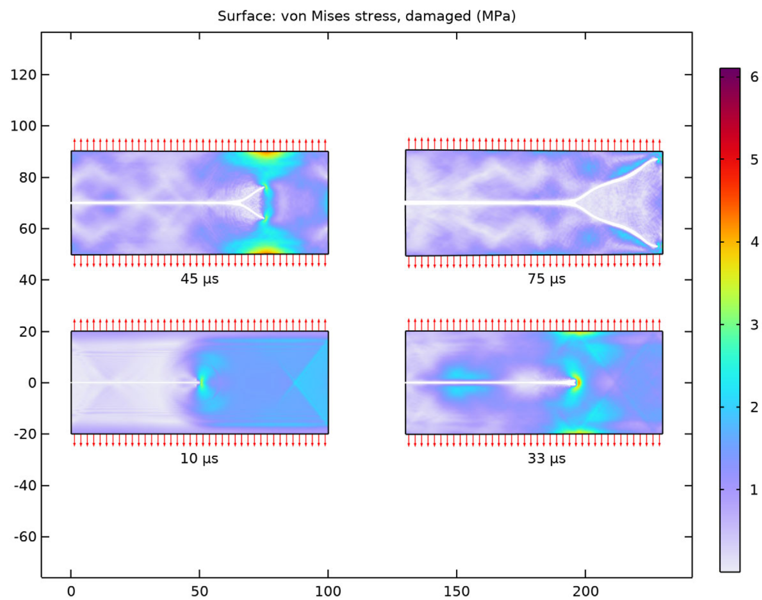

Figure 2: Damaged stress and crack path at selected times.

The phase field at the end of the simulation is shown in Figure 3. The upper crack branch is overlayed with circular points indicating the location of the crack front as a function of time.

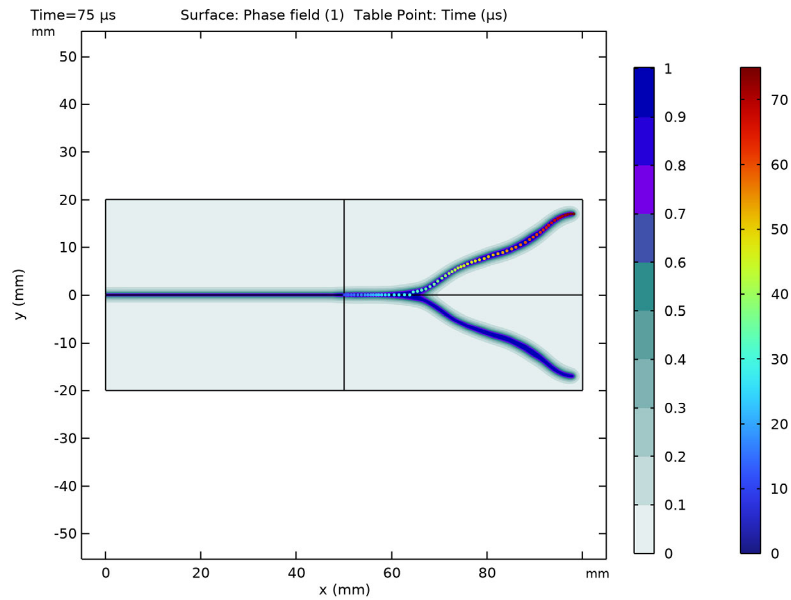

Figure 3: Phase field at the end of the simulation overlayed with colored points indicating the location of the crack front as a function of time.

The total elastic and fracture energy in the sample are plotted as functions of time in Figure 4, and the temporal evolution of the crack length and the crack velocity are shown in Figure 5. The results compare well with those reported in Ref. 1 for the AT2 phase-field damage model.

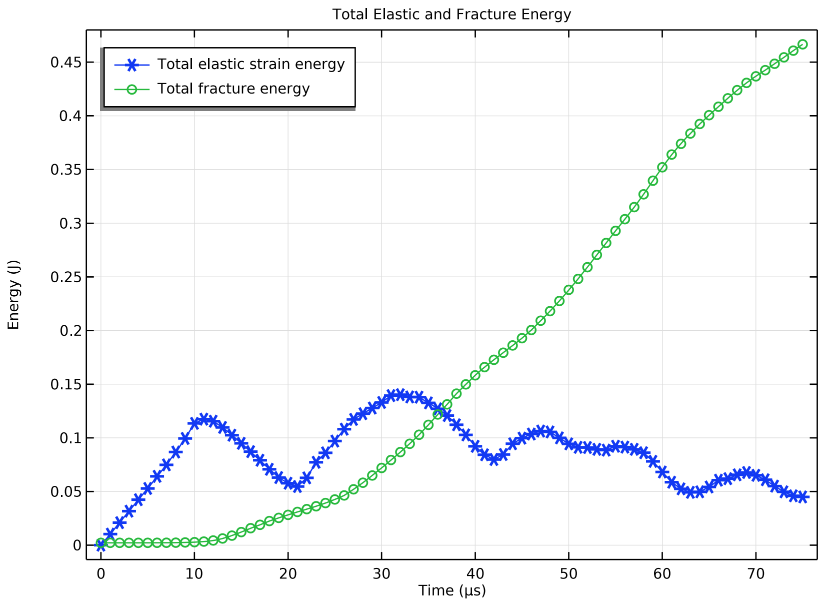

Figure 4: Total elastic and fracture energy as functions of time.

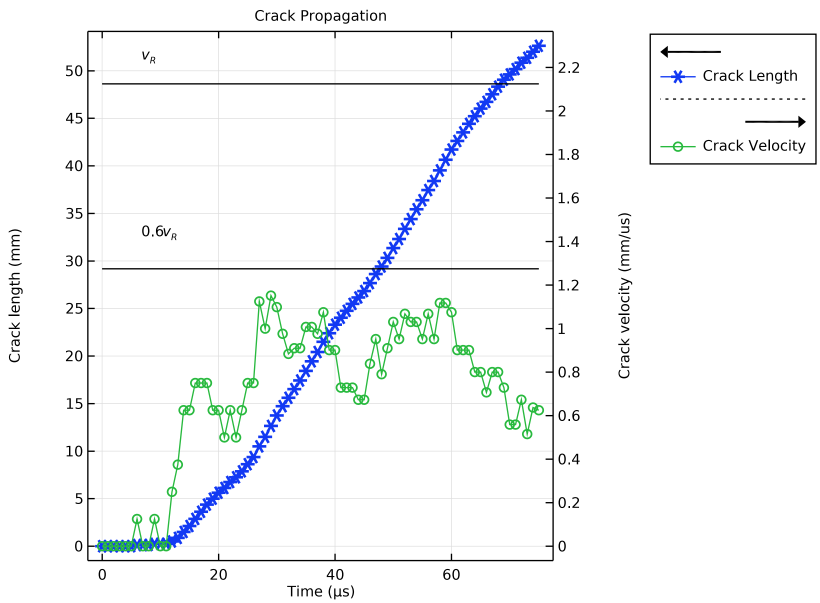

Figure 5: Crack length and crack tip velocity as functions of time. The crack tip velocity stays below 0.6 times the Rayleigh wave speed v_R, as noted in Ref. 1.

Notes About the COMSOL Implementation

To implement the AT1 phase-field damage model, the Solid Mechanics and Phase Field in Solids interfaces are coupled using the **Phase-Field Damage** multiphysics coupling node.

The crack front is tracked in postprocessing by the use of a heuristic indicator function that is the product between the phase field and its time derivative, two quantities that should be maximal at the crack tip during crack propagation. The crack trajectory is then evaluated using a model method. Note that the method editor is only available in the Windows® version of the COMSOL Desktop.

Reference

1. M.J. Borden, C.V. Verhoosel, M.A. Scott, T.J.R. Hughes, and C.M. Landis, *A phase-field description of dynamic brittle fracture*, ICES REPORT 11-14, The Institute for Computational Engineering and Sciences, The University of Texas at Austin, 2011.

Application Library path: `Nonlinear_Structural_Materials_Module/Damage/dynamic_crack_branching`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.

2 In the **Select Physics** tree, select **Structural Mechanics** > **Phase-Field Damage**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies** > **Time Dependent**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
E0	32[GPa]	3.2E10 Pa	Young's modulus
nu0	0.2	0.2	Poisson's ratio
rho0	2450[kg/m^3]	2450 kg/m³	Density
vR	2125[m/s]	2125 m/s	Rayleigh wave speed
Gc0	3[J/m^2]	3 J/m²	Critical energy release rate
lint	0.5[mm]	5E-4 m	Internal length scale
epsc	sqrt(3*Gc0/(8*lint*E0))	2.6517E-4	Critical strain, AT1

Name	Expression	Value	Description
load	1[MPa]	1E6 Pa	Applied load
height	40[mm]	0.04 m	Sample height
width	100[mm]	0.1 m	Sample width
d0	1[m]	1 m	Sample thickness
he	lint/4	1.25E-4 m	Element size
dtmax	5e-8[s]	5E-8 s	Time step
eta	1e-7	1E-7	Residual stiffness factor

Step 1 (step1)

Define a smooth **Step** function that will be used to apply the instantaneous tensile load.

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global** > **Step**.

2 In the **Settings** window for **Step**, locate the **Parameters** section.

3 In the **Location** text field, type `0[s]`.

4 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type `dtmax`.

5 From the **Location definition** list, choose **Beginning of step**.

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `width`.

4 In the **Height** text field, type `height/2`.

5 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	width/2

6 Select the **Layers to the left** checkbox.

7 Clear the **Layers on bottom** checkbox.

8 Click **Build All Objects**.

MATERIALS

Material 1 (mat1)

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.
2. In the **Settings** window for **Material**, locate the **Material Contents** section.
3. In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Young's modulus	E	`E0`	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	`nu0`	1	Young's modulus and Poisson's ratio
Density	rho	`rho0`	kg/m³	Basic

SOLID MECHANICS (SOLID)

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.
2. In the **Settings** window for **Solid Mechanics**, locate the **Thickness** section.
3. In the d text field, type `d0`.
4. Locate the **Structural Transient Behavior** section. From the list, choose **Include inertial terms**.
5. Click to expand the **Discretization** section. From the **Displacement field** list, choose **Linear**.

Symmetry 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.
2. Select Boundary 5 only.

Boundary Load 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.
2. Select Boundaries 3 and 6 only.
3. In the **Settings** window for **Boundary Load**, locate the **Force** section.
4. Specify the $\mathbf{f}_A$ vector as

`0`	x
`load*step1(t)`	y

PHASE FIELD IN SOLIDS (PFS)

Phase Field Model, AT1

1. In the **Model Builder** window, under **Component 1 (comp1)** > **Phase Field in Solids (pfs)** click **Phase Field Model 1**.
2. In the **Settings** window for **Phase Field Model**, type `Phase Field Model, AT1` in the **Label** text field.
3. Locate the **Potential** section. From the **Potential function** list, choose **User defined**.
4. In the Q text field, type `3*phi/8`.
5. Locate the **Phase Field Model** section. In the l_{int} text field, type `lint`.
6. In the D text field, type `3/4`.

Symmetry 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.
2. Select Boundary 5 only.

Prescribed Phase Field 1

Model the pre-existing crack by prescribing the phase field to 1, corresponding to a fully damaged material.

1. In the **Physics** toolbar, click **Boundaries** and choose **Prescribed Phase Field**.
2. Select Boundary 2 only.
3. In the **Settings** window for **Prescribed Phase Field**, locate the **Prescribed Phase Field** section.
4. In the ϕ_0 text field, type `1`.

DEFINITIONS

To set up the AT1 phase-field damage model, start by defining a few help variables as described in the model documentation.

Variables 1

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.
2. In the **Settings** window for **Variables**, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
Wc0	0.5*E0*epsc^2	Pa	Critical strain energy density, AT1
Wfrac	Gc0*pfs.gcl	J/m³	Fracture energy density
Wfrac_tot	2*intop1(Wfrac*d0)		Total fracture energy
Ws_tot	2*solid.Ws_tot	J	Total elastic strain energy

Integration 1 (intop1)

Define the integration operator `intop1` on domain 2 only in order to exclude the pre-existing crack from the evaluation of the total fracture energy. Note that the factor 2 is due to symmetry about the X-axis.

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Selection** list, choose **Manual**.

4 Select Domain 2 only.

5 Locate the **Advanced** section. In the **Integration order** text field, type `2`.

6 From the **Frame** list, choose **Material (X, Y, Z)**.

MULTIPHYSICS

Set up the **Phase-Field Damage** multiphysics coupling. For the AT1 model, we need to enforce that crack propagation only occurs above the critical strain energy density `Wc0`. We can do this using the `max`-operator, the built-in variable `pfdmg1.Ws0` for the positive part of the strain energy density, and the help variable `Wc0`.

Phase-Field Damage 1 (pfdmg1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Multiphysics** click **Phase-Field Damage 1 (pfdmg1)**.

2 In the **Settings** window for **Phase-Field Damage**, locate the **Damage Model** section.

3 From the **Parent material model** list, choose **Linear Elastic Material 1**.

4 From the D_d list, choose **User defined**.

5 In the text field, type `max(pfdmg1.Ws0,Wc0)*pfdmg1.lint/Gc0`.

6 From the **Exclude compressive energy** list, choose **Volumetric only**.

7 Click to expand the **Advanced** section. In the d_max text field, type `1-eta`.

MESH 1

Mapped 1

In the **Mesh** toolbar, click **Mapped**.

Size 1

1 Right-click **Mapped 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domain 2 only.

5 Locate the **Element Size** section. From the **Predefined** list, choose **Extremely fine**.

6 Click the **Custom** button.

7 Locate the **Element Size Parameters** section.

8 Select the **Maximum element size** checkbox. In the associated text field, type `he`.

Size 2

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domain 1 only.

5 Locate the **Element Size** section. From the **Predefined** list, choose **Extremely fine**.

6 Click the **Custom** button.

7 Locate the **Element Size Parameters** section.

8 Select the **Maximum element size** checkbox. In the associated text field, type `10*he`.

9 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

STUDY 1

The crack needs about 75 μs to propagate across the sample. To resolve the dynamics, use the implicit generalized-α method with manual control over the time step. The parameter `dtmax` is set to be approximately equal to the time required for a Rayleigh wave to propagate across one element. The crack speed is typically slower than the Rayleigh wave.

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 From the **Time unit** list, choose **μs**.

4 In the **Output times** text field, type `range(0,1,75)`.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Dependent Variables 1** node.

4 In the **Model Builder** window, under **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** click **Time-Dependent Solver 1**.

5 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

6 From the **Method** list, choose **Generalized alpha**.

7 From the **Steps taken by solver** list, choose **Manual**.

8 Clear the **Interpolate solution at end time** checkbox.

9 In the **Time step** text field, type `dtmax`.

10 In the **Amplification for high frequency** text field, type `0.5`.

Use a tolerance-based convergence criterion on the outer segregated solver.

11 In the **Model Builder** window, under **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Time-Dependent Solver 1** click **Segregated 1**.

12 In the **Settings** window for **Segregated**, locate the **General** section.

13 From the **Termination technique** list, choose **Tolerance**.

Before solving, compute the initial values in order to set up a plot while solving for monitoring the solution process.

14 In the **Study** toolbar, click **Get Initial Value**.

RESULTS

Use a **Mirror** dataset to visualize the complete sample.

Mirror 2D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 2D**.

2 In the **Settings** window for **Mirror 2D**, locate the **Axis Data** section.

3 In row **Point 2**, set **X** to `1`.

4 In row **Point 2**, set **Y** to `0`.

5 Click **Plot**.

Stress (solid)

1. In the **Model Builder** window, under **Results** click **Stress (solid)**.
2. In the **Settings** window for **2D Plot Group**, locate the **Data** section.
3. From the **Dataset** list, choose **Mirror 2D 1**.
4. Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

Surface 1

1. In the **Model Builder** window, expand the **Stress (solid)** node, then click **Surface 1**.
2. In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Solid Mechanics** > **Damage** > **solid.misesdGp - von Mises stress, damaged - N/m²**.
3. Locate the **Expression** section. From the **Unit** list, choose **MPa**.

Deformation

1. In the **Model Builder** window, expand the **Surface 1** node, then click **Deformation**.
2. In the **Settings** window for **Deformation**, locate the **Scale** section.
3. Select the **Scale factor** checkbox. In the associated text field, type `10`.

Surface 1

Use a **Filter** to remove the damaged elements from the plot to visualize the crack trajectory.

Filter 1

1. In the **Model Builder** window, right-click **Surface 1** and choose **Filter**.
2. In the **Settings** window for **Filter**, locate the **Element Selection** section.
3. In the **Logical expression for inclusion** text field, type `solid.dmg<0.95`.

Arrow Line 1

1. In the **Model Builder** window, right-click **Stress (solid)** and choose **Arrow Line**.
2. In the **Settings** window for **Arrow Line**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Solid Mechanics** > **Load** > **solid.fax,solid.fay - Force per deformed area (spatial frame)**.
3. Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.
4. Clear the **Arrow scale factor** checkbox.
5. Clear the **Color** checkbox.
6. Clear the **Color and data range** checkbox.

Deformation 1

Right-click **Arrow Line 1** and choose **Deformation**.

Line 1

1. In the **Model Builder** window, right-click **Stress (solid)** and choose **Line**.
2. In the **Settings** window for **Line**, locate the **Expression** section.
3. In the **Expression** text field, type 1.
4. Click to expand the **Title** section. From the **Title type** list, choose **None**.
5. Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.
6. From the **Color** list, choose **Black**.
7. Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.
8. Clear the **Color** checkbox.
9. Clear the **Color and data range** checkbox.
10. Clear the **Height scale factor** checkbox.
11. Clear the **Tube radius scale factor** checkbox.

Deformation 1

Right-click **Line 1** and choose **Deformation**.

Selection 1

1. In the **Model Builder** window, right-click **Line 1** and choose **Selection**.
2. Select Boundaries 1, 3, 6, and 7 only.

Filter 1

In the **Model Builder** window, under **Results** > **Stress (solid)** > **Surface 1** right-click **Filter 1** and choose **Copy**.

Filter 1

In the **Model Builder** window, right-click **Line 1** and choose **Paste Filter**.

STUDY 1

Step 1: Time Dependent

1. In the **Settings** window for **Time Dependent**, click to expand the **Results While Solving** section.
2. Select the **Plot** checkbox.
3. From the **Update at** list, choose **Time steps taken by solver**.
4. In the **Study** toolbar, click **Compute**.

RESULTS

Plot the total elastic strain energy and the regularized fracture energy.

Total Elastic and Fracture Energy

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Total Elastic and Fracture Energy` in the **Label** text field.

Global 1

1 Right-click **Total Elastic and Fracture Energy** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`Ws_tot`	`J`	`Total elastic strain energy`
`Wfrac_tot`	`J`	`Total fracture energy`

4 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

Total Elastic and Fracture Energy

1 In the **Model Builder** window, click **Total Elastic and Fracture Energy**.

2 In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **Label**.

4 Locate the **Plot Settings** section.

5 Select the **y-axis label** checkbox. In the associated text field, type `Energy (J)`.

6 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Crack Front Indicator

Next, evaluate the trajectory, velocity, and length of the crack. To start with, we first need to track the crack front as a function of time. Here, we will identify the crack front using a suitable indicator function. A simple condition that can be used in this case is to look for the maximum of the product between the phase field and its time derivative.

1 In the **Results** toolbar, click **Evaluation Group**.

2 In the **Settings** window for **Evaluation Group**, type `Crack Front Indicator` in the **Label** text field.

Surface Maximum 1

1 Right-click **Crack Front Indicator** and choose **Maximum** > **Surface Maximum**.

2 Select Domain 2 only.

3 In the **Settings** window for **Surface Maximum**, locate the **Expressions** section.

4 In the table, enter the following settings:

Expression	Unit	Description
`if(phi*phit>1e3,phi*phit,-1)`	`1/s`	`Crack front indicator`

5 Click to expand the **Configuration** section. From the **Point type** list, choose **Mesh vertices**.

6 Select the **Include position** checkbox.

7 In the **Crack Front Indicator** toolbar, click **Evaluate**.

Phase Field (pfs)

Visualize the coordinates of the crack front on top of the phase field to verify that the moving front is captured correctly.

1 In the **Model Builder** window, under **Results** click **Phase Field (pfs)**.

2 In the **Settings** window for **2D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Mirror 2D 1**.

4 From the **Time (µs)** list, choose **Last (75)**.

Table Point 1

1 In the **Phase Field (pfs)** toolbar, click **More Plots** and choose **Table Point**.

2 In the **Settings** window for **Table Point**, locate the **Data** section.

3 From the **Source** list, choose **Evaluation group**.

4 From the **x-axis column** list, choose **x (mm)**.

5 From the **y-axis column** list, choose **y (mm)**.

6 From the **Data column** list, choose **Time (µs)**.

7 In the **Phase Field (pfs)** toolbar, click **Plot**.

APPLICATION BUILDER

With the coordinates of the crack front at hand, the crack length and velocity can be evaluated using a model method. Note that the method editor is only available in the Windows® version of the COMSOL Desktop.

In the **Home** toolbar, click **Application Builder**.

NEW METHOD

1 In the **Application Builder** window, right-click **Methods** and choose **New Method**.

2 In the **New Method** dialog, type `computeCrackTrajectory` in the **Name** text field.

3 Click **OK**.

computeCrackTrajectory

1 In the **Application Builder** window, under **Methods** click **computeCrackTrajectory**.

2 Copy the following code into the **computeCrackTrajectory** window:

```
// Extract crack length and crack velocity
EvaluationGroupFeature eg = model.result().evaluationGroup("eg1");
eg.run();
double[][] data = eg.getReal(); // {time, indicator function, x-coordinate, y-
coordinate}
int n = data.length;

// Initialize a 2d array for output data
double[][] out = new double[n][5]; // {time, x-coordinate, y-coordinate, crack
length, crack velocity}
double xprev = 50.0; // Initial crack front x-coordinate
double yprev = 0.0; // Initial crack front y-coordinate

// Make sure the crack front data are initialized correctly
out[0] = new double[]{data[0][0], xprev, yprev, 0.0, 0.0};
data[0][2] = xprev;
data[0][3] = yprev;

// Compute crack length and velocity
double clength = 0.0;

for (int i = 1; i < n; i++) {
  double x = data[i][2];
  double y = data[i][3];
  if (data[i][1] < 0) {
    // if the indicator function is negative, no crack propagation has been
detected
    x = xprev;
    y = yprev;
  }
  // Crack length
  double dx = x-xprev; // x-increment
  double dy = y-yprev; // y-increment
  clength += Math.sqrt(Math.pow(dx, 2)+Math.pow(dy, 2));

  // Crack velocity
  double dt = data[i][0]-data[i-1][0]; // Time step
  double vx = dx/dt; // x-velocity
  double vy = dy/dt; // y-velocity
  double cvel = Math.sqrt(Math.pow(vx, 2)+Math.pow(vy, 2));

  // Update output data array and previous coordinates
  out[i] = new double[]{data[i][0], x, y, clength, cvel};
  xprev = x;
  yprev = y;
}

String tblTag = "tblcrk";
TableFeature tbl;
if (contains(model.result().table().tags(), tblTag)) {
```

```
    tbl = model.result().table(tblTag);
    tbl.clearTableData();
  } else {
    tbl = model.result().table().create(tblTag, "Table");
  }
  tbl.setColumnHeaders(new String[]{"Time (us)", "x-coordinate (mm)", "y-
  coordinate (mm)", "Crack length (mm)", "Crack velocity (mm/us)"});
  tbl.setTableData(out);
  tbl.label("Crack Trajectory");
```

METHODS

1 In the **Home** toolbar, click **Model Builder** to switch to the main desktop.

Add a **Method Call** to `computeCrackTrajectory` in order to run it.

2 In the **Developer** toolbar, click **Method Call** and choose **computeCrackTrajectory**.

GLOBAL DEFINITIONS

Compute Crack Trajectory

1 In the **Model Builder** window, under **Global Definitions** click **ComputeCrackTrajectory 1**.

2 In the **Settings** window for **Method Call**, type `Compute Crack Trajectory` in the **Label** text field.

3 Click **Run**.

RESULTS

Crack Propagation

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Crack Propagation` in the **Label** text field.

Crack Length

1 Right-click **Crack Propagation** and choose **Table Graph**.

2 In the **Settings** window for **Table Graph**, type `Crack Length` in the **Label** text field.

3 Locate the **Data** section. From the **Plot columns** list, choose **Manual**.

4 In the **Columns** list, select **Crack length (mm)**.

5 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

6 Click to expand the **Legends** section. Select the **Show legends** checkbox.

7 Find the **Include** subsection. Select the **Label** checkbox.

8 Clear the **Headers** checkbox.

Crack Velocity

1 Right-click **Crack Length** and choose **Duplicate**.

2 In the **Settings** window for **Table Graph**, type `Crack Velocity` in the **Label** text field.

3 Locate the **Data** section. In the **Columns** list, select **Crack velocity (mm/us)**.

Crack Propagation

Compare the crack velocity with the Rayleigh wave speed.

Right-click **Crack Velocity** and choose **Global**.

Rayleigh Wave Speed

1 In the **Settings** window for **Global**, type `Rayleigh Wave Speed` in the **Label** text field.

2 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
vR	mm/us	Rayleigh wave speed
0.6*vR	mm/us	

3 Click to expand the **Coloring and Style** section. From the **Color** list, choose **From theme**.

4 Click to expand the **Legends** section. Clear the **Show legends** checkbox.

Crack Propagation

In the **Crack Propagation** toolbar, click **More Plots** and choose **Table Annotation**.

Table Annotation 1

1 In the **Settings** window for **Table Annotation**, locate the **Data** section.

2 From the **Source** list, choose **Local table**.

3 In the table, enter the following settings:

x-coordinate	y-coordinate	Annotation
5	1.5	$0.6 v_R$
5	2.3	v_R

4 Select the **LaTeX markup** checkbox.

5 Locate the **Coloring and Style** section. Clear the **Show point** checkbox.

Crack Propagation

1 In the **Model Builder** window, click **Crack Propagation**.

2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.

3 Select the **x-axis label** checkbox. In the associated text field, type `Time (µs)`.

4 Select the **Two y-axes** checkbox.

5 In the table, select the **Plot on secondary y-axis** checkboxes for **Crack Velocity**, **Rayleigh Wave Speed**, and **Table Annotation 1**.

6 Locate the **Legend** section. From the **Layout** list, choose **Outside graph axis area**.

7 In the **Model Builder** window, click **Crack Propagation**.

8 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

9 In the **Crack Propagation** toolbar, click **Plot**.