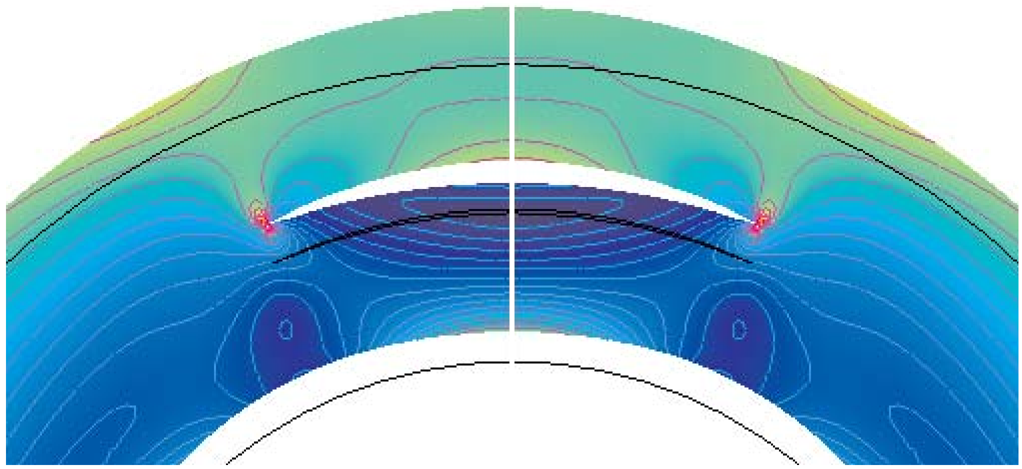
Cracked Heat Exchanger

Introduction to the Lesson

This example simulates the stress concentration due to a crack formed between two layers in a heat exchanger pipe. A gap such as this severely impedes heat flow. The resulting temperature difference creates thermal stresses that tend to propagate the crack along the tube interface. For this problem the stress is caused by thermal expansion in the pipe. Thermal expansion is not part of the structural application in the base COMSOL software. It is included in the structural mechanics module. In this problem we show how to add thermal expansion to the base equations.



Key Instructive Elements

The key elements we hope you will learn in this example are

1 Practice simulating heat transfer and plane stress

2 Adding thermal expansion to Navier's structural equations

3 Adding mesh near a point to improve the solution accuracy near a singularity

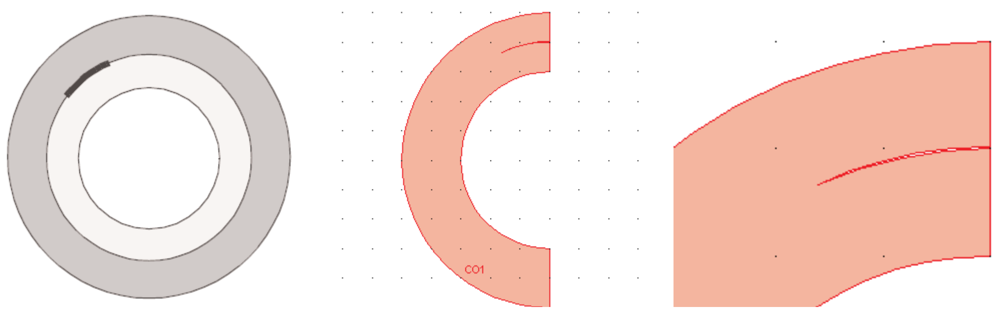
PHYSICS

Tubes for heat exchangers in pulp mills must stand up to considerably different chemicals on their inside and outside surfaces. The tubes thus generally consist of two layers, each made of a different stainless steel. A flaw in the joint between the inner and outer layers, here visible as a crack, severely impedes heat flow. The resulting temperature difference creates thermal stresses that tend to propagate the crack along the tube interface.

In developing a model, note that the thermal and elastic properties of the two steels are similar enough that you can consider them essentially equal. In addition, you can handle the crack as a thermal insulator. The restriction that the crack surfaces are constrained and won't penetrate each other constitutes a contact condition: If they are separated, the normal force is zero, and if they are in contact, the normal displacements must be equal, and the normal forces are inward.

At the crack tip there is a singularity in stress distribution, as the stress will highly increase around that point. To get accurate results a mesh refinement is needed around the crack tip.The following simulation assumes that the crack surfaces are separated. In other words, set the normal forces to zero and then compute the displacements. This analysis also assumes that the crack will propagate according to its opening mode.

The real geometry is a 3D geometry, but it can be simplified as there are several symmetries. Consider the tube to be long enough so that the strain along the length is null. This means that you do not need to model the complete 3D geometry, only a 2D cross section using a plane strain assumption. It is also possible to model only a half of the 2D geometry as there is symmetry for geometry, loads, boundary conditions, and material.



This model involve both heat transfer and linear stress analysis using the plane strain assumption.The unknown of the system are:

- u, displacement along x-axis
- v, displacement along y-axis
- T, temperature.The equation of the system are:
- The Navier's equation including the thermal effect:

$$-\nabla \cdot \left(c\nabla u + \alpha T\right) = K$$

where *c* is the matrix including the Young's modulus and Poisson' ratio, α is the matrix handling the thermal expansion and *K* the volume forces.

Note: α in the above equation stands for the COMSOL PDE coefficients and not the thermal expansion coefficient.

See "The Plane Strain Application Mode" on page 193 in the Modeling Guide for details.

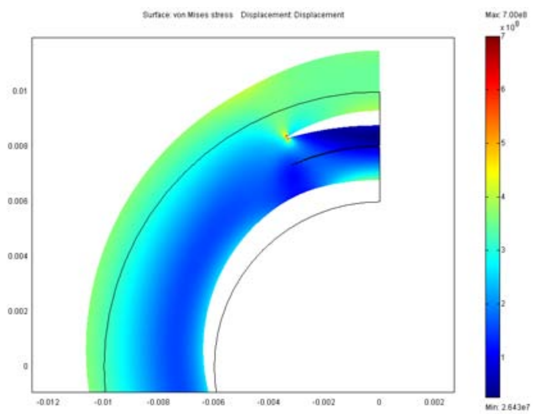
- The heat equation

$$-\nabla \cdot (k \nabla T) = Q$$

where k is the thermal conduction and Q the heat source.

Results and Discussion

In the following plot you can see the stress concentration around the crack. The plot shows only a smaller range of the stress values (between 26 MPa and 700 MPa). Because of the stress concentration, using the complete range for the stresses means that you cannot see the stress distribution in all the tube.



With a mesh size of 1×10^{-5} around the crack tip, the analysis shows good accuracy as the stress in the immediate neighborhood of the tip has converged.

You can also see that the crack is opening, a so-called modal deformation.

Modeling in COMSOL Multiphysics

Modeling this system in COMSOL Multiphysics requires coupling two application modes: Heat Transfer by Conduction and Plane Strain.

The equations link by adding thermal expansion from the Heat Transfer application mode to the solids model given by the plane strain formulation of Navier's equations.

All displacements result only from the heat source.

The thermal stress can be written as:

$$\sigma = \begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \end{bmatrix} = D\left(-\begin{bmatrix} \alpha \\ \alpha \\ \alpha \\ 0 \end{bmatrix}(T - T_{ref})\right)$$

Here the reference temperature will be equal to 0, and D is the elasticity matrix. So we have:

$$\sigma = \begin{bmatrix} -E\dfrac{\alpha}{(1-2\nu)} \\ -E\dfrac{\alpha}{(1-2\nu)} \\ -E\dfrac{\alpha}{(1-2\nu)} \\ 0 \end{bmatrix} T$$

The σ_x and σ_y stress should be added to the α coefficient for the equation system in the modeling domain. The normal stress variables also need to be edited to add the thermal part.

In this model there is a stress concentration due to the crack. At the crack tip the stress tends to infinity as the mesh size approaches zero. In the immediate neighborhood of the crack tip, the stress must approach a constant value for the model to converge.

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 In the **Model Navigator** click the **Multiphysics** button.
2 Select **COMSOL Multiphysics>Heat Transfer>Conduction>Steady-state analysis** in the list of application modes.
3 Click **Add**.

4 Select **COMSOL Multiphysics>Structural Mechanics>Plane Strain>Static analysis** in the list of application modes (do NOT click **Add** yet!) .

5 Select **Lagrange - Cubic** in the **Element** list.

6 Click **Add**.

7 Select **Plane Strain (pn)** in the **Ruling application mode** list. This makes a linear stationary solver the default solver.

8 Click **OK**.

OPTIONS AND SETTINGS

1 From the **Options** menu, choose **Axes/Grid Settings**.

2 Enter the following axis and grid settings:

AXIS		GRID	
x min	-0.016	x spacing	0.002
x max	0.016	Extra x	
y min	-0.012	y spacing	0.002
y max	0.012	Extra y	-0.001 0.001

3 Click **OK**.

4 From the **Options** menu, choose **Constants**.

5 Define the following constant names and expressions:

NAME	EXPRESSION	DESCRIPTION
E_S	21e10	Young's modulus
nu_S	0.3	Poisson's ratio
al_S	17.3e-6	Thermal expansion coefficient
rho_S	7870	density
C_S	449	Heat capacity
k_S	82	Thermal conductivity
Tout	100	Outer temperature
Tin	0	Inner temperature

6 Click **OK**.

GEOMETRY MODELING

1 Draw a circle centered at (0, 0) with a radius of 0.01.

2 Draw a circle centered at (0, 0) with a radius of 0.008.

3 Choose **Object Properties** from the **Draw** menu. Type `0.0075` in the **Radius** edit field and `0.00055` in the **y-center** edit field.

4 Click **OK** to change the radius and center position of the circle.

5 Draw one more circle centered at (0, 0) with a radius of 0.008.

6 Draw a circle centered at (0, 0) with a radius of 0.006.

7 Draw a rectangle with corners at (0, -0.01) and (0.01, 0.01).

8 Select all objects.

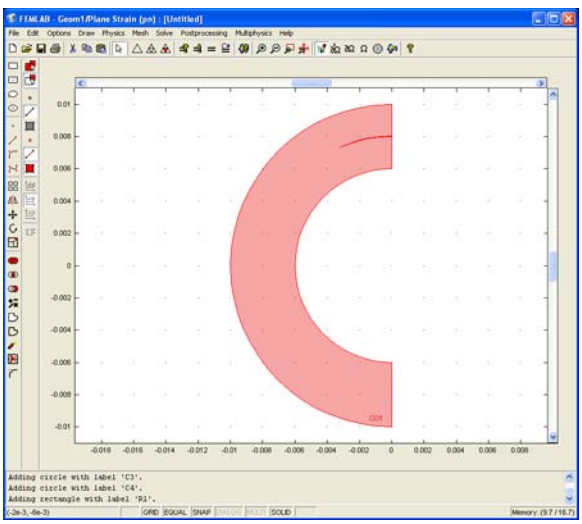
9 Choose **Create Composite Object** from the **Draw** menu.

10 Type the Boolean expression `(((C1-(C2-C3))+C3)-C4)-R1` in the **Set formula** edit field.

11 Clear the **Keep interior boundaries** check box.

12 Click **OK** to create the geometry.

The circles centered at (0, 0) form the tube's basic geometry. The noncentered circle creates a crack with a maximum width of 50 µm. The rectangle removes the right half of the domain to take advantage of the symmetry along the y-axis.



PHYSICS SETTINGS

Solver Parameters

The default settings for the equation form is the General form, but as this is a linear model, you can use the Coefficient form, which makes it easier to enter the multiphysics couplings. To change the solution form:

1. From the **Physics** menu, choose **Model Settings**.
2. Select **Coefficient** in the **Equation system form** list.
3. Click **OK**.

Boundary Conditions

1. From the **Physics** menu, open the **Boundary Settings** dialog box.
2. Select boundary number 1 to 5.
3. Select the **Rx** check box. This sets the displacement to zero in the x-direction.
4. Click **OK**.
5. In the **Multiphysics** menu, select **Heat Transfer by Conduction (ht)**.
6. Open the **Boundary Settings** dialog box and enter the following settings:

BOUNDARY	1-5,10-11	6-7	8-9
Type	Insulation	Temperature	Temperature
T_0		`Tout`	`Tin`

7. Click **OK**.

Point Settings

When using these boundary conditions, the solution is not unique: stresses and strains are uniquely determined, but there is no specification of the rigid-body translation in the y direction. To force uniqueness, you need to lock the displacement of one point only. On the symmetry line, the condition requires that the x displacement u is zero. Therefore it is sufficient to force the y displacement v to zero in one point on this line.

1. From the **Multiphysics** menu, choose **Plane Strain (pn)**.
2. From the **Physics** menu, choose **Point Settings**.
3. In the **Point Settings** dialog box, select point 4.
4. Select the **Ry** check box. This sets a constraint along the y-axis and hence locks the lower right corner of the geometry.
5. Click **OK**.

Subdomain Settings

Now apply the material properties on the domain:

1 From the **Physics** menu, choose **Subdomain Settings**.

2 In the **Subdomain Settings** dialog box, click the **Material** tab and set the following material properties:

SUBDOMAIN	1
E	E_S
ν	nu_S
ρ	rho_S

3 Click **OK**.

4 From the **Multiphysics** menu, choose **Heat Transfer by Conduction (ht)**.

5 Open the **Subdomain Settings** dialog box. This time, set the material properties for the Heat Transfer application mode:

SUBDOMAIN	1
k *(isotropic)*	k_S
ρ	rho_S
Cp	C_S

6 Click **OK**.

7 Choose **Equations systems** and **Subdomain Settings** in the **Physics** menu. Here you can couple the two physics by adding the Temperature T in the Navier's equation in the convection coefficient α.

8 Click the α tab and enter in the first column, second and third line, the following expressions:

$\alpha(2,1)$	-E_S/(1-2*nu_S)*al_S 0
$\alpha(3,1)$	0 -E_S/(1-2*nu_S)*al_S

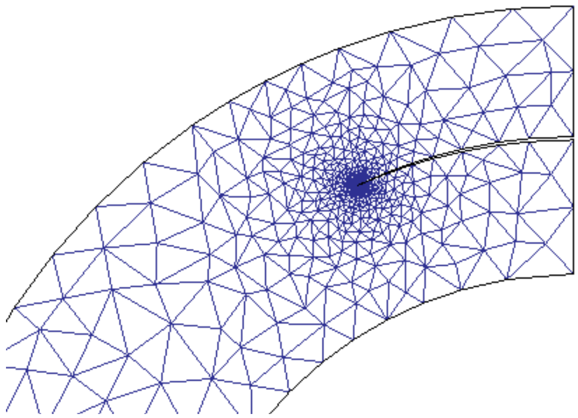
9 Click the **Variables** tab and add the following expression to the definition of the normal stresses (make sure you add them; do NOT erase the existing expressions:

VARIABLE	EXPRESSION TO ADD
sx_pn	-E_S/(1-2*nu_S)*al_S*T
sy_pn	-E_S/(1-2*nu_S)*al_S*T
sz_pn	-E_S/(1-2*nu_S)*al_S*T

MESH GENERATION

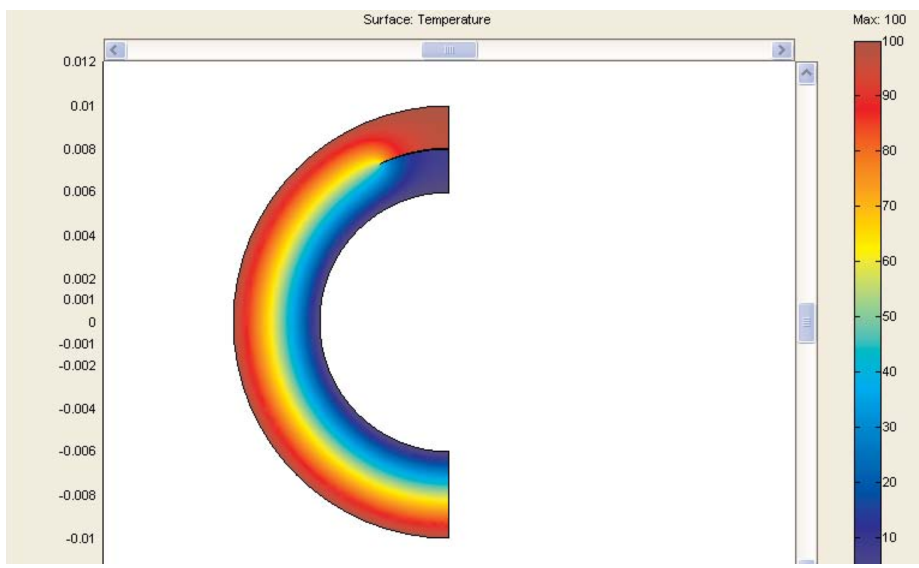
Because of the singularity on the crack tip, you must increase the mesh density around that point:

1 Open the **Mesh Parameters** dialog box from the **Mesh** menu.
2 Click the **Subdomain** tab and type `1e-3` in the **Maximum element size** edit field.
3 Click the **Point** tab.
4 Select point number 3.
5 Type `1e-5` in the **Maximum element size** edit field.
6 Click **Remesh** and then click **OK.**



SOLVE PROBLEM

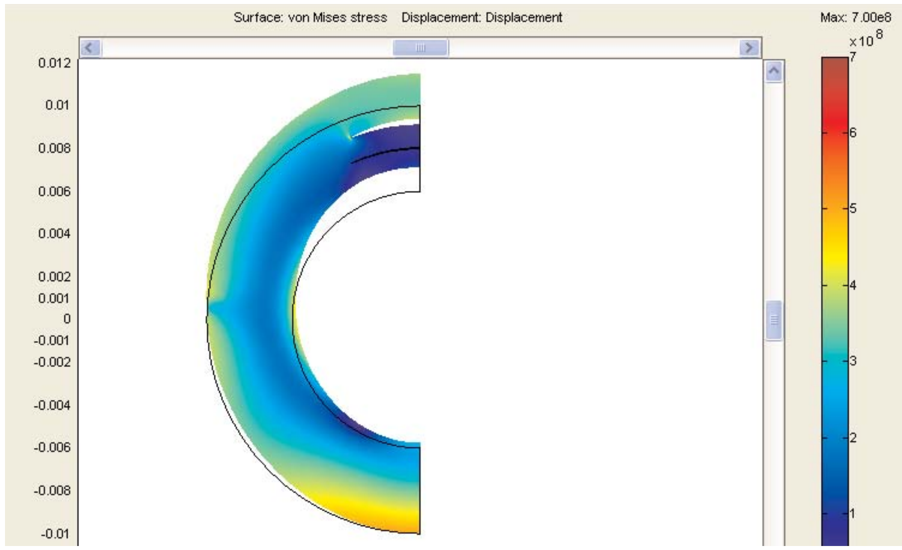
- This is a linear problem, to speed the solution let us change the solver from nonlinear to linear: Choose **Solve > Solver Parameters** and change the solver from **Stationary Nonlinear** to **Stationary Linear**. Hit **OK**.
- Pick **Solve > Solve Problem** or the Button with the equals sign on it. Once the problem is solved, you should see the following thermal plot



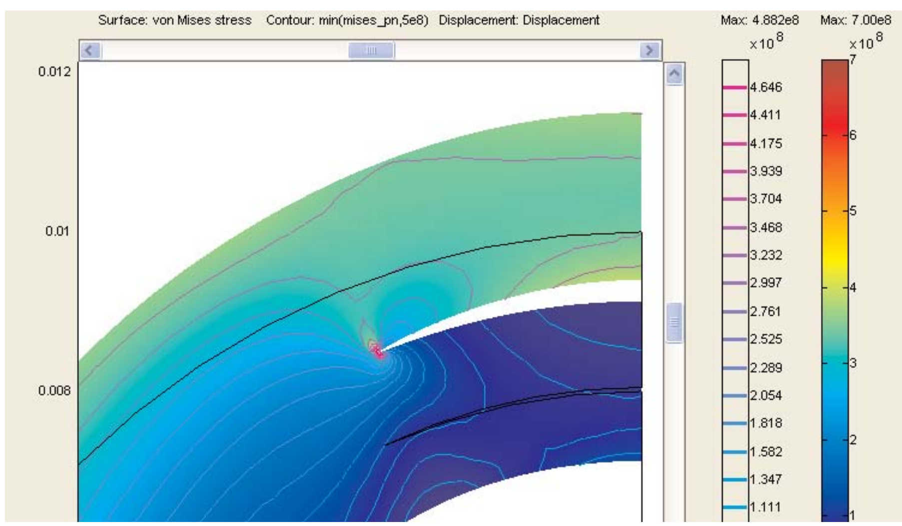
POSTPROCESSING AND VISUALIZATION

- Open **Postprocessing > Plot Parameters**. On the General page, <u>select</u> the **Deformed shape** check box
- Click the **Deform tab**. Under **Deformation data**, select **Displacement (pn)** in the **Predefined quantities** list.
- Click the **Surface tab** and select **von Misses stress (pn)** in the **Predefined quantities** list

under **Surface data**. Due to the stress concentration on the crack tip the automatic range is not suitable. Click **Range**. Deselect the **Auto** check box and type `7e8` in the **Max** edit field. Click **OK** to set the range and again hit **OK** to apply the postprocessing. You should see



- Pick **Postprocessing > Plot Parameters** and then the **Contour Tab**. Select the **Contour plot** checkbox in the upper left corner and enter `min(mises_pn,5e8)` as the **expression**, Hit **OK**. Zoom in on the top of the geometry.



RESULTS AND DISCUSSION

In the plots above you can see the stress concentration around the crack. The plot shows only a smaller range of the stress values (between 26 MPa and 700 MPa). Because of the stress concentration, using the complete range for the stresses means that you cannot see the stress distribution in all the tube.With a mesh size of 1e-5 around the crack tip, the analysis shows good accuracy as the stress in the immediate neighborhood of the tip has converged. You can also see that the crack is opening due to thermal deformation.