**RESERVOIR RUPTURE**

A comprehensive treatment of reservoir rupture is beyond the scope of what we will do today (or in this course; those interested in more details should track down *Grosfils*, J. Volc. Geotherm. Res., 2007), but provided that Dtmc > 3\*Rmc then a spherical reservoir will rupture at the crest and solid.sphi – the hoop stress, operating in and out of the plane of the screen – will indicate the marginal stress state accurately; for shallower reservoirs we would also have to calculate the stress parallel to the reservoir wall but within the plane of the screen rather than perpendicular to it, creating new Variables etc to do so; we’ll ignore this for now, but in any exploring you do using just solid.sphi make sure that Dtmc remains >3\*Rmc!

For rupture to occur, we must pressurize the system to the point at which some part of the wall first becomes positive (tensile) so that an intrusion becomes possible!

**USE MODEL PROVIDED FOR TODAY**

**OPEN COMSOL, THEN SELECT FILE > OPEN AND POINT TO FILE**

**SKIP TO BLACK LINE BELOW AND PROCEED**

* Using your base model from Day 2, with **reservoir depth of 7.5 km** and **radius of 1 km**, create a new **1D Plot Group** and add a **Line Plot** to it. Rename this **Rupture**.
* For Selection, we want to plot the stresses along the reservoir wall, so select these two boundaries **(#6 and #7**)
* Set **y-axis** data to **solid.sphi** and **Unit** to **MPa**
* Set **x-axis** data to the **Expression -z-Dcmc+Rmc**.
  + If you think this through, you’ll see that this will vary from 0 at the crest to 2\*Rmc at the base, and so it is a measure of depth below the crest.
* **Plot** the result with **OP = 5 MPa**, and you should see that at the crest (x-axis value of 0 m) the hoop stress solid.sphi is roughly -165 MPa, while at the reservoir base (x-axis value of 2000 m) the hoop stress is close to -215 MPa.
  + What this means is that the stress at the margins of the reservoir, in spite of the OP, remains highly compressive! To rupture in tension and feed a dike, the y-axis value must exceed zero. Wherever it does so first is where rupture will initiate, but apparently a lot more OP will be required…
  + What uplift magnitude accompanies this OP condition?
* Fortunately, we have a model where this is easy to do! Go back to your Parameter list and **change** **OP** from 5[MPa] to a much larger value, say **100[MPa]** and then rerun the model.
  + Re-examine your Rupture line graph. The values are smaller, and the line looks a bit more chunked up, but they’re still quite negative. **Try a bigger value, maybe 300[MPa].**
  + This gets close, with solid.sphi values at the crest of about -10 MPa, but the line is now really wiggly. To fix this we need to adjust the mesh so that we have more elements (higher resolution calculation) at the reservoir wall, which will increase the accuracy but also the duration of our calculation. Right now the distance between nodes is too great, so the solution is being averaged across too large an area.
    - Select **Mesh**, and set **Element Size** to **Extremely Fine**, then re-run the model and observe the difference this makes.
      * The Rupture plot is still wiggly, but a lot less so. It would be nice to improve this further, and to build a mesh that is high resolution where we need it (near the reservoir) and coarser elsewhere.
    - Return **Element Size** to **Normal**, then set **Mesh sequence type** to **User-controlled mesh**.
      * Note that new options have now appeared under Mesh in the Model Builder window.
    - Next, **RC Mesh > More Operations > Edge** and then, under Selection choose the two reservoir wall segments **(#6 and #7**).
      * We’re going to refine things near these edges that define the reservoir wall, which is where we are really interested in the solution.
    - Next, under **Mesh**, **select and drag upward the Edge 1 item** under Mesh
      * The order of items under Mesh should now be Size, then Edge, then Free Triangular.
    - Next, **RC Edge 1** and select **Size**. Click on the new Size icon beneath Edge 1, and in the *Settings* window for **Element Size** click the **Custom** button.
      * Activate the **Maximum Element Size** and replace the default value with **50**. Now, to see if you have it right, click **Build All**.
        + If all has gone well, you’ll now see a new, more sophisticated version of the mesh with big elements most places but very small ones near the reservoir…
    - **Re-run your model** and again examine the Rupture plot. It should be looking a *lot* better… the wiggles are almost gone.
  + To conclude, continue adjusting the OP until the crest of the reservoir exceeds zero. When you identify the OP at which the line *just* exceeds zero, you have found the OP needed to initiate intrusion (and relieve the magma pressure as magma escapes vertically via a dike!).
    - You should find an OP of about 332 MPa is required.
      * Note that the lithostatic stress at the top of the reservoir is (2600 \* 9.8 \* 6500) / 1e6 = 165.5 MPa, and we expected the OP value to be twice this… which would be 331.24. Not too bad!
    - You might also take a peek at the uplift, which is now on the order of 18 cm, before moving on.

**PUT IT TOGETHER**

* Imagine the Widely Used Model is correct. Calling on high pore pressure, or something similar, is physically plausible, but it means that only very low OP is required to cause rupture, thereby relieving the pressure and halting the inflation.
  + What amount of uplift is produced if the OP is small, say 5 MPa?
  + Is this because gravity is engaged? Set g = 0 m/s2 to test the idea… if the model is running properly, you’ll see that the uplift is identical!
    - This tells us that uplift depends only on the magnitude of the OP (should make sense… is there a gravity term in the Mogi equation?), and since the OP is the same in both the loaded and unloaded situations the uplift is identical as well!
* If special conditions such as high pore pressure are not invoked, and so high OP is needed to cause rupture, what amount of uplift results? You did this before (see top of page!)
* Considering the data above, and the uplift data typical of active systems, what are the implications for the state of the reservoir and the surrounding system if an elastic model properly describes the situation?
* Lets imagine for a moment that it is difficult to generate very high overpressures. Can you think of any condition that would allow the host rock to accommodate/relieve a build of pressure, permitting significant uplift, without rupturing?

**AFTERMATH**

Because you used Parameters to construct your model, you can rapidly change things there and rerun. For example, change Dcmc to 5[km] and rerun the model. When you look at the Stress>Surface plot you’ll quickly see that the location of the reservoir has changed. If you examine the Rupture plot, you’ll discover that the entire reservoir wall is now in tension – which can’t be realistic since the first break would have relieved the pressure. To identify the first rupture pressure, you’ll need to experiment again with the OP, reducing the magnitude until only a single spot on the reservoir wall—the place where first rupture occurs—exceeds zero. By varying the depth and OP you can quickly characterize the pressure required for rupture as a function of reservoir depth, how the uplift changes, etc.

With a little more effort, you could add equations to consider rupture orientation (there are two primary directions parallel to the reservoir wall, not the single one we have assumed here… as noted previously, for shallow reservoirs you’d need to look at both, and would discover that the rupture location, fracture/intrusion alignment, etc., can all be predicted and matched to examples observed in the field). To get more sophisticated yet, you could consider different geometries (oblate and prolate ellipsoids, say), maybe start to think about where the intrusions will go once they escape the reservoir (their path will depend on the stress in the rock), perhaps look at faulting (for instance caldera formation), or even add factors such as tectonic stresses (which requires moving to fully 3D model formulations)!

As a final note, COMSOL is excellent for addressing mechanical problems in the geosciences and many other fields, but there is a *lot* more that the software is good for. We will return to COMSOL next week when we think about thermal cooling problems, but for anyone interested in exploring what types of things COMSOL can do more broadly I’d encourage you to visit their web page at [www.comsol.com](http://www.comsol.com) .