

Geology 131: Physical Volcanology

Fall 2015

Using COMSOL: Magma Reservoir Inflation and Failure

One of the clearest indicators of volcanic unrest is surface uplift. While far from a smoking gun—eruptions can happen without surface deformation, surface deformation can occur without producing an eruption, and surface deformation can result from many processes (i.e., new magma injecting into a shallow system can cause uplift, but the additional heat can also drive changes in shallow hydrothermal systems to produce similar effects)—a lot of useful information can be deduced from surface displacement data.

This week you have already explored surface deformation using an analytical Mogi model, which for very specific circumstances can be used to predict surface uplift in response to changes in pressure or volume of a spherical reservoir at sufficient depth. This analytical model lent itself well to exploration using Excel, and it is widely used (and often abused) by volcanologists.

Results from this analytical model can be used to calibrate and test more general numerical models, and that is one of our goals today. We will use a powerful finite element software package called COMSOL Multiphysics. Once built, the numerical model has considerable advantages over analytical approaches because it can be used to examine a much wider array of conditions... something we'll also touch upon today and in the next class session!

Okay, time to get started!

Introduction: Building a Model in COMSOL (v. 5.1)

*To start, the **Model Wizard** window will help you set up the type of simulation you want to perform, in this case one looking at solid structural mechanics (as opposed to fluid dynamics, electromagnetics, or something else...).*

Start COMSOL! (It's only on the Mac side of the computer.)

1. **MODEL WIZARD** – tell the software what kind of model you plan to run.
 - a. Select **File > New** and then click the **Model Wizard** icon
 - b. Select **2D Axisymmetric**. This tells COMSOL that we will be assuming axisymmetry for our model, which means we can build a single slice through our reservoir-crust system that will then get rotated about a central axis to build the geometry we want.
 - c. On the *Select Physics* page, select **Structural Mechanics > Solid Mechanics (solid)**; click **Add**, then once Solid Mechanics has been added to the “added physics interfaces” box proceed to the next page by clicking the green **Study** arrow (i.e., Next up: Study settings...)
 - d. On the *Studies* page, select **Preset Studies > Stationary**. This tells COMSOL that we will be performing a static computation, meaning we are interested in the final outcome, not all the intervening processes that might be important if we wanted a time-dependent solution (we'll work with a time-dependent model next week).
 - e. Now that you have told COMSOL what kind of problem style you'll be working with, click **Done** to begin building the specific model itself.

Now that the model framework has been defined, the Model Builder and Settings windows are where you can control all aspects of your model... build the geometric shape representing the reservoir in the host rock, apply material properties and boundary conditions, collect your results, etc. The Graphics and “Assorted” (Messages, Progress, etc.) windows will be self-explanatory.

*These windows can be manipulated as you work. If you ever get too enthusiastic moving them around and need to reset them, choose **Windows > Desktop Layout > Reset Desktop**.*

2. ESTABLISH GLOBAL DEFINITIONS

Global Definitions are attributes of the model that will be needed throughout, i.e. they will be available as data for all elements of the model you construct. These can take various forms, but in simplest terms this is where you can enter constants, define variables you may wish to examine, and even write equations

for things you'd like to have calculated. Thinking carefully about what you may need down the road in a model is an important skill to develop, and one we will work on a bit today.

a. PARAMETERS

Parameters are single-value variables. They can be given a fixed value (e.g., used to define your gravitational acceleration), but they can also be used to define variables you will change from model run to model run so that you can examine how changes affect the results. For example, you might define a parameter you call "DtC" that establishes the depth-to-center of your magma reservoir, setting it in one model to 5 km (run, get results) then changing it to 7 km for the next model run.

You'll begin by using parameters to define the size of your model's study area along with some other pieces you'll use later.

- i. **RC** (right click) **Global Definitions** and choose **Parameters**.
- ii. With Parameters selected, in the Settings window you'll see four columns: Name, Expression, Value and Description. Under **Name**, in the first row type **rsize**. Under **Expression**, enter **15[km]**, and note that this provides a *Value* that is reported in the appropriate SI units. You can then add a **Description** as well (e.g. "**size of the study region in the r-direction**"), a good habit to be in so you remember later on what everything is!
- iii. Now, create parameters **g** ($9.8[\text{m/s}^2]$), **dens** ($2600[\text{kg/m}^3]$), **zsize** ($15[\text{km}]$), **Rmc** (radius, magma chamber) $1[\text{km}]$, **Dcmc** (depth, center of magma chamber) $7.5[\text{km}]$, and **Dtmc** (depth, top of magma chamber) $\text{Dcmc}-\text{Rmc}$ (type in this expression, not the numbers, and note that the reported value is 6500 m).
 1. **Note:** if you ever want to eliminate gravitational effects, for instance to simulate a Mogi model result directly, you can set $g = 0[\text{m/s}^2]$ and anywhere gravity plays a role in an equation you have built this gravitational effect will be zeroed out...

b. VARIABLES

We will come back to this, but variables basically allow you to define an equation. For instance, if needed we could define sphvol as $(4/3)*\pi*\text{Rmc}^3$. Equations are most useful for obtaining values (stress, porosity, temperature-dependent viscosity, etc.) throughout the model space.

- i. **RC** (right click) **Global Definitions** and choose **Variables**. This puts the Variables holder into place for later...

Now, with a few parameters defined and ready to be used, we are ready to build our model, which we do under Component 1 the Model Builder window.

For an axisymmetric model, $r = 0$ km defines the vertical symmetry axis about which the cross-section we define will be rotated. In our case we will want to construct a rectangle (rotated will create a cylinder) with a half-circle cut out of it (rotated will create a sphere). This is our reservoir within the host rock, and we will set that host rock up so that the top edge lies at $z=0$... that level will be our surface of the Earth!

3. GEOMETRY 1

- a. Rectangle 1 – this will become our host rock region
 - i. **RC Model 1 > Geometry 1** and choose **Rectangle**
 - ii. Go to the *Settings* window for Rectangle
 1. For **Width** type **xsize**, then set **Height** to **zsize**. Notice that xsize is red... that means COMSOL doesn't know what this is. Whoops! We entered the wrong thing... we called the Parameter for our model width rsize, not xsize. Set **Width** to **rsize** instead of xsize to fix the problem...
 - iii. Set the **Base** as **Corner** with **r = 0** and **z = -zsize**. This will ensure that the top of our rectangle (cylinder once rotated) lies at $z = 0$ km (i.e., surface of the Earth) and extends down to a depth of -15 km into the Earth.
 1. To verify that you have it right, in the *Settings* window click **Build Selected**. You should now see a 15 km by 15 km square anchored at $(r,z) = (0,-15 \text{ km})$
- b. Circle 1 – this will create the hemisphere we use to make our reservoir
 - i. **RC Model 1 > Geometry 1** and choose **Circle**
 - ii. Go to the *Settings* window for Circle
 1. Set **Radius = Rmc**
 2. Set **Base > Center**
 - a. $r = 0$
 - b. $z = -D_{cmc}$
 - iii. Click **Build Selected**. You should now have a circle that is 1 km in radius overlapping the left hand edge of the rectangle you created
- c. Difference 1 – this will carve our reservoir out of the host rock
 - i. **RC Geometry 1** and choose **Booleans and Partitions > Difference**

- ii. Under Geometry 1 *highlight Difference 1*, then examine the *Settings* window
 1. Make sure the button next to the “Objects to Add” area is **blue**, indicating that this area is **active**. If it isn’t, click on the button to turn it blue and activate it
 2. Click on the rectangle you created in the Graphics window; **r1** should appear in the “**Objects to Add**” area. This is the base feature from which something will be cut out.
 3. Now, activate the “**Objects to Subtract**” space, and explore until you figure out how to add the circle to it. When you have it, **c1** should appear in the area. This is the shape that will be used as the cookie cutter!
- iii. Click **Build Selected**. You’ve now created a new shape – a slice through our cylindrical volume with a hemisphere carved out of it. Picture rotating this about the $r=0$ axis... your model will be a cylinder with a sphere carved out of it at 7.5 km depth. You’re ready now to begin characterizing the space further.

With the basic model shape (volume) defined, you’re now ready to characterize things further.

*For the next step you will explore how to create a mesh that breaks our area (volume) up into small **elements**. We will use simple triangular elements to define this mesh, and the corners of each triangle, the **nodes**, are where our numerical solutions will be obtained when we finally complete and run the model.*

Having more nodes means higher resolution and greater solution accuracy, but it also means more computational time is required. You’ll explore the interplay between these a bit as you move forward, this balance between accuracy and runtime is a key concept in numerical simulation...

4. MESH

- a. In the *Model Builder* window **select Mesh 1** and examine the accompanying *Settings* window
- b. In the *Settings* window set **Sequence type** to **Physics-controlled mesh**. This effectively tells COMSOL to assess the existing model geometry and mesh it automatically to try and obtain a decent solution for the type of model you’re building.
 - i. The other option, User-controlled, requires more care but allows you to tune the mesh characteristics to the needs of your study; you’ll do more of this later on.

- c. Click **Build All**, and the mesh upon which stresses and displacements (etc.) are calculated will be created.
 - i. Explore the effects of **varying Element size**, selecting Build All each time you make a change to observe the results.
 - 1. More elements (the number is reported in the Messages window at bottom right) give you greater spatial precision—and more degrees of mathematical freedom, which requires more calculations, thus more time to solve, etc. You'll need to match the mesh characteristics to your problem needs... you *could* design a mesh that will take 4 hours to solve, but if the solution is essentially identical to a much cruder mesh that requires 6 s to solve you will need to have a very good reason to wait for the former model results!
- d. When you are done, return **Element size** to **Normal** and **Build All**.

Now that you have created your shape (volume) and meshed it (broken it into chunks for calculation purposes), it is time to tell it what the material is made of and how it will respond! If our host rock was made of wood it might react very differently, when squeezed or stretched, than if it is made out of rock... and of course different rocks behave very differently as well!

As by now you are becoming familiar with the notations I have been using, and where to find different things, I will begin to use a slightly more abbreviated notation. Be sure to ask me if anything is unclear or you can't find something!

5. MATERIALS – (where would you look for this do you think?)

a. RC Materials > Add Material

- i. A new *Add Material* window appears. Use it to **Search** for **Granite**., which is one of the few built-in geological materials (we could easily define others ourselves). **RC** on **Granite**, select **Add Material to Component 1** (if your model had lots of different bits and you wanted to add to one of them, you could).
- ii. When “Granite” is showing under Materials in the *Model Builder* window, you can close the *Add Material* window you used to search for granite.

b. Go to the *Settings* window for Granite

- i. You'll see that Domain 1 has already been added to the Selection pane; this tells COMSOL the material in the shape we built is granite. If we had lots of shapes (maybe we use a triangle to define a conical volcano at the surface), we could assign different

materials to different parts of the model, but that's not needed today.

- ii. You could add or remove domains using the same approach you used to assign shapes to the Add and Subtract portions of the Differencing process earlier...
- c. On the *Settings* window, look at the Material Contents and notice that the density, Young's Modulus and Poisson's Ratio have all been set to default values.
 - i. An elastic material is characterized by four elastic constants—Young's Modulus, Poisson's Ratio, Bulk Modulus, and Shear Modulus—that are all related to one another via two equations. Thus, by defining two elastic constants we know the others as well!
 - ii. *NOTE: Shear Modulus = Young's Modulus / (2*(1+Poisson's ratio)). The default Youngs Modulus of 60 GPa, with Poisson's Ratio = 0.25, thus indicates a Shear Modulus = 24 GPa. Poisson's Ratio is assumed constant for our purposes, but if you want to employ a different Shear Modulus then you need to enter a new Young's Modulus Value.*
 - iii. You might want to use a different density in the future, so replace the starting value with the Parameter ***dens*** you defined earlier as the value assigned to rho in the Material Contents table. I.e., delete the existing 2600 value and type *dens* there instead. That way, if you change the value for *dens* under your Parameter listing, it will automatically assign your new value to the material!

At this stage you have built your area (volume), broken it into chunks for calculation purposes, and have told COMSOL to treat the material as granite, so that when you compress it the response will be that of granite and not wood or some other material.

Our hunk of granite, however, is still just floating in space... to make it behave properly we need to lock it down so that it acts as if we had carved our cylinder of rock out of a much larger region. Put another way, any bit of rock beneath our feet has weight (so we'd better assign gravitational loading), and acts to buttress adjacent areas of rock (so our outer edge, at $r=15$ km, had better not expand into free space, but should behave as if it was in equilibrium with rock at greater lateral distances), and can't sink (buttressed by the rock beneath our model space). Finally, if we want our magma chamber to be something other than an empty cavity, we had better set things up so that we simulate magma in that cavity, pressurized enough to stay open at bare minimum. To put all these conditions into place, we need to finish building our model by applying a set of Boundary (applied to edges) and Body (applied to a volume) Load conditions, and any initial conditions needed, to approximate the conditions that we know (or assume) exist in the real world.

6. SOLID MECHANICS

a. BOUNDARY LOADS

The goal of the current problem is to examine, quantitatively, how pressurization of a spherical cavity within a rock host concentrates stresses and creates surface displacements. We thus need to apply appropriate boundary conditions to ensure that our host rock behaves in a physically correct way!

- i. **RC Solid Mechanics**, select **Roller** to add a roller boundary to the model.

1. In the Settings window, preceding as you did in similar selection processes, select the $r = 15$ km (right, #5) and $z = -15$ km (bottom, #2) edges to add each to the Selection. This will define each of these edges as a Roller boundary.

- a. A roller boundary allows no displacement normal to the edge but free displacement parallel to it, literally behaving as if the boundary was on rollers! The bottom of our cylinder can move horizontally but not vertically while the outer edge of our cylinder can move only vertically.

- b. If you think about it, this simulates buttressing by the surrounding rock quite effectively!

- ii. **RC Solid Mechanics**, select **Boundary Load**

1. In the Settings window, select the two curved boundaries of the spherical cavity (#6 and #7) to add them to the Selection pane

2. Because we will be treating our magma, which exerts load on the wall of the reservoir, as a fluid, it will push outward without creating shear stresses.

- a. To help us do this, set the **Coordinate System** to **Boundary System 1** and note what this does to the *Force* table options at the bottom of the window... this change will allow us to apply stress (force per unit area) normal (n) and/or tangential (t1) to the curved surface automatically, whereas we'd have to do some complex math to translate our global r- and z-coordinates into wall-normal stresses otherwise.

3. Set **Force > Load Type** to **Load defined as force per unit area**. This just means we will apply stress directly to the surfaces.

- a. In the *Force* table, set the load in the **n-direction** (normal to the surface) as **MagmaLd**. Note that COMSOL makes the text red, indicating that it doesn't yet know what MagmaLd means. *That's okay, we will come back to this in a moment...*

b. BODY LOADS

- i. **RC Solid Mechanics**, select **Body Forces > Gravity**.

The rock in our volume has weight, and this creates a depth-dependent stress in the vertical direction of magnitude $\rho g z$ where ρ is the rock density (look at your material, currently set by *dens* to 2600 kg m⁻³), g is gravitational acceleration, and z is depth. **In COMSOL, compressive stresses are negative by convention and extensional stresses are positive.**

1. Make sure that you assign gravity to your slice... make sure Selection is Activated (button is blue) and click on the rectangle (Domain #1).
2. At the bottom of the window, we'll use our own Parameter here, so **replace -g_const with -g**.

c. INITIAL CONDITIONS

When subjected to a vertical load, the instantaneous elastic response of the host rock will yield horizontal stresses one-third the magnitude of the vertical load when Poisson's Ratio = 0.25. This endmember situation is called a state of *uniaxial strain*, and it occurs because you have taken rock with no loads, snapped your fingers, and then suddenly applied weight all at once.

Over long time periods, however, rock flows to try and equilibrate this imbalance, creating an endmember that is called a *lithostatic* stress state in which the vertical and horizontal stresses are equal. Of these two endmembers, the condition we will use in our models is lithostatic as it is far more commonly approximated within the Earth where stress states have been measured.

- i. **RC Solid Mechanics > Linear Elastic Material 1** and select Initial Stress and Strain 1. Under Initial Stress, set the 1-1, 2-2 and 3-3 boxes (i.e., upper left to lower right diagonal) to $\text{dens} * g * z$. This ensures that the host rock stress is in what is called a lithostatic state.

d. MAGMA LOAD

- i. Finally, Return to Global Definitions > Variables, where we need to define a MagmaLd equation. **MagmaLd will be the name, and what it is equal to (the other side of the equation) is what will be entered for the Expression.** To remain open and at

equilibrium with its surroundings, the pressure within the magma chamber must be equal in magnitude but opposite in direction to the stress in the surrounding host rock. Assuming for simplicity that the rock and magma have the same density, we can thus define an equation (thinking carefully about the signs involved) that keeps the magma pressure and host rock in equilibrium as follows:

$$\text{MagmaLd} = \text{dens} * g * \text{Dtmc} + \text{dens} * g * (z - \text{Dtmc}).$$

ii. As a variable, enter the **Name** and **Expression** using the equation above.

1. Notice that, because it knows all of the units involved, it recognizes that the associated unit is Pascals – a good sign!
2. At the crest of the reservoir, where $z = \text{Dtmc}$, the second term is zero, and the MagmaLd simply equals the weight of the rock column above it, i.e. its pressure is pushing upward just as hard as the load from the overlying rock is pushing downward.
3. As the depth increases, the weight from the column of magma gradually increases... just as in a swimming pool, where pressure from the water is zero at the top and maximum in magnitude at the bottom due to the water's weight, so too will the MagmaLd increase in magnitude from the crest to the base of the reservoir. Because we are assuming for simplicity that magma and rock weight are equal, this increase in pressure with depth exactly mimics the increase in weight of the rock column at the same depth, so the reservoir is in equilibrium throughout.

iii. To perturb this equilibrium, however, we will want to increase the magma pressure by a uniform amount (the overpressure, **OP**), reflecting for instance addition of new magma into the chamber or bubble nucleation. This will subject the surrounding rock to an extra pressure, so modify your original entry from above so that the final MagmaLd equation becomes:

$$\text{MagmaLd} = \text{dens} * g * \text{Dtmc} + \text{dens} * g * (z - \text{Dtmc}) - \text{OP}.$$

1. Think through the signs... do they make sense? If not, see me before proceeding.
2. To wrap up, since we've just used something called the **OP**, we need to define this **Parameter**. Do so now, assigning it a value, for the time being, of **5[MPa]**.

7. RUN MODEL

- a. **Save** your model before running it!
- b. To run the model, **RC Study 1**, then select **Compute** (green 'equals' sign)
- c. How long did it take? (Check Messages area... mine was about 2 s)

Congratulations, you just successfully executed your first COMSOL model! ☺

Now, what sorts of things can you learn from the data results that COMSOL provides? Time to explore some of the results and figure out how best to visualize them. If you haven't stood up and stretched in a while, now is a good time to do so!

8. EXPLORE RESULTS

There are a lot of different things you can explore, and you may want to delve into this on your own in greater depth when we're done.

- a. In the *Model Builder* window, expand **Results**, then expand **Stress** (solid) and then expand **Surface 1**
- b. Note the deformed shape on the plot in the *Graphics* window. This is highly exaggerated (my run by a factor of about 37x!) and is distracting in practice, so let's disable it.
 - i. Under Surface 1, **RC Deformation** and select **Disable**

Note: if we delete something, it is gone; if we disable it we can always turn it back on again...)

Note the change in the Plot. Without the distortion, things look more sensible, but it is always a good idea to check and make sure that the results make good sense and work the way we expect. Conduct the following tests:

1. With **Surface 1** selected, in the Settings window replace **solid.mises**, which is what was initially plotted, with the Expression **solid.sz** (COMSOL's notation for the stress in the z-direction). Change the **unit** to **MPa** so that the numbers become more manageable, then click **Plot**.

Note how values are increasingly negative (compressive) with depth. Perform by hand a $\text{dens} \cdot g \cdot z$ calculation to figure out what the vertical stress should be at the bottom of the 15 km deep model space, and compare it with the most compressive value reported in the model.

This may be a good moment to explore zooming in and out of the model Plot! When you are zoomed in nicely along the

bottom edge you can click on the plot, and the value will be reported in the “Evaluation 2D” tab along the base (near Messages tab).

There should be an excellent match, roughly -382 MPa.

2. Since we are trying to create a lithostatic state of stress, the vertical and horizontal stresses away from the reservoir (near the reservoir equilibrium is perturbed by the OP we applied) should be equal. Test this by **Plotting** three different **Expressions**, one at a time: **solid.sz-solid.sr**, **solid.sz-sphi** and **solid.sr-solid.sphi**. In COMSOL, solid.sr is the horizontal stress in the radial direction, and solid.sphi is the horizontal stress called the hoop stress, aligned orthogonal to (in/out of) the computer screen.

The differences should all Plot as zero away from the reservoir, while the areas near the reservoir will show signs that the inflation we caused via our OP value has produced a pattern of stress that will induce deformation; **if this isn't what you are getting, please see me before proceeding.**

3. We also want to make sure our MagmaLd equation is working the way we expect, i.e. that our magma reservoir is expanding outward (we have applied an overpressure) not imploding (pulled when we meant to push, probably meaning we have a sign wrong somewhere) or something similar! To test this, **plot** the **Expression w** (displacement in the vertical direction) and then **u** (displacement in the radial direction).

If you think about what should happen, at the crest w should be moving in the $+z$ direction (upward), at the base w should be moving in the $-z$ direction (downward) and at the mid-depth of the reservoir, where MagmaLd pushes only laterally, w should be zero. Is this what you see?

Similarly, displacement u should be positive (right-moving) at the mid-depth of the reservoir, and should be zero at the crest and base. **If you don't see these patterns, see me before proceeding.**

If all has gone well, then you have obtained a working solution, and you should save it, but now we want to use the model to examine the questions of interest: what surface uplift is created when the reservoir is subjected to overpressure and inflates.

Then, moving beyond the Mogi model, tomorrow we will explore how much overpressure is required to rupture the reservoir wall, leading to intrusion and creating the possibility for an eruption...

9. VISUALIZING SURFACE UPLIFT

Now that we have run and tested our model, we will take a look at the surface deformation produced by the inflation (OP) we used. To do this, you will need to learn how to create a graph in COMSOL. While you can always export the data and manipulate them in Excel, the COMSOL plots will update automatically each time you make a change and re-run the model, so for rapid visualization knowing how to work with graphs in COMSOL is really useful.

- a. In the *Model Builder* window, **RC Results** and select **1D Plot Group** to add a new Plot group to the Results. 1D essentially means a graph, 2D means a surface plot like the one you've been looking at, and 3D... well, explore 3D on your own if there's time!
- b. Now, on the new 1D Plot Group that was added, **RC 1D Plot Group**, select **Line Graph**. RC on Line Graph 1 then rename it Uplift.
 - i. **Note:** it is good practice to rename anything that you create in the Results so that what it is/does/plots is clear to anyone!
 - ii. In the *Settings* window, for the Selection, choose the upper surface (#4). This is where we extract data from for our graph.
 - iii. For the y-axis expression, choose w (vertical displacement), because that's what we would like to explore.
 1. **Note:** if you don't know what COMSOL calls something, you can use the little + sign or green/red arrows to explore all of the things that COMSOL calculates by default.
 - iv. For the x-axis Parameter, select **Expression** and then type r because we want to look at uplift w (y-axis) as a function of radial distance r (x-axis) for the surface we selected.
- c. **Plot** the result, and you should see a graph of vertical displacement as a function of radial distance, with a maximum vertical displacement a little less than 0.003 m.

If you want other units for the plot, you can change them! Maybe try cm for the y-axis and km for the x-axis... but you'll need to keep track of what you use if you export the data for use in Excel, i.e. to compare with a Mogi solution. **For now, to be safe, leave all units as meters!**

- d. **Save** your model, then explore the effect of varying some of the parameters.
- In **Parameters**, change your **OP** to **15[MPa]**, and re-run the model. Going back to your graph, what happens to the uplift when you triple the overpressure (jot down the answer)?
 - What happens if you change **OP** to **-5[MPa]**, in effect an underpressure, and re-run the model (**after each change from here on out, you'll need to re-run the model**)? Using the surface plot as you did before, do the displacements around the reservoir behave the way you expect?
 - Reset the **OP** to **5[MPa]**. Now, let's test the impact of having a shallower reservoir. Change the depth to the **center** of the reservoir so that it lies **3 km** below the surface. Do you see that the uplift becomes narrower and taller (~1.8 cm now)?
 - What happens if you change the reservoir radius but keep the depth to the top the same? To simulate what you had in iii, where the top of the reservoir was 2 km below the surface, change the Parameters so that the **radius** is **0.5 km** but the **top** remains at **2 km** depth. What happens to the surface displacement? (should be a bit above 0.3 cm)
 - Finally, what happens if you set the reservoir **center** to **7.5 km** and make the **radius** **5 km**, so that you're looking at a huge shallow reservoir?

Before looking at the graph, **examine the surface plot of u** . Notice how the areas well away from the reservoir are no longer zero? This indicates that the deformation around the reservoir is "feeling" the bottom and lateral edges, which suggests that the model space is too small for the reservoir size we created. In essence, the stress perturbation from the reservoir has not dropped off to zero like we saw before when we plotted stress differences (e.g., $\text{solid-sr} - \text{solid-sz}$), and so it would like to push these down (bottom) and out (right edge) where ideally we want that perturbation to have no effect at these locations. Since it can't, that effort at displacement will have to be directed somewhere else!

To see this a different way, look at the graph, and zoom into the area around **$r=15$ km**. Notice how the value is on the order of 6 cm?

Now, try setting **rsize** and **zsize** to **50 km** each, so that the edges of the model are much further from the reservoir. When you **zoom** into the **15 km** distance now, note that the displacement is much smaller, on the order of **3 cm**. Because your reservoir stress perturbation "felt" the edges of the model, a factor of 2 error was

introduced. **Avoiding undesirable edge effects is an important consideration when building a numerical simulation!**

- e. **Return your model to a radius of 1 km, a center depth of 7.5 km, and model size of 15 km in each direction before continuing.**

Now that you know how to examine the displacement in COMSOL, time to see how well this numerical approximation does when compared with the analytical Mogi solution. If we create identical scenarios – same depths, elastic parameters, etc – then the numerical model results should closely approximate the Mogi solution. That's what we want to test, and the test is really important since we want to be sure that COMSOL is giving us an approximate solution we can trust!

Keep in mind, however, that the Mogi solution, while exact, is limited to specific circumstances: spherical reservoirs, far from the surface, located within an isotropic host rock. Our COMSOL model has no such limitations! We could make our reservoir some other axisymmetric shape (ellipsoidal for example), or add depth-dependent thermal effects that change elastic properties, or pile a volcano on top, or look at much shallower reservoirs than the Mogi solution permits, or... the limitations are subject only to our imagination and modeling skills!

10. IN-LAB ASSIGNMENTS – Test Vertical Displacement

a. Design and Run a Mogi Model for a 1 km radius reservoir in Excel

- i. Make certain as you do so that you don't violate the Mogi assumptions, i.e. keep your reservoir depth much greater than the radius of your reservoir (I'll suggest at least a 5:1 ratio between these values to keep things really clean while still yielding a good surface displacement result). Use an overpressure value large enough to produce a meaningful (not crazy!) vertical surface displacement, maybe a few up to ten cm.

b. Design an Identical COMSOL Model, then Run it Two Ways. Bring Results Into Excel, Plot on Same Graph as Mogi Solution.

- i. To extract COMSOL data to Excel, see Appendix.
- ii. Check to make sure that your elastic constant values are identical for the Mogi and COMSOL models!
- iii. **First COMSOL Model:** run using a Mesh resolution of “Normal” and extract results to Excel.
- iv. **Second COMSOL Model:** run using a Mesh resolution of “Extremely Fine” and extract results to Excel.

c. Assess, and when ready discuss your answer and insights with me.

- d. Now... try replacing your initial **sphere** with an **ellipsoid** instead so that you can define an ellipsoidal reservoir. You will need to define major and minor axis Parameters, and deal with the differencing (subtracting it from the rectangle), but everything else will, happily take care of itself! Note that this is not a bad way to proceed in general... if you set the major and minor axes of an ellipse equal to one another what do you have? ☺

- i. What is the equation for the volume of an ellipse?
- ii. Using this equation, define two oblate ellipses with the **same volume as your 1 km radius sphere**, and place them so that their **crest** is at the same depth as the crest of your sphere (why?).
 1. For one of these, repeat 10b(iv) and 10c, adding the results to the same plot as before (work at symbolizing things sensibly!) and then assessing the impact on the deformation.

- iii. If time permits, do the same thing for a prolate ellipsoid.

- e. **Final Question:** limited GPS stations can be placed in the field. Considering what you have learned, if you had three GPS stations to place, where would you put them to maximize the potential to detect an ongoing uplift and differentiate between possible source geometries? You may want to compare the spherical reservoir signature for two pressures as you consider your answer to this question!

APPENDIX: Exporting COMSOL Graph Data for Use in Excel

- To Export the uplift data and bring them into Excel:
 - **RC Uplift** (Line Graph 1 with your data) and select **Add Plot Data to Export**
 - Notice that Plot 1 shows up now under Results > Export
 - Select Export > Plot 1 and examine the Plot window
 - In the *Settings* window, under **Output, Browse** to pick a filename and save location (you are exporting a text file), **and ensure that the Data format back in the Plot window is set to spreadsheet**
 - Click **Export** to save your data.
- Find and **Open** the text file you created.
 - At the top will be a bunch of info about COMSOL and then there will be columns entitled R (the x-axis value, but note the unit is not recorded, so good thing you know it is meters, right?) and Height (the y-axis value, again no unit).
 - Close your text file when done.
- To bring these into Excel, open Excel (or create/select a new worksheet if Excel is already open) and then:
 - **File > Import**, choose **Text file**, then click **Import**. Navigate to your file, and select **Get Data** to begin the import
 - Adjust the “**Start import at row**” value until R and Height are showing in the preview pane, but none of the other COMSOL info. Normally this means you’ll start your import at row **8**.
 - Set the file type to **Delimited**, then click **Next**
 - Make sure that **Space** is the **only delimiter selected**, and in the data preview window note that Excel thinks there are two columns of data, but offsets the headers because of the %-space before the R. We can edit this easily in Excel, and would want to anyway to ensure clarity of the information. So, let it remain as-is for now and we’ll fix it shortly.
 - There are other things we could set, but at this point select **Finish** and, in the blank sheet you have open, indicate **Existing Sheet at cell \$A\$1** (*or create a new one if necessary*), then click **OK**.
 - Finally, edit the header... column A might become, from the Plot established above, r (m) while column B’s header is entered as uplift (m)... what is needed here will depend upon what you exported.
 - Be sure to save your working Excel file if needed ☺