LaModel

Frequently Asked Questions (FAQs)

(Updated 1/18/08)

1. Consistent Units in the LaModel Input File.

I looked at the output file from LamPre (the input file to LaModel) and I noticed that several of the numbers that I entered, such as the layer thickness, have been changed from feet to inches?

The LaModel program (as most numerical programs) requires a "consistent" set of units in order to calculate appropriate results. For instance, in English units, LaModel uses inches (in) for distances and pound per square inch (psi) for pressure. (For metric units, LaModel uses meters (m) for distance and Pascals for pressure (Pa).) LamPre, the preprocessor for the LaModel program has always converted the units of certain parameters in order to derive a consistent set of units. For instance, if the layer thickness is input in feet, it is converted to inches in the LaModel ".inp" file.

With the introduction of LaModel version 2.0, the units entered in the preprocessor (LamPre) are carried through the calculation phase (LaModel) and into the post-processor (LamPlt). Therefore, if you entered distances in feet in LamPre, LamPlt should plot the "seam convergence" in feet, although the actual calculation was done in inches in LaModel.

2. The Rows are Numbered Differently between LamPre and LamPlt.

I noticed that in my Lamplt output file the row numbers are in the opposite direction than they were numbered in the grid created in Lampre. Did I really mess something up or is the Lamplt output row numbered from the bottom of the plot instead from the top as in Lampre? The column numbers appear the same.

You are correct. The rows in the LaModel output are numbered from the bottom up, normal Cartesian coordinates. In the LamPre grid editor the rows are numbered from the top down, normal Excel format. The software that we use for the grid editor in LamPre is modeled after an Excel spreadsheet and numbers the rows from the top down, and we are not able to change the ordering in the grid editor. However, in LaModel and LamPlt, the rows are number from the bottom up, as in a normal Cartesion coordinate system. This keeps the output consistent with other engineering software drafting such as AutoCAD.

3. LamPre Does Not Remember Yield Zone Properties

In the Material Wizard, I generated yielding coal properties, but when I re-open the file in LamPre, the program does not recognize that I have sets of yielding coal?

In the Material Wizard, I change the coal strength to generate the material properties, but when I load the input file back into LamPre, it still has the default coal strength?

Both of these problems stem from the same condition. The output file from LamPre (the input file to LaModel) contains the final material properties generated by the Material Wizard, but it does not contain any information on the specific parameters used in the Material Wizard to generate these material properties. Therefore, LamPre only knows about the yield zones, coal strength, gob width, gob stress, etc. while LamPre is running. Once LamPre is exited and re-started, all information about the yield zones and other Material Wizard parameters revert back to the system defaults.

As a work-around to the problem of remembering the yield zones, you can always re-generated the yielding coal properties when you come back into LamPre. The Material Wizard will just write over the old material properties with identical material properties, but now it will know about the sets of yield zones so that they can be applied in the Grid Editor.

4. The Material Editor Can be Used to Manually Input Material Properties.

Can I directly edit or input the material properties in LamPre without using the Material Wizards?

Yes, you can. The form is a little hidden, but you can directly edit (or input) the material properties of any material by going to the "Edit Data" pull-down menu and selecting "Material Models->Edit" to bring up the "In-Seam Material Models" editing form.

In this form, you can select the material number to edit using the slider in the upper right corner. Then you can select the material model using the radio buttons, and type in the appropriate material properties using the edit boxes. This form was used before the Material Wizards were developed.

5. Selecting More Than One Gob Material.

Can I select more than one material for gob? If yes, how?

Yes, you can select more than one material for gob. Using the Gob Material Wizard, it will just keep adding gob materials to the end of the defined materials. So, if you initially specify 12 materials and use the first 9 for coal, you can then generate 3 gob materials using different input parameters.

Or, you can directly edit the material properties of any material by going to the "Edit Data" pull-down menu and selecting "Material Models->Edit" to bring up the "In-Seam Material Models" editing form.

6. Multiple-Seams can have Large Edge Effects

I am having a problem with boundary conditions for the multi-seam model. T1 is a single seam model, and all looks fine, but with T2 there is some problem with boundary conditions. What am I doing wrong, or what do I need to do to improve it?

A good question.

You are doing nothing wrong. In LaModel, or any of these multi-seam boundary-element models, there are 2 types of edge effects. There is a single seam edge effect where a rigid boundary supports the roof around the edge of the model and limits the stress and convergence in this area. In LaModel, the symmetric boundary condition is a numerical technique to eliminate the rigid boundary and simulate a mirror image boundary without explicitly modeling the area outside the boundary of the model.

There is also a multiple-seam edge effect. In this case, the "other" seam projects stress to the "present" seam. However, around the edge of the grid in the present seam there is no stress effect coming from the area outside the grid in the other seam. Therefore, it is missing a part of its multiple-seam stress and typically shows an under stressed condition. (This condition can be extreme, where a gob area with high convergence is at the edge of the grid.) Unfortunately the single seam "symmetrical boundary" condition does not help with this multiple-seam edge effect.

The user needs to make the model big enough that the multiple-seam edge effect does not affect the area of interest in the middle of the model. As a rule-of-thumb and angle-of-draw of 30 degrees between the seams is generally conservative. This multiple-seam edge effect is something that user needs to understand and consider when evaluating the output from the model.

7. Out-of-Seam Extraction Ratios and Safety Factors in Lam2D

How does the "Out-Of-Seam Extraction Ratio" work in Lam2D?

My 2D Safety Factors do not look correct.

When using LaM2D, the pillar stress, displacement and SF would be determined as for an infinitely long pillar into the screen, the true 2-D scenario that is implied by the grid. When using an out-of-plane extraction ratio in LaM2D, we are able to somewhat adjust the displacements and stresses for the limited length of the pillars into the screen, and provide values that are more accurate for an infinite system of limited length pillars into the screen.

To determine the out-of-plane extraction ratio for an 80X80 ft pillar with 20 ft entries, just look at the out-of-plane dimensions. In this case, with an 80 foot wide pillar in the out-of-plane direction and a 20 ft wide entry, the correct extraction ratio to use would be 20%. The in-plane grid "inherently" takes care of the in-plane extraction ratio.

At the moment, when the out-of-plane extraction ratio is used, the pillar SF is a bit of a bastard calculation. We are using the higher stresses of a limited length pillar with the high strength of an infinite pillar. In this case the calculated SF would be higher than reality. Also, any pillar yielding is not truly reflected in the pillar stress when the out-of-plane extraction ratio is used.

8. I am having trouble getting my off-seam subsidence calculation to align with my mine grid?

In LaModel the mine grids and off-seam grids must be aligned with the internal/natural coordinate system. That is positive X is East and positive Y is North. You can generate a grid from a rotated mine in AutoCAD, but in LaModel it analyzes that grid as if it is aligned with the grid axes.

In LamPre, you specify the coordinates of the origin of your mine grid. For instance, the grid origin is often specified as simply (0,0) (but you can use true mine coordinates if you like) Then in the off-seam form, you specify the origin and size of the off-seam grid. These 2 origin specification must be on a consistent coordinate system. This is what aligns the underground mine grid with the surface grid for the calculation.

The actual coordinates of the surface point at which you are calculating the subsidence is a little confusing, unless you understand the system that is used. In LaModel, the surface grid is defined like any other grid in the program, such that the value for the grid element (in this case subsidence) is calculated at the CENTER of the grid element.

For instance, you defined your mine grid with an origin at (0,0). Now you want to calculate the surface subsidence on an east-west line 500 ft north of the Y origin and starting 200 feet west of the X origin. You decide to use 10 ft off-seam block sizes with 100 grid element is the X direction and 1 grid element in the Y direction. The first thought might be to define the off-seam origin as (-200, 500) (and at 0 depth). This would not be exactly correct. In this case, the center of the first (10 ft) grid element is at (-195, 505) and this is where the subsidence of that element would be calculated. The correct origin definition for a line of subsidence calculations 500 ft north of the Y origin and starting 200 feet west of the X origin with 10 ft elements, would be (-205, 495). The origin must be offset by half of the element width to get the element's centers at the locations you desire.

You can examine the coordinates of the off-seam plane by looking at the header lines in the ".off" file in a text editor (notepad, wordpad, etc.)

The first line contains: The number of steps, The total number of elements, the element width, the number of X elements, the number of Y elements, the X coordinate of the grid origin, the Y coordinate of the grid origin.

9. My topographic stress grid does not appear to produce the correct overburden stress, or it appears to be shifted.

My topographic stress has "low" stress values around the edges of the mine grid.

There are 2 problems that typically occur with the generating the topographic stress.

1) First, your topo grid needs to be physical bigger than your mine grid. It needs to extend over any area on the surface that may influence the mine grid. Generally, I use a conservative 45 degree angle "of draw" such that your mine overburden grid needs to extend outside of the mine grid on all sides by a distance equal to the depth. This error can cause low stresses around the edge of your mine grid.

In the Lake Lynn Mine example that is used for the topographic tutorial and is included on the distribution disk, the mine grid starts at (5350, 3000) and is 140 X 170 ten-foot elements which therefore extends to (6750, 4700). The greatest depthp in thi example is approsimately 500 ft. Therefore, the topographic grid in this example starts at (4850,2500) which is 500 ft down (south) and left (west) of the mine grid. This topo grid is 48 X 54 fifty-foot elements which extend the topo grid to (7250, 5200). Again, this is 500 ft past the mine grid on the top (north) and left (east).

2) The second problem occurs when the relative origin between the mine grid and the overburden grid is not right. If the overburden grid is "shifted" in relation to the mine grid then the overburden stresses are shifted also.

The problem often arises because of the many coordinate systems and unit conversions that occur when automatically generating the mine grid and topo grid. Some users have their AutoCAD set up in inches, some in ft, then some users input the Mine grid into LaModel in inches some in ft and then some use the "true" mine coordinates (which come from the map) and some put in the fictitious (0,0) for the mine coordinates. If the user is not very careful to use consistent units in this process, the origin of the topo grid can get erroneously offset from the mine grid.

In the Lake Lynn Mine example that is used for the topographic tutorial and is included on the distribution disk, there is and AutoCAD map of LakeLynn Mine. In this map, you can find that the origin of the mine grid is: (5350,3000) in ft. This should be input into LamPre in Ft as (5350, 3000).

If you check the LakeLynn.inp file (in a word editor) this has automatically been converted to inches as: (64200,36000).

From the LakeLynn input file:

```
LakeLynn Model with Auto Mine and Topo Grids
2.50000E-001, 3.00000E+006, 6.00000E+002,   1,   1
  1
  1, 3.00000E+006, 1.20000E+006, 0.00000E+000, 0.00000E+000,
0.00000E+000
1.20000E+002, 140, 170, 9.16667E-002
6.42000E+004, 3.60000E+004, -6.00000E+003, 3.00000E+002
1.65000E+000, 1.00000E-005,    5000,   1,   1,   0,   0,   1,   1,   0
  0,   1,   1,   1
....
```

In the AutoCAD map of LakeLynn, you can also find that the origin of the topo grid is: (4850, 2500) in ft. 500 ft down and left of the mine grid. This should be automatically read into the LakeLynn.top file in inches as: (58200, 30000).

From LakeLynn.top file:

```
1,2592,600.000000,48,54,58200.000000,30000.000000,3802.409470
1,Overburden Depth
1,Inch
...
```

If you are having trouble with your topographic grid, check the relative coordinates of the mine and topographic grid to make sure they are correct.

9. What is the datum elevation in the topographic input file?

The datum elevation in the topo file is the average value of the overburden in the topo file. More importantly, in a numerical sense, it is the datum from which the "Boussinesg" type stress calculation is performed (using the laminated formulas). From this elevation, the overburden is added (or subtracted) as a surcharge on the datum and the resultant stress is projected down to the seam to get the overburden stress on the seam.

Where to put this datum is a very good question that has not been studied much to my knowledge. If one would put the datum directly at the seam elevation, then the stresses on the seam would be exactly equal to the depth of overburden directly over that point. As the datum is moved away from the seam, the overburden stresses would smooth towards the average overburden stress. We know that some smoothing goes on in reality, but the exact amount is not known. As a reasonable first cut, I am using the average overburden depth for a moderate amount of smoothing.

The datum elevation in the topo file has nothing to do with the overburden depth input into LamPre. Once you say you are using a topographic input file, LaModel uses the topo file as the overburden for seam 1, and then just uses the difference in depths between the seams as input in LamPre to determine the new depth to recalculate overburden stresses for subsequent seams.

10. Can I use LaModel for predicting surface subsidence?

Should I use LaModel for predicting surface subsidence?

LaModel does have the capability to predict surface subsidence built into the model, but, for traditional surface subsidence predictions, there are many other models specifically designed for this that are easier to use and have much more field experience and are therefore more accurately calibrated. I do not recommend using LaModel for surface subsidence prediction, unless you have unusual circumstances. LaModel is primarily designed for stress redistribution calculation in single and multiple-seam situations

Every year or so, somebody does end up using LaModel for predicting subsidence. They may be too cheap to buy a more traditional subsidence prediction program, or have an in-seam situation that is not handled by the traditional program. Somebody is using it for a multiple-seam subsidence prediction at this time.

A little history. Originally when I programmed Salamon's laminated model, he was using it for subsidence prediction quite well. The influence function that comes from the laminated model formulation has the same mathematical form as the influence function that has been empirically used for subsidence prediction for many years. With the laminated model, the parameters come from the properties of the overburden, while empirically, the influence function parameters are back calculated from experience. I thought that here is our chance to have a subsidence prediction program where the parameters come from first-principals, measurable quantities. You would not need to calibrate the subsidence for each site, you could just use site properties. Also, the same program that calculates your stress and displacements underground could calculate your surface subsidence.

After the LaModel was developed and I gained some experience with it, I found you could predict surface subsidence with reasonable accuracy (after calibration) like most other subsidence prediction programs. It was a bit more cumbersome to use, because instead of a simple subsidence factor you needed to calibrate your gob properties, and instead of an angle of draw, you need to work with lamination thickness and modulus. Also, because it does much more calculations, it takes much longer to run. On the plus side, it does naturally account for multiple-seam subsidence, pillar failure/gateroad subsidence, etc.

Originally, I had hoped that one program would give good underground stresses and displacements, and good subsidence predictions. As I found out, you can calculate good underground stresses or good subsidence, but the parameters are quite different for each. Essentially, to get good subsidence prediction, you have to use very flexible overburden (thin laminations, low modulus) which gives very high, short abutment stresses, etc. For the parameters that give you good stresses (stiffer overburden), your subsidence prediction is quite low.

To calculate subsidence using LaModel, you have to use an "off-seam" grid. This option is a check box in the first screen. Once this is checked, you will get another form for defining the off-seam plane. In this form, you essentially create a grid of elements at a plane off of the seam (in the case of subsidence, at the surface). LaModel will then calculate the displacement at the center of each of these off-seam elements. LaModel will store the calculation results in a *.off files. This can be read by the post processor or imported in to EXCEL. To get realistic subsidence prediction, you need to use the "Free-Surface Effects" Option on the last form (which makes the computer run much longer) or you can double the non-free surface calculated subsidence as a first approximation. To calibrate the subsidence, you should use the gob final modulus to stiffen or soften the gob as necessary and us the lamination thickness to soften or stiffen the overburden as necessary.

Keith