

When we first configured the model to do the CAPTEX release number two, we made what appeared to be at the time, some arbitrary decisions. There is an option within the graphical user interface to test and optimize the HYSPLIT inputs and provide suggestions for changes.

To start with this test, let's go back to the original configuration that we used when we set up the CAPTEX experiment. Let's retrieve the CONTROL file, and the name list file, and this was the simulation using just the North American Reanalysis data. Let's go back and change the particle number, this is menu number 4, from the 50,000, which at the time I said was sufficient for this, to what the default would be if you had not set this number at all. The default when HYSPLIT is run would be 2500 particles that are released over each emission cycle. Now we don't actually have to do this simulation, we already know that the results would be quite noisy; 2500 particles is not sufficient. So under the special runs tab, there is an option to test the inputs. And this invokes a special version of HYSPLIT that does not really do any calculations, except it goes through the entire meteorological data file and it determines whether or not some of the settings are sufficient given the data that we have.

It's really quite simple, just three steps, the first is to analyze the file, the CONTROL file, and the name list file. And in this case the test was completed and there were no warning or error messages. If there is some kind of fatal error in the configuration, that would appear here. Now

just because this completed without any warning messages does not mean that the configuration is optimal.

We should now look at the various buttons viewing the output files as result of this test. Looking at the warning message you can see there are three messages. The default method for HYSPLIT, to determine the strength of the turbulence or mixing, is to use the heat and momentum fluxes that are available in many of the meteorological data files. In this case the fluxes were not found and the model will use the wind and temperature profiles to determine stability. The thing it noticed was that the concentration grid was set to terminate after the end of the simulation and 68 hours from the start of the simulation would've been the 28th at hour 13. And the last thing that was noted, was that the particle release rate of 2500, was insufficient by a factor of six, and it's suggesting to use a rate of at least 17,000. So this confirms that the original suggestion of using 50,000 is approximately twice this number, so it would've been sufficient.

Now the test program determines this by looking at the magnitude, or in making an estimate of how much mixing occurs along the path, and it also looks at the size of the concentration grid cells in both horizontal and vertical, and makes a determination of how many particles are needed, based on the number of particles it estimates would be sampled at any one time using the existing grid cell resolution.

The other message file can be examined here. This is similar to the MESSAGE file that you would get when you run the model, except that it only follows one particle in this test calculation. But the main point is it also makes a recommendation for the CONTROL file and the setup file. And it opens up two windows, the original CONTROL file and the suggested modified CONTROL file. And you can see that the only change that was made here was the end time for the sampling and was changed to hour 13, which corresponds to 68 hours after the start time. The 68 hours was not really an error, it was done on purpose to go just a little bit beyond the sampling. If you were to go back and look at the measured data file, you would see that the last sampling period that was on, would've ended on the 28th at 9 UTC. By giving it a little bit of an extra three hours sampling window, that would take it to 12 UTC, and going one hour beyond was the normal termination time at 13. So, in fact changing this to 13 really would have no effect on anything or any of the calculations, because the sampling essentially ended at 9 UTC. Of course the test program does not know this.

And the last thing is the name list file, and there are two of these, the modified one and the original one. And you can see the modified one is much longer than the original one. And the reason for this is that the original one is created by the graphical user interface and not all name list options are available to be changed through the graphical user interface. Whereas, the modified setup file was written by the test evaluation program, and it has of course all the name list variables. And you can see here

the recommendation to change the particle release rate from 2500 to 17,000. It also change the stability method, here's the previous one too, so that you do not get the error message again about that the fluxes are not available and that is going to use temperature, temperature and wind profiles, to determine the mixing. This change you did not really have to make because HYSPLIT will do this automatically. This is a, when you configure the model, certain options during the model initialization phase are evaluated, and changes are made accordingly, and these changes are written to the MESSAGE file.

And the last step, if you want to go with the recommendation of the program, is you would click the copy command and it would copy the modified files back into the original names of CONTROL and setup.cfg and then you can run the model with those changed files.

And this concludes the discussion of how to evaluate the model input files.