In section four we will discuss trajectory calculations in more detail. We will start by doing a single trajectory that corresponds with the start of the tracer release for CAPTEX experiment number two. Start by opening the graphical user interface under the trajectory tab and select set up run.

Here we have several options that need to be entered. First enter the starting time of the trajectory. If you don't remember, the experiment started in 1983, September 25th, at 1700 UTC. So that's when we will start the trajectory calculation. We're going to need to select the number of locations in which we will start. In this case we will only select one. You can start multiple trajectories at the same time from different locations. Here we can enter the starting location or you can select from a list. It happens to be, we already added Dayton to the list. So if I were to open this and scroll down to the US, here, you'll see Dayton, Ohio. I can select that, and that is the location of the airport at Dayton, where the experiment was conducted. And we’re going to have a default starting height of 10 m above ground level. By the way, this file, when you select list, is just a file in your directory, in your working directory, called plants.txt. You can enter in this file locations that are of interest to you, so you don't necessarily have to use this file.

Continuing on, we know that the sampling for the tracer release was conducted over a 68 hour period. So will follow this trajectory for 68 hours in a forward direction. We can come compute trajectories forward or backward.

The top of the model, this is a default. This is just the height at which we stop processing the input meteorological data. Some data files have data well high up into the stratosphere. We're doing calculations here mostly in the boundary layer, so it does save some computer time by skipping over the upper levels of the atmosphere.

We have options to use different vertical motion methods. The recommended default is to always use the vertical motion field that is provided with the meteorological data file. There are other options which we will be discussing in more detail in a later section. As an example, you could be doing an isobaric calculation, which means that the trajectory is forced to stay on a constant pressure surface. Or you could be doing a constant density trajectory, which means that the trajectory will stay on the same density level. As an example you might want to use a constant density calculation if you are following a balloon, which will stay on a constant density surface.

And the last, well not the last but next to last, thing is that we want to enter the name of the output file to which the trajectory end points will be written. In this case, we're defaulting to a filename called tdump, short for trajectory dump. But you can obviously name this anything you want.

And now we need to select the meteorological data file. So unless you want to use this file, which is the sample data file that is provided, clear this, and add file.   Clicking the add button will add multiple files.   You can use more than one meteorological data file for a calculation.

Now we know that the data for CAPTEX2 are contained in the tutorial directory.   So we going to tutorial/captex, and most of our calculations will use the North American Regional Reanalysis, so we select that file here, and that completes the entries that we need to do a single trajectory.

At this point you click on save and what happens is in the hysplit working directory, when you clicked on save, it wrote a file called default_traj, which if I were to open that up, is really just a summary of the information that we just entered: the starting time, the number of starting locations, the trajectory starting location, the duration, the vertical motion method, the top of the model, that there is one meteorological data file located in the tutorial/captex directory, and the name of the file followed by the directory of the output and the name of the output file.

Now whenever you open the GUI, the graphical user interface, it reads these default files, default_traj, default_conc, and so on, and enters that information into the GUI variables. So that when you start up again, you'll have the last configuration that you had.

Now the next, this is a three step process to run the model.   Now when you click on the run model tab it essentially executes the trajectory executable model.   But before it executes that, let’s do that calculation. What happens is the trajectory executable reads the file name of CONTROL and the control file is essentially identical to that file default_traj. So before you click on run, what happens is the script in the graphical user interface copies default_traj into CONTROL and then it executes the executable for trajectories which is called hyts_std, and then that file creates the executable, rather, creates the output file which is called tdump, which if I were to look at, contains the latitude/longitude points of the trajectory. The default is to write out the trajectory end points every hour. So is this would be the first hour, then the second hour, and so on. You can select more frequent intervals for output, but we will discuss that in later section.

Now the third step, the third and last step, is to display the trajectory. So we will go to that tab, select display, and you can see here that this information is prefilled based upon the entries that you made in the setup menu. So it's going to read the endpoints file tdump, it's going to create a Postscript file called trajplot. It's going to use this map background file which is the default. We will discuss other map background options in a later section. The projection will be created automatically. We will not create any graphical GIS output and you can, I'm not going to discuss here, but you can force the map, you can force the size of the map, and you can label the time intervals along the trajectory more frequently. And you can select different vertical coordinates. The default in this case is pressure. Zoom controls how much whitespace is around the trajectory.

If I execute display, you should have this result. Now this result shows the trajectory starting at 10 m going along in the US Canadian border. If you recall from the animation that we did, that I showed you early on from the CAPTEX tracer measurements that were made, most of the tracer measurements were made in New York and Pennsylvania. So virtually all of the tracer was to the south of this trajectory. So you can see in this very simple example a single trajectory is not going to be representative of a more complex dispersion scenario.

The last thing I would like you to do, as we know that this simple trajectory worked well, so let's go back to the setup menu, and save this configuration. We're going to give a special name, and we will just call it traj_base, for the base case, and it is a control file, and we will give it a suffix of txt. And you can see here, now in the working directory, we've created this traj_base_control.txt, which we will use later on to simplify entering data.

And that concludes the single trajectory calculation.