**Wind Tunnel Sandbox**

Just is just a sandbox for experimenting in a wind tunnel. Select an e

**Experiment 1 – Lift and Drag**

See if you can create a wing with a lift to drag of above 2 (hint, the angle of attack matters).

**Experiment1**

In this first experiment you will see how the speed of a fluid effects the drag force on an object at low speeds. [wait shouldn’t this be the other way round?]

[In wind tunnel simulations we typically move the wind and have the object stationary. It’s hard to test a car travelling at 100km/h, but we can test a stationary car with 100kh/h wind. Force due to air resistance is only interested in the RELATIVE velocity, so this handy trick works super well.]

You mission is to play with the fluid speed slider on the right. As you do you can hit the ‘plot’ button which will plot the results on the graph below.

Try this at a few different speeds.

What happened? Is the drag force really proportion to the velocity? Can you find any scenario where this isn’t true? How about other object types (you can select them from the right). How do they change?

When you think you’re done see if you can calculate the “b” constant in “R=-bv” and enter it here

[I’d rather not]

[No worries, just click here to move onto Experiment 2

This kind of flow is called LAMINAR flow. Laminar flow is wonderfully easy to model, however the real world isn’t always so nice to us. Time to break things. Let’s move onto Experiment-2

**Experiment2**

Now that we’ve seen laminar flow in action we’re going to try something a little more caitoc.

The experiment below is just like the previous one, however now we’ve turned up the speed.

Like last time, try different speeds and plot the drag forces at regular intervals. This time round you might need to hit the reset button after every experiment to carm things down before the next run.

So what’s happening here?

Did you notice those strange circles behind the object. These are called “vortexes”. They’re produced in non laminar flow. Now I’m not super clear on the physics, but as far as I understand it the complex wake generated by the object under these circumstance induces a lot more drag. And that drag is ruffly proprional to V^2 [ruffly?]

[Yeah, at these speeds things are complicated. Look in your results, it’s not a perfect parabolic curve. Also things get even worse at trans and super sonic speeds. Fluids are a complex beast. Often our only real method to tame them is simulations like this or wind tunnels.]

Some things you might want to try:

Try seeting the speed to x, then after turbulence sets in decrease it to y. Make a note of the drag

Now try hitting the reset button and running the experiment again with the same y (but don’t go to x first). What is the drag now? Why would that change?

What shape has the maximum vecloity before turpulance. Why would this be? Can you think of any other shapes that would have an even higher pre turbulence speed? [later I’ll let people draw their own].