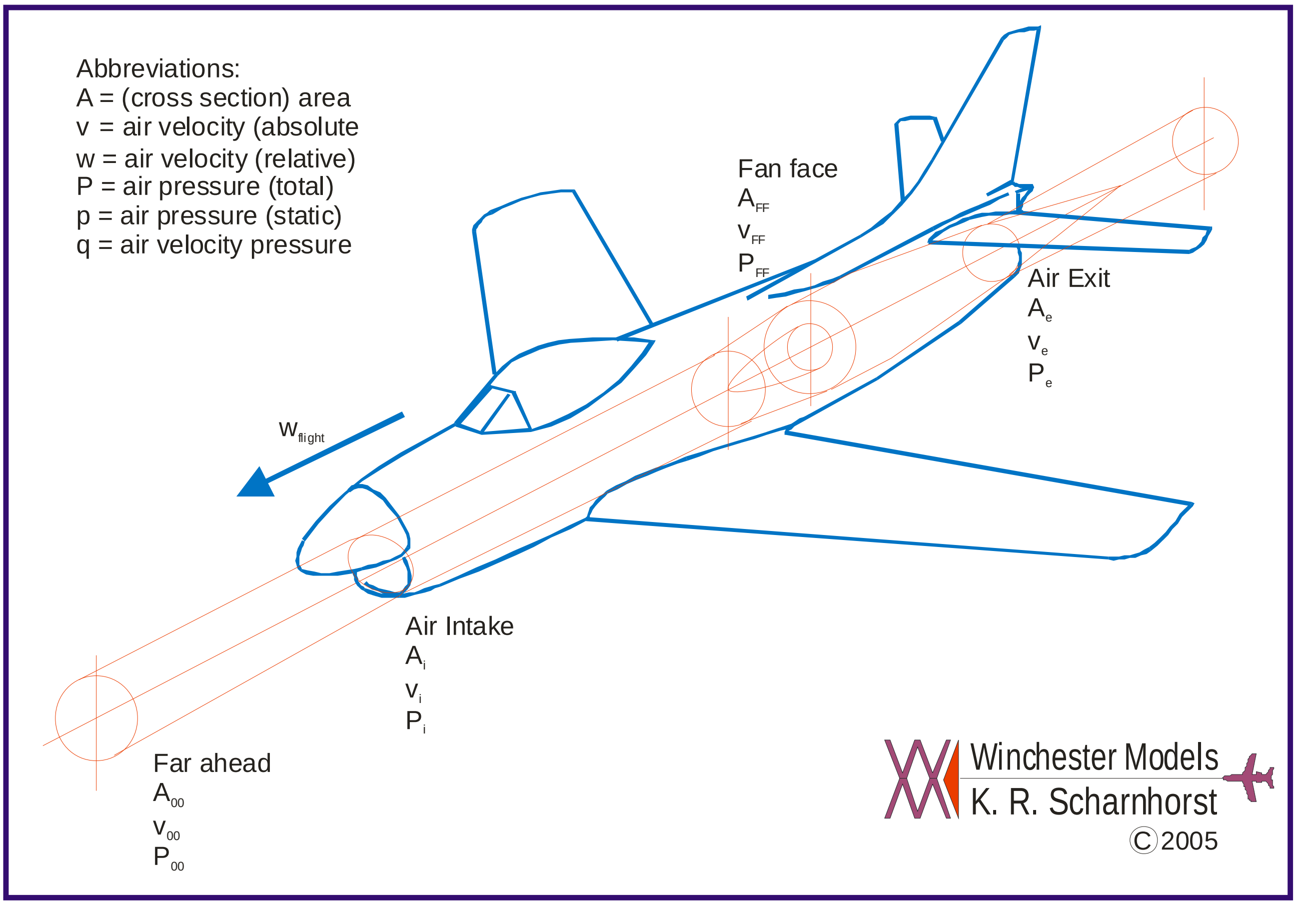
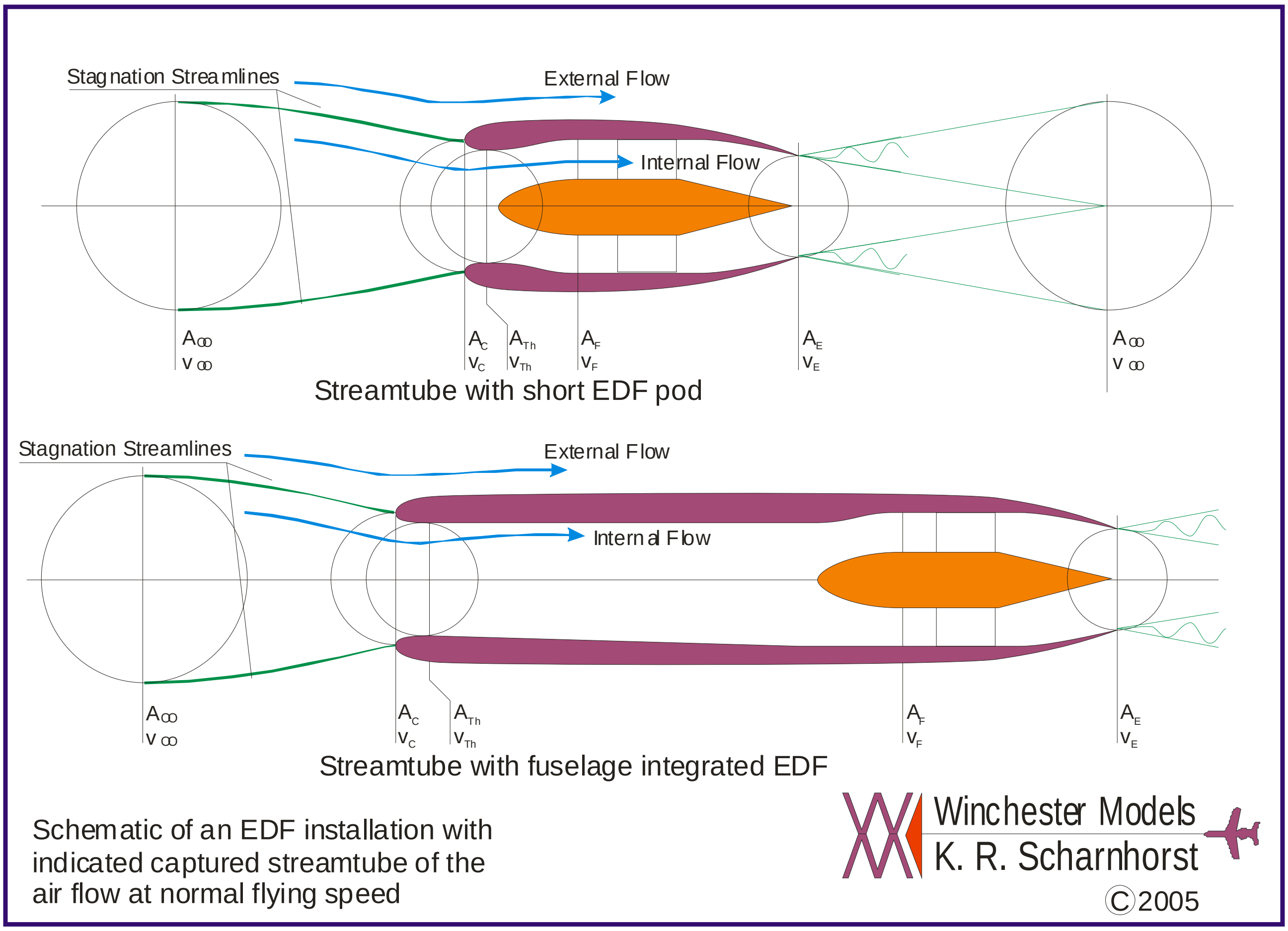


An EDF installation in a model aeroplane can be a daunting task. In most cases the manufacturers have not supplied much information and by the looks of it, the info gained from various other sources is not very consistent. There are many aspects to EDFs which have to be observed to achieve a successful installation, one of the most important one is the ducting to and from the fan.
To help in visualising what is actually happening it has been proven useful to introduce the concept of the aerodynamic duct. Principally this is just an extension of the actual duct as indicated in the picture above. One can see that there is a “virtual duct” added to the front, which is thought to behave like a real one and a similar is added to the rear.
The internal details of the duct are added as far as necessary, which is mainly the fan face, the only concession to reality needed in this context for the possibility to add or subtract energy.
For our understanding of the working of the EDF in a model plane it is interesting to know what actually determines the amount of air flowing through the duct. Is it the air intake, the fan face area or the exit area? The usual answers show that the understanding of the underlying principles are widely misunderstood.
Considering the above picture and Bernoulli’s theorem, it is evident that since the air flow volume or mass through the duct has a constant value over the length, the velocities at the different stations must be inverse proportional to the areas. Let’s assume that the plane moves through the air with a velocity w and the air exit velocity is ve, i.e. w plus Δv, which is necessary to produce the required thrust for that velocity, then we can see that the local axial velocities at any station are only dependent on the cross section area of the exit (as long as we do not try to brake through the sound barrier).
If for example we have made the intake area smaller than the exit area (because of the scale model which we have chosen also has a smaller intake area) then the air velocity at the intake must be higher than the exit velocity! And this unfortunate relation will persist at any flying speed and any power setting. It should be said though that the above example is only chosen for demonstration purposes, one should avoid too small intake areas for reasons which will be explained later on in detail.

The air velocity at the station F_{00} naturally is always equal to the flying velocity w, which means that the aerodynamic duct cross section area well ahead of the plane is larger than at the air intake (for the above described condition). This can change when we land the plane with no or only little power, in which case the air velocity through the aerodynamic duct can be less or the same than the flying speed.
Under normal (model) circumstances one can usually arrange to have a larger air intake area than the exit. The example above was only used to visualise and highlight the effects and to show which of the cross section areas determines the air mass flow through the aerodynamic and the real duct: only and alone the exit area! The air velocity at all other cross sections has to adopt to this, even if it is not desirable as above. I will try to explain in more detail in a following paragraph how to deal with abnormal circumstances. So much should be said here already, that a (too) small intake area causes excessive duct drag and should be avoided if at all possible.

Now we will have a look at more usual layouts of ducting and how to optimise them for best efficiency and thrust at all flying speeds.



Schematic of an EDF installation with indicated captured streamtube of the air flow at normal flying speed

In the top of the picture we see a relatively short pod installation, which we know from the previous explanations. The aerodynamic duct here is again evidently defined by the area A_{00}, the stagnation stream tube, the pod itself and the exit flow up to the downstream A_{00}. The exit flow at A_E has the highest velocity anywhere in the aerodynamic duct in this case, where we assume that the intake area A_C , the so called catchment or highlight area, is equal or larger than A_{Th} (the throat area), which again is larger than the exit and equal to A_F. I have chosen these dimensions with care, because this situation is widely used in reality as well. We can write:

$$A_{00}>(=)A_C > A_{TH} = A_F > A_E$$

This is the arrangement which offers the least internal resistance of all possible combinations, because we have a continuously accelerating air flow up to the exit and practically no ducting to speak off. This is also the arrangement which is used for testing EDFs (or should be). It is very

difficult to accurately calculate the influence of the little ducting which still is there on the actually measured thrust, but a reasonable figure seems to be around 3% - 5% maximum, which I have calculated (deduced) on various occasions. One can therefore use this arrangement as a benchmarking baseline for comparison with more elaborate and complicated installation.
I will therefore use it to show and explain the influence of other area ratios in a generalized way, by using this as the unit 1 value. Additional ducting and more complicated duct arrangements like bifurcated intakes will be dealt afterwards.