ANSYS Fluent Vs. Autodesk Flow Design
By: Jacob Heathorn

Numerical Architecture:

- ANSYS allows the user to customize the solver methods and models to vary the speed and accuracy in finding the solution.
- Autodesk Flow Design does not allow the user to custom define the solver methods and models. The solver is preset to a general method and it solves extremely quickly. After importing a geometry, Flow Design automatically begins to simulate flow over the geometry in a virtual wind tunnel. The user can change the orientation of the geometry and the size of the wind tunnel, but cannot view the solver methods. This allows for a very quick and easy process, but is not as accurate as ANSYS.

Strengths/Weaknesses

Strengths:
• Extremely easy to use and Fast – used for ‘early conceptual understanding’ [1]
• Users can interactively make changes and see the affects on the system very quickly [2]
• Very ‘Geometry Tolerant’ and can accept simple and complex geometries very easily[2]

Weaknesses:
• Autodesk Flow Design is simple and easy to use, but it cannot handle supersonic flow, heat transfer, or detailed boundary layer customizations.[2]
• Flow Design can only analyze air flow and does not include any other fluids[2]


Availability/Cost and User Base

- Autodesk Flow Design is available to everyone:
  - a. $35/ month for individual licenses
    [link](http://www.autodesk.com/store/flow-design?licenseType=desktopSub&support=basic&term=monthly)
  - b. Free for college students not using the software for a profit
    [link](http://www.autodesk.com/education/free-software/flow-design)
  - c. Companies require a company license
    [link](http://www.autodesk.com/company/legal-notices-trademarks/code-of-business-conduct)

- Vehicle designers use it to analyse the aerodynamics and architectural designers use it to analyze wind flow over buildings
  [link](http://www.3dcadworld.com/autodesk-goes-flow-releases-new-simulation-tool/)

- The user base for Flow Design is more oriented towards individuals because it is cheap to obtain a single license as compared to ANSYS and it is much simpler to use
  [link](http://www.worldcadaccess.com/blog/2014/08/autodesk-flow-design-the-easiest-to-use-cfd-program-ever.html)
Recommendations

- My recommendations depend on the speed, accuracy, and type of fluids required for the simulation. If the user needs to view the affects of wind flow over a geometry very quickly in the prototyping stage of development of a product, then Autodesk Flow Design is the perfect software to use. However, if the user requires a more accurate or particular solver to analyze air or any other fluid flow, then ANSYS is the preferred software. To prove the accuracy of an analysis, it is important to state the meshing resolution, boundary conditions, and solver types and resolution. Flow Design does not offer this type of proof because the user must trust the pre-selected methods and conditions and can only change the air speed. ANSYS is much more dependable in a wider range of applications that require this level of accuracy, but it can be very slow. In a small variety of applications, Autodesk Flow Design can be a more useful software simply because of the speed and ease of use.