

“We’ve got a digital flow bench, we don’t need no stinkin’ flow balls!”

BY DAVID C. “WOODY” WOODRUFF

Now before you get your knickers in a knot – you can keep your flow balls - for now! Just give me a chance to present another tool for your tool box and enlightenment!

Whenever I ask what you measure on a flow bench the answer is invariably CFM. What you really measure is differential pressure and you calculate CFM. Same with the pitot tubes! When you use flow balls you disrupt a local pressure field and get a new CFM number. The flaw with this is that all the things and strings we put in the port alter the very thing we are trying to measure. Not useless mind you, but error prone and very sensitive to technique! (And yes, some of you are very good at it!)

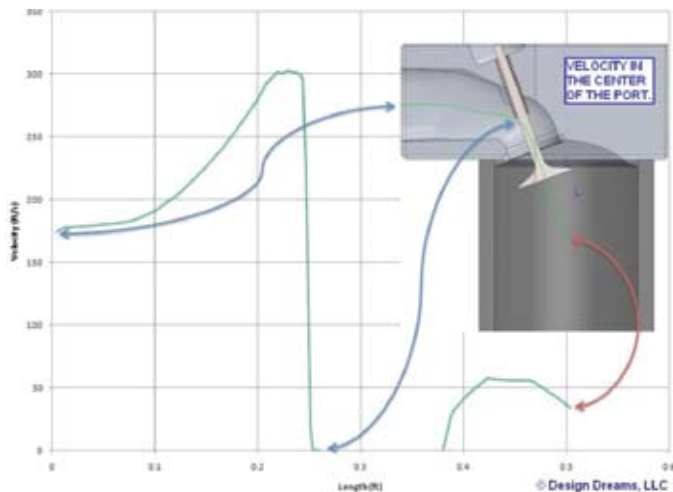
CFD or Computational Fluid Dynamics is now available as general purpose code and is usable in a variety of applications as well as becoming much more affordable. My current CFD selection is FlowSimulation which is an add-on for SolidWorks. www.solidworks.com

Let’s be clear, although the software is user friendly, CFD is not easy, but neither is cylinder head porting. There will be a learning curve and a transition period but I think much shorter than mastering the die grinder. Think of it as moving from checkers to chess. Everyone knows the board moves but there are only a few grand masters. Now before you panic, look how much of our lives are already digital. If I can get a computer to help me play the game better then why not! Instead of sculpting in the metal do it virtually then send the results quicker and more accurately to your CNC.

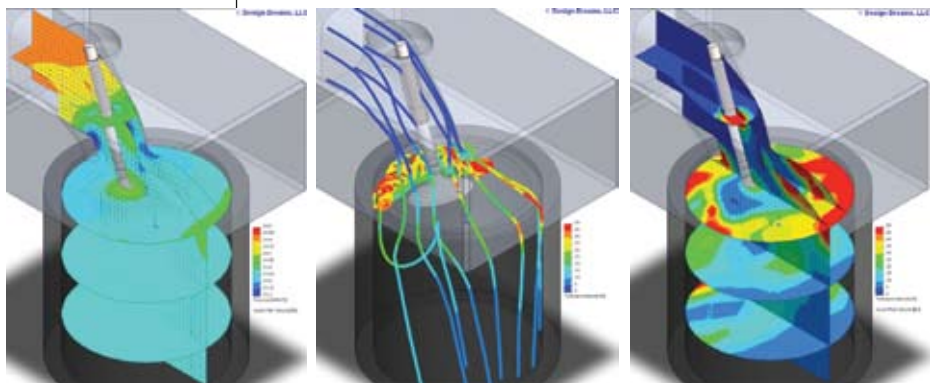
Since you already have your own ideas and tools for porting then let’s just look at some results from a digital flow bench on a generic, virtual intake port @ 28”H2O and see what you think!



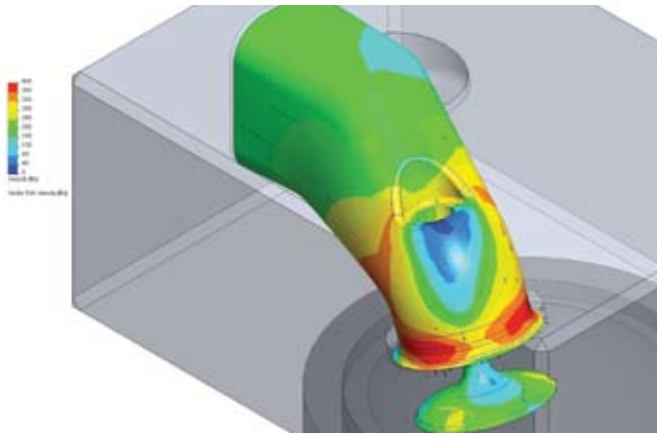
Port volume can be 3D printed instead of making rubber molds but the virtual volume can be solved for average pressure, velocity and other parameters of interest with no slicing, dicing or tracing on graph paper.



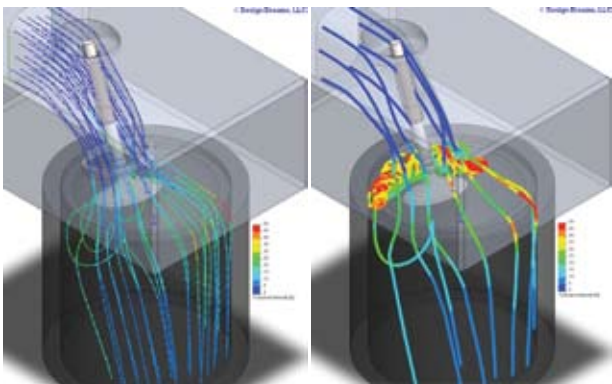
XY plots are simple plots down the center of the port in this case. These can also be plotted along a flow trajectory path.



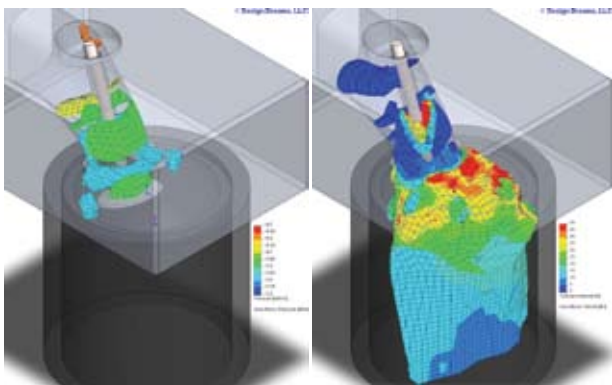
Cut plots are 2D slices typically with velocity & pressure plots with velocity vectors. How many pitot readings to get these plots? (Time = \$\$\$) Can you even see all these little strings (velocity vector arrows)? How would you determine the turbulence intensity or its area of influence?



Surface plots can be shown with velocity or pressure plots with vectors. Can you even plot these velocities at or near the surfaces (boundary layer)? How about on the “dark” side of the valve?



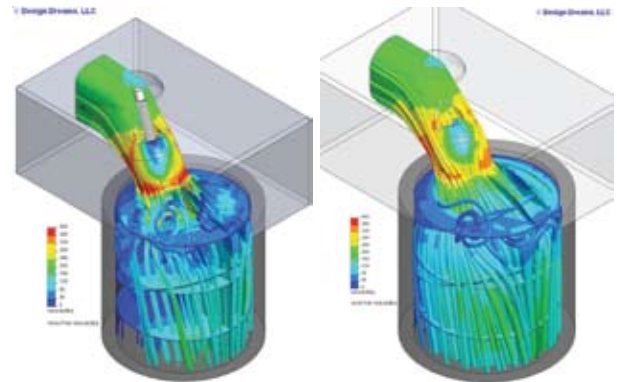
Flow trajectories are virtual smoke traces shown here first from the inlet – no collection of inlet radii or cylinder adapters required here (save \$\$\$)! But can you blow smoke to and from the valve seat only – it’s very different, eh? Can you plot the pressure or velocity along the smoke path? Now don’t let any smoke blow up your shop coat!



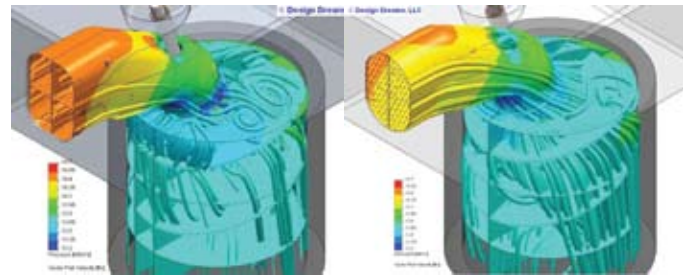
ISO surfaces are specific pressure or velocity 3D “bubbles” that can have other parameters plotted on them. And yes, these can keep you up at night wondering what is really going on!

Knowing what you know and seeing the possibilities are you ready now to play in the virtual world? A good place to start would be to have studies done on two known models – one bad and one good – then see what the CFD reveals about the two. Don’t get too hung up on absolute numerical values at first, the real power of simulation is in the visualization of the interactions! (BTW: This applies to FEA also!) As you develop your new methods and techniques the virtual model can be

tweaked to better agree with the empirical results! You can easily drown in data from CFD so take it in simple, basic steps at first and then drill down to the areas of opportunity as you proceed. Remember the 80/20 rule – 80% of the solutions come from fixing 20% of the problem. I like to say KISS – Keep It (a) Simple Simulation! And remember the words of G.E. Box, “All models are wrong, but some are useful!”



Now if we only change the port inlet from rectangular to oval here’s what happens! The intake inlet area goes down ~10%, CFM goes up ~1% but the inlet velocity goes up ~12%. However, if we look at the results of volumetric parameters then port volume goes down ~5% and the velocity only goes up ~6%. (This is the resultant velocity in all directions!) But the dead zone behind the valve guide & boss is much improved and the flow directions over the short side radius and into the cylinder change a lot, too! I would venture a guess that some of this agrees with current observations but some of it will get you thinking along some new lines aka flow trajectories!



Now what about that exhaust port at low lift, near sonic velocity and 1200°F? Will your flow balls really tell you what you need to know there?

Remember that some will be first to adopt and some will be last but you will both be easily identified by your finishing positions at the track! ■



David C. “Woody” Woodruff, CMfgE, is president of Design Dreams, LLC. For more information, call (513) 403-3165, e-mail Design_Dreams@cinci.rr.com — or stop and see him at the IMIS Show in Indy, Booth #2441.

