CFD Modeling of Wind Tunnel Flow over a Rotating Cylinder

John Middendorf
Student Number 3049731

Computation Fluid Dynamics
Professors Tracie Barber/Eddie Leonardi
May 30, 2003

I hereby certify that this is my own and original work, and that all references have been cited.

signed

John Middendorf

Flow over a Rotating Cylinder

Background

Viscous flow over a rotating cylinder is an interesting problem in fluid dynamics. The rotation of a no-slip cylinder wall creates a pressure gradient which in turn creates lift. The pressure gradient can be explained simply by Bernoulli's principle, in which pressure and velocity are inversely proportional. The phenomena of a rotating cylinder's lift is know as the Magnus effect, named after a 19th century German engineer, and is related to the circulation around an object in a flow field. Rayleigh studied the lift of a rotating cylinder for an inviscid ("frictionless") fluid, and related lift to the circulation of a rotating cylinder by the following formula:

$L = \rho U \Gamma$ in which the circulation, Γ, is given by:

$\Gamma = \oint v \bullet dl = v(2\pi R) = \omega R(2\pi R)$, therefore,

$$L = \rho U(2\pi\omega R^2)$$

The relationship between lift and circulation is known as the Kutta Joukowsky relationship and applies to all shapes, particularly to the aerodynamic shapes such as an airplane wing.

In a viscous fluid, like air, the cylinder is subjected to both pressure and viscous forces, and the explanation is more complex. Studies (Smith, 1979) indicate that the circulation does not result from the common explanation of the air set into an opposing rotation by the friction of a no-slip wall, as this only occurs in a very thin boundary layer next to the surface. But this motion of the fluid in the boundary layer does affect the manner in which the flow separates from the cylinder. Boundary layer separation is moved back on the side of the cylinder that is moving with the fluid, and is moved forward on the side opposing the main stream. The wake then shifts to the side moving against the main stream causing the flow to be deflected on that side, and the resulting change in free stream flow creates a force on the spinning cylinder.

Problem Statement and Parameters.

A cylinder 0.66m in diameter (R=0.33) is modeled as a wind tunnel test. Two parameters for the wall boundaries were modeled in the different grids created, one with walls at 6R above and below the wheel boundaries, giving a total height of the wind tunnel as 4.62 meters, and the other (for some of the later CFD models) using a height of 6R from the center, giving a total height of the wind tunnel at 4 meters. Using Fluent, the problem was analyzed in two dimensions, which in effect models an infinite width cylinder. The cylinder is rotating at 168 radians/sec, and the windspeed flow in the wind tunnel is 200 km/hr (55.6m/s). There are two Reynolds Numbers of interest: one is the Re of the cylinder, which is:

$$\mathrm{Re}_{cyl} = \frac{\rho UD}{\mu} = \frac{1.225 kg/m^3 \times 55.6 m/s \times 0.66 m}{17.9 \times 10^{-6} Ns/m^2} = 2.51 \times 10^6$$

The other Reynolds Number of interest, used for turbulence parameters is the free stream Reynolds number, given by the characteristic length of the wind tunnel (using the 4 meter wind tunnel height parameter):

$$\mathrm{Re}_{freestream} = \frac{\rho UD}{\mu} = \frac{1.225 Kg/m^3 \times 55.6 m/s \times 4m}{17.9 \times 10^{-6} Ns/m^2} = 1.52 \times 10^7$$

Boundary Conditions of Wind Tunnel
It is important to accurately represent in the computational model the free stream flow in the wind tunnel by the choice of an appropriate length. Since CFD is driven by boundary conditions, where the property values of all the elements inside the BCs are essentially "filled in" to attain an equilibrium solution, the placement of the inlet and outflow affects the solution. The inlet boundary is set far enough in front of the cylinder so as to represent fully developed flow by the time the flow reaches the cylinder (since the inlet will be set at a initial velocity, constant for all values). The modeled developed flow when it reaches the cylinder will vary from the inlet boundary conditions because the no-slip wind tunnel walls will slow the velocity near the wall boundaries, affecting also the modeled turbulence as the flow approaches the cylinder. Ten times the diameter of the circle was chosen as a minimum inlet distance in the CFD models.

The outlet boundaries also need to be far away from the point of interest, but for additional reasons related to the assumptions of the solution process. CFD models have two main types of free flow outlet boundary settings: outflow, and pressure outlet (generally constant pressure). Pressure outlet boundaries are used to define the static pressure at the outlet, and are recommended with compressible flow problems. Outflow boundary conditions are used when details of the flow velocity and pressure are unknown prior to solution of the flow problem, and assume a zero gradient for all variables except pressure. Outflow BCs are only appropriate when exit flow is in a fully developed condition. For this to occur after a turbulent wake, at least ten times the diameter of the object causing the wake is recommended (Versteeg and Malalasekera, 1995). The outflow BC was used, and values of at least twenty times the diameter of the circle was chosen for the outlet boundaries.

The initial inlet flow of the wind tunnel is not known completely, but it was assumed that there pre-exists a specific amount of turbulence in the flow field. The computational algorithm uses parameters that are related to the creation and dissipation of turbulence. Some, like the Intensity and Length scale parameters, are based on the assumption that the characteristic velocity and the characteristic length of the larger turbulent eddies in the solution are of the same order as the velocity and length scale of the mean flow, a major assumption of CFD turbulence models. Wind tunnel flows vary in their turbulence characteristics, with turbulence intensity factors being as low as 0.05%. Intensity and length scales can be based on fully developed duct flow by the formula and approximate relationship below:

$$I \equiv 0.16(\mathrm{Re}_{D_H})^{-1/8} = 1.99\% \qquad l = 0.07L = 0.07(4m) = 0.28$$

which gives values of k and e of:

$$\kappa = \frac{3}{2}(U_{avg} I)^2 = 1.83 \qquad \varepsilon = C\mu^{3/4}\frac{k^{3/2}}{l} = 1.45$$

The choice of these turbulence parameters are used for the inlet BC's and will change as the model solves. The converged solution should be independent of the initial values, yet it is important to start with reasonable values for the k and e, as it affects the downstream flow. Other turbulence factors can be derived from these numbers, including the Modified Turbulent Viscosity and the Specific Dissipation Rate (used for K-omega models).

Another, separate, parameter used to define the turbulence in CFD models in the Turbulent Viscosity ratio. This ratio is directly proportional to the turbulent Reynolds number, which is defined in terms of the turbulent kinetic energy, the dissipation rate and the viscosity, and is usually in the range of a value between one and ten.

Each one of the turbulent initial boundary conditions has assumptions which relate one to another, and is a source of error. Some, like the relationship between the turbulent dissipation rate and the length scale are based on empirical constants. To accurately model a wind tunnel situation these initial turbulence values would need to be calculated from empirical data to obtain accurate results. Fluent recommends for wind tunnel situation in which the model is mounted in a test section downstream of a mesh screen of using a value of e based on the decaying turbulence in the length of the wind tunnel. If we assume that turbulence decay (for the main flow) to be 10% in a 20 meter wind tunnel, we get an approximate value for e to be:

$$\varepsilon = \frac{\Delta k U_\infty}{L_\infty} = \frac{.1(55.6)}{20} = 0.278$$

which eliminates the need for the length scale assumption based on the hydraulic diameter. Using this method, it is important to check turbulent viscosity ratios to ensure they are not out of range. Another way is to specify the initial guess for e so that the resulting eddy viscosity $(C_\mu \frac{k^2}{\varepsilon})$ is sufficiently large in comparison to the molecular viscosity, and base the value for e on the rule of thumb that the turbulent viscosity is roughly two orders of magnitude greater than the molecular viscosity. As can be seen, the range in which the initial turbulence values vary, based on assumptions, is considerable. It is possible that the turbulent initial conditions should be set at the scale of the cylinder, in which case the length scale would be on the order of 0.66m x 0.07. Initial runs, were, in fact run with this figure.

The effect of the wind tunnel walls.

The wind tunnel walls were models as the standard no-slip WALL type boundary in Fluent. They were quite close to the cylinder, computationally speaking, and have an effect on the solution both because of the way the solution computes, as well as the differences between bounded and unbounded flow. Since out model is modeling bounded wind tunnel flow, is important to model the wind tunnel walls quite carefully with enough grid points to accurately model a boundary condition that reflects a no-slip wall velocity profile. In my earlier grid models, this was not done to the precision required for this problem and is likely a major source of error. Checking the effects of the walls is done by changing the boundary condition for the top and bottom walls from WALL to OUTFLOW, and can be seen in the last part of the results.

Grid

Gambit was the preprocessor used to create the geometry and the grid. Both triangular and quad meshes were created, in total over 15 grids of varying dimensions and element types. Problems became immediately apparent with a poor grid, namely immediate high y+ values at the boundaries. In the results following, it was clear from the high y+ values (on the order of 1500-3500) that even my finest grid was too coarse in the boundary layers. The aspect ratio can be a source of error when the longer side of an element is facing the direction of flow; it is preferable to have elements with a large aspect ratio with the short side facing the flow. The other factors that are important to consider when creating a grid are the skew, and the gradient from one element to another. For proper turbulence modeling, opinions differ, but a gradient of 5% (change in size from one element to the next) should be adequate. More comments regarding grid types and geometry can be found in the results. It was helpful to check the grid in Fluent, and reordering the domain sped things up significantly, especially for the models which had multiple faces. The ideal quad grid for this problem has very fine resolution at the circle and fine resolution at the wind tunnel wall boundaries. Spacing of mesh points in between the wall and the circle can be larger. Similarly, to minimize excessive grid points, the spacing of mesh seeds behind the circle could grow to create elements with very long aspect ratio which, aligned with the flow, should solve adequately.

Grid Convergence.
It is possible to use the results from successively finer grids to calculate the error resulting from the grid. Grid convergence calculations on a sequence of grids based on results pages 16 and following are presented here:

Grid	*Number of Nodes*	*Functional (-)*	*Functional*
Medium	31368	Lift=1024.7	Drag=960.1
Fine	47095	Lift=1179.2	Drag=930.2
"SuperFine"	97096	Lift =1278.5	Drag=908.0

The refinement ratio between the grids is given by

$$r_{fine-medium} = \frac{47095}{31368} = 1.5 \qquad r_{superfine-fine} = \frac{97096}{47095} = 2.1$$

and the relative errors:

$$e_{fine-medium}(lift) = \frac{1024.7 - 1179.2}{1179.2} = -0.131$$

$$e_{superfine-fine}(lift) = \frac{1179.2 - 1278.5}{1278.5} = -0.0777$$

The grid convergence indexes using the fine and the medium grids are:

$$GCI_{fineGrid} = \frac{3(0.131)}{1.5^2 - 1} = 0.3144$$

$$GCI_{mediumGrid} = \frac{3(0.131)1.5^2}{1.5^2 - 1} = 0.707$$

The grid convergence indexes using the fine and superfine grids are:

$$GCI_{superfineGrid} = \frac{3(0.0777)}{1.5^2 - 1} = 0.186$$

$$GCI_{fineGrid} = \frac{3(0.0777)1.5^2}{1.5^2 - 1} = 0.419$$

Using these estimate we can estimate the error in our solutions due from the grid, we find the value for lift for the superfine grid is in about 18% error due to grid parameters.

CFD MODELING AND PARAMETERS

General

CFD models solve fluid problems with five basic equations of state:

- Mass conservation (continuity)
- x,y,z momentum equations
- Energy equations

Among the unknowns are density, pressure, temperature, and internal energy. Additional relationships among the variables are derived from the Navier-Stokes equations, in which the viscous stress components are integrated with the momentum equations. For the rotating cylinder problem, we assume that the fluid (air) is behaving as an incompressible fluid at constant temperature (since the Mach number is low), and therefore do not need the energy equation. The results following only use the mass conservation and momentum equations of state.

For turbulent modeling, additional transport equations are used to account for time averaged fluctuations in the local fluid velocities that occur with turbulence, which give rise to additional stress on the fluid, the Reynolds stresses. Reynolds stresses create 6 more unknowns in the equations (three additional unknowns for 2D problems), and are accounted for by additional approximated equations, depending on the model.

Discretization

The solution process depends on solving partial differential equations. CFD uses a control volume technique to convert the PDE's into algebraic equations that can be solved numerically, resulting in a solution which satisfies the governing equations in every element in the grid. Fluent uses a upwind/ central differencing scheme, in which the convection terms are solved using upwinding, and the diffusion terms are centrally differenced. First order upwind schemes assume that the cell center values of the variables represent the cell-average value and the face values of the control volume have the same value. Second order schemes include the second order term of a Taylor series expansion of the PDE's, and are more accurate. Fluent also offers the QUICK Scheme which are based on averaging the second order upwind and central differences of a variable. The QUICK scheme is recommended for structured quad grids which are aligned with the flow direction.

Convergence and Under Relaxation

The model is considered solved mathematically when the values in the entire grid are "converged" and do not change significantly from one iteration to the next. The change in value is known as the residual. Depending on the model chosen, the necessary residual levels for convergence varies. First order schemes generally converged adequately when the residual level was set at 0.001, while the second order schemes required a lower residual value, not necessarily at 0.0001 but somewhere in between. There are no hard and fast rules relating convergence to the residuals; for example, if the initial conditions are initialized at the values close to the final solution, there will be small residuals; likewise, if the initialization is very different from the final solution, there will be a larger drop in residuals. Generally convergence level was monitored by the slope of the residual plot, and by looking at the change of a functional. When the functional value stabilized to a certain value, then convergence was likely. This can be monitored visually by turning on the plot forces charts.

Sometimes models converge very slowly, in this case, it is possible to change the under relaxation factors of the solution process. This needed to be done somewhat carefully, as the solution could seemingly converge quicker with higher under relaxation factors. For some models, like the Reynold's Stress Model, Fluent recommends lowing the under relaxation numbers dramatically, from the defaults (on the order of 0.7) to 0.2 or 0.3. Fluent also recommends using residual levels of 0.0001 for the RSM.

Near wall treatments and boundary layer approximations for turbulence models.
In free stream flow, where Reynolds numbers are high (on the order of 100,000 to 10,000,000), the inertia forces in the fluid are much greater than the viscous forces, and the equations are solved on the basis of small viscous components. However, at wall boundaries, where velocities approach zero, the viscous forces will be equal in order of magnitude of the inertia forces, or even larger. Therefore, a separate calculation is required for such regions which include the effect of the viscous forces. A boundary layer can be sub divided into 3 parts:

Viscous sublayer: the layer closest to the wall where viscous effects dominate and the flow is laminar (as turbulent eddying motions must stop in layers where there is no velocity).
Transition layer: where both viscous and inertia effects are considered.
Outer layer: where inertia forces are dominant and the flow is turbulent.

In CFD models, the measurement which determines the layer region (and thus the specific calculations that are performed) is the value known as y+, which is a dimensionless value based on the values of density, viscosity, shear stresses due to velocity, and the distance from the wall ($y+ \equiv \rho u_t y / \mu$).
For low values of y+ (y+<11.225 in Fluent), the flow is modeled as laminar. For the transition layer, with mid range values of y+ (30 < y+ < 60), the flow is modeled as turbulent where viscous forces are included in the calculations. Shear stresses are assumed constant in this region and equal to the wall shear stress. The outer, turbulent layer is modeled with inertia forces as dominant in the CFD calculations, and direct viscous effects are ignored.

Fluent uses two equations to calculate the wall shear stress based on the value of y+. If the value of y+ in the cell adjacent to the wall is within 11.225, Fluent uses the laminar stress strain relationship:

$$\frac{u}{u_t} \equiv u^+ = y^+$$

If the value of y+ is greater than 11.225, then fluent uses the Law of the Wall equation to solve the flow for the cell adjacent to the wall boundary:

$$u^+ = \frac{1}{\kappa} \ln E(y^+)$$

where k and E are constants based on empirical data. E is related to wall roughness.

In modeling with CFD, it is important to ensure that the grid is fine enough at the wall boundaries to ensure that the y+ values are not too high, as it is in this region where the CFD programs include viscous shear stresses in the calculations. The y+ values at the wall boundaries is marginally affected by the choice of model (which may vary in their calculations of turbulent viscosity), but the main factor that affects the y+ values is the mesh. Generally, the first thing to do after running a model is to plot y+ values at the boundary to check to see the range of values. If the y+ values of the cells adjoining the wall surface are greater than 500 (at best), the results there will be in error, as the program will be including viscous effects in regions of the model that are outside the transition boundary layer. A finer mesh is required in such cases. y+ values of 60 or less are desired, but note that the maximum allowable y+ value, where the values of the log of y+ plotted against u+ are linearly related, increases as the Reynolds number increases.

Fluent uses two main choices for modeling the wall. The Enhanced Wall Treatment is the preferred model, as it includes reasonable equations for all points in the mesh. It is chosen as an option in fluent, which is somewhat deceiving, since one would assume that solving as accurately as possible with the equations of state and the turbulent equations in every cell would be the default. Fluent recommends having a mesh which results in y+ values between 1 and 5 in the mesh points adjoining the wall boundary as a minimum, something I did not see as being possible in a mesh of less that 300,000 nodes for our problem. However, Fluent uses a two layer approach which includes a blending term which allows coarser meshes to solve even if the mesh elements adjacent to a wall are not within the inner boundary layers. If the Enhanced Wall Model is used, there is an option to include pressure gradient effects.

The other model Fluent uses is the Wall Functions, and is the default. Wall functions make additional assumptions regarding the transition layer, and when wall function is selected, Fluent does not solve directly for flow in this viscosity affected region, rather, calculations are based on assumptions arising from the linear part of the log y+/u+ plot. Still, it is recommended to keep y+ values close to 30 when using this model, although values up to 500 may be acceptable for high Reynold's number flows. Above these numbers, the wake component becomes larger (where velocities may change direction).
The non-equilibrium wall functions option is a variation to the standard wall functions which include pressure gradient calculations, and are recommended for flows which involve separation (although the enhanced wall treatment is more applicable in such flows).

Turbulence models
Various models were utilized to study the rotating cylinder in the wind tunnel, and varied considerable in the results. The total number of choices and sub-choices are staggering. Excluding the variations of the setting of initial conditions, which varied from model to model, as well excluding energy solution parameters, the total number of turbulent model options exceeds 50, if one includes the choices of wall treatments for each model. Without direct experience with each specific model, it is difficult to choose the best one for our scenario. For example, it is difficult to know on what scale "swirl" is fluid dynamically defined without seeing computational examples of it. Many models were tried, and many failed. The basis of each turbulence model is discussed here.

Spalart-Allmaras Turbulence Models
The Spalart-Allmaras is a one-equation (in addition to the general transport equations) turbulence model which solves for the turbulent eddy viscosity, and eliminates the need to calculate the length scale related to local turbulence. Although designed to be used with meshes that fully resolve the inner boundary layer (i.e, cells with y+ values of 1-11.25), in which case the enhanced wall treatment option should be selected, it has been shown to be adequate for coarser meshes if the wall function option is chosen. When using this model, the Strain/Vorticity based turbulence production option should probably be used for flows which include separation, as it includes the effects of the mean strain due to rotational velocity gradients. My runs show mixed results with the Spalart Allmaras.

K-e Turbulence Models
The K-e models are the workhorse of the CFD industry, and have been shown to be robust in a variety of applications. K-e models use two equations to calculate turbulent Reynolds stresses, based on calculations of turbulent kinetic energy (k), and turbulent dissipation (e). The k-e model uses additional constants based on empirical data from a wide range of flows. However, although the turbulent kinetic energy can be calculated quite precisely, the dissipation rate is a bit of a fudge, as there are many unknown and unmeasurable terms in the dissipation equations. The basic assumption is that turbulent dissipation is on the same order of the turbulent production term, and the K-e model applies mostly to flows that are fully turbulent, where the effects of molecular viscosity are negligible. The model has

problems with flows that have curved boundary layers, as our rotating cylinder problem does, yet it was used for many of the runs in the results because of its consistent results.

Besides the standard K-e model, two more variations are available in Fluent. The RNG model also uses two equations for turbulence kinetic energy and dissipation, yet has different constants and calculates the effective viscosity differently. It is considered more responsive to flows with rapidly changing strains, and for flows with high streamline curvature. Two options are available with RNG: "differential viscosity", which accounts for low Reynolds number changes to turbulent viscosity, and is only recommended for grids which are refined in the boundary layer. The other is "swirl dominated flow", which also affects the turbulent viscosity calculations and used for flows which have significant swirl. When using the RNG model, it is recommended to begin with a solved solution from the standard K-e model, although I didn't have much luck with this technique. The third K-e model is the Realizable model, which uses a similar equation for turbulent kinetic energy, but calculates turbulent dissipation independent of the rate of production. This makes the model more likely to diverge with a poor model, but is shown to produce better results for flows which include mixing layers and boundary separation. Good results were had with the Realizable model.

K-omega Turbulence Model
The K-omega model also uses two equations to represent the turbulence, but instead of e, it calculates a specific turbulence dissipation rate, which can be considered the ratio of e to k. It is recommended for low Reynolds number flows, and does not offer the standard wall function choices in Fluent, as by default, it uses the enhanced wall treatment methods described above; that is, it solves for near wall treatments when the mesh is fine enough, and uses wall functions when the mesh elements are not placed entirely within the boundary layer. The SST (Shear-Stress Transport) K-omega model takes into account the transport of turbulent shear stress, and makes a gradual change of solution variables from the standard K-omega model in the inner region of the boundary layer to a high Reynold's number version of the K-e model in the outer part of the boundary layer. The SST K-omega model is more reliable for flows which have adverse pressure gradients, and would be the preferred model, of the two k-omega choices,for modeling the rotating cylinder, as it includes boundary separation. Options for the K-omega model include transitional flows (a low Reynolds number correction to turbulent viscosity), and for the standard K-omega model, shear flow corrections.

Reynold's Stress Equation Turbulence Models
The most advanced (requiring the most computational resources) classic turbulence model is the Reynold's Stress Model (RSM), which uses equations to solve directly for the Reynold's stresses. Five additional equations are involved in the 2D model (seven in 3D), and because it makes fewer assumptions about the turbulent eddy formations and dissapations, it is considered the "simplest" and most accurate of the turbulence models for fluid flows.

Recommendations
The standard K-e model with enhanced wall functions, is the model that seemed to work overall the best for my models, despite its not being recommended for problems due to the high pressure gradients and separated flow. There were also good runs with the K-e Realizable with enhanced wall functions. Other possible models would include the K-omega SST with the transitional flows option, and perhaps even the Spalart Allmaras with the Strain/Vorticity option and with standard wall functions, for the coarser meshes. For a three dimensional model, perhaps swirl would be a consideration on the ends of the cylinder, and would require a model like the RNG with swirl modifications. The RSM model solved occasionally in my runs, but requires better meshwork than I was able to provide. Perhaps it too would be the best choice for a 3D model. Second order descretization is also preferred.

Errors, Validation, Verification
Our goal here was to predict the characteristics of flow over a rotating cylinder in a wind tunnel. The CFD model contour plots give a good indication of the general flow characteristics, however, there are many sources of error, including the 2D assumption, the approximations made in the discretiazation calculations, the compressibility of the flow (this model is in the subsonic range), the mesh error, the lack of energy modeling, and the assumptions made in the boundary conditions.

Validation
The rotating cylinder problem is modeled as 2D, and assumes an infinite width cylinder, and thus does not take into account flows around the edges that would occur in a real world test. To accurate model the wind tunnel test, a 3D problem would need to be set up. In addition, more precise knowledge would need to be obtained about the actual turbulence conditions in the wind tunnel. The exact temperature and pressure of the flow would also need to be monitored and recorded. The boundary condition set at the outflow is also an assumption (constant pressure). In addition, the cylinder walls are modeled as smooth with default roughness variables. The exact surface characteristics and how they affected the creation and dissipation of turbulence would also require accurate measurement to compare the CFD model with the physical one. From my data, I was getting results which indicate a lift/drag ratio which varied from about 1.6 to 1.1. Empirical data suggests the actual values for lift/drag of a rotating cylinder to be on the order of 1, but perhaps this takes into account edge effects, which would lower the L/D ratio. The theoretical inviscid solution gives lift values of around 7000, my values were on the order of 1200. This is no help in validation, but the drag coefficient of an infinitely long (non-rotating) cylinder is available, which gives a value of approximately 0.6, which is in range of my results:

$$C_d = \frac{\sim 850}{1/2\rho U^2 D} \approx 0.68$$

.

Verification
With the results from the CFD models, we can verify the appropriateness of the result by examining the assumptions made in the calculations. The grid errors are discussed above, as well as the quality, which involves the grid size, shape and type. We can also verify the assumptions made in each of the turbulent models, related to how the actual turbulence is created, and what factors are included, as well as the entire CFD process, which, for example, truncates all the third order and higher terms from the Taylor and power series expansion when solving the PDE's. CFD methodology is discussed previously. Looking at the contours, we can verify if the results from the CFD are realistic in terms of the flow characteristics (see discussion following). And of course, the CFD results can be verified by tests done in a wind tunnel if the conditions can be modeled exactly. This would include studying and measuring the boundary layer and moment of separation, pressure distribution along the cylinder walls, and measurements for turbulence around the cylinder.

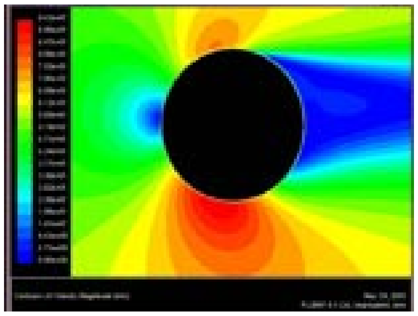
Summary
CFD is a delicate process of balancing many assumptions to predict real world fluid flows. To accurately model a fluid flow, it is important to have a excellent understanding of the flow features prior to creating a grid and selecting a CFD model, as different models make different assumptions which are specific to certain types of flows. A good understanding of every process in the CFD analysis is vital to obtaining reasonable results, and extensive experience in comparing computed results to experimental tests would be essential for a researcher to have prior to predicting untested flows.

Results and General Flow Features

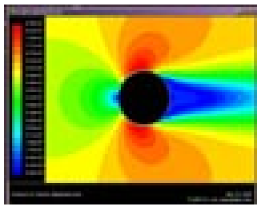
Please see the next 20 or so pages for results. Pertinent results can be found on the last three pages, from which produced the following plots. Cylinder is rotating counter clockwise is all runs.

General Flow Features

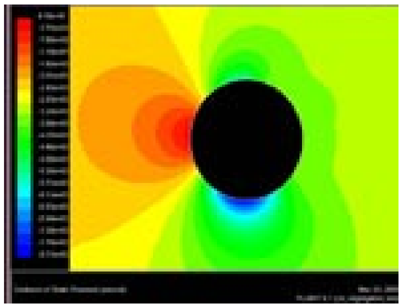
Right: This velocity contour shows a stagnation point at the center front position, and a forward upper separation point and a delayed lower separation point. The wake behind the cylinder is shifted upwards relative to a non-rotating cylinder.



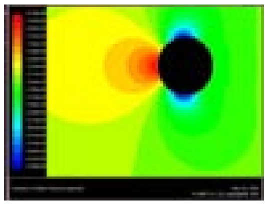
Right: zero rotation velocity contour for comparison



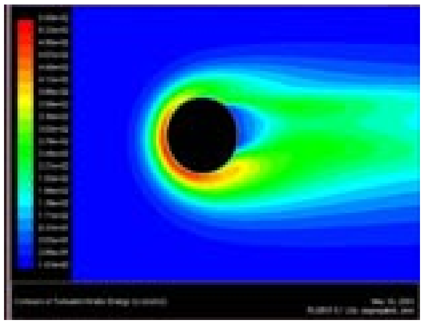
Right: The pressure contour shows a high pressure spot for front center, with a pressure imbalance on the top and bottom center points. The lower pressure area is the greater than the lower pressure area on the zero rotation model, below, while the upper pressure area on the rotating model is higher than the zero rotation model.



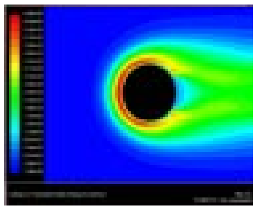
Right: Zero rotation pressure plot for comparison

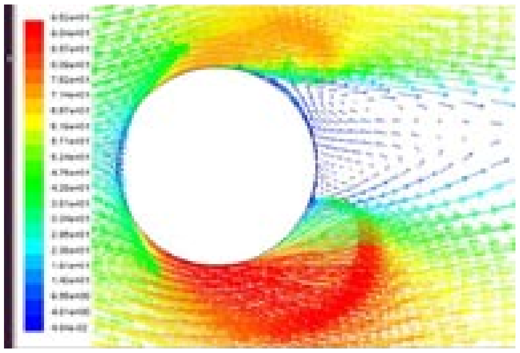


Right: The turbulence contour plot shows high turbulence along the front edge of the cylinder, with a pocket of lower turbulence in the wake near the cylinder. The displaced wake diverts the main flow, which in turn affects the velocity differential, affecting the pressure imbalance which creates the lift.

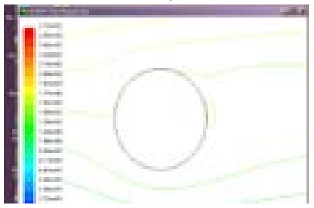


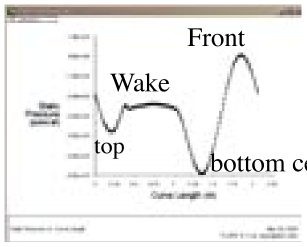
Right: Zero rotation turbulence plot for comparison



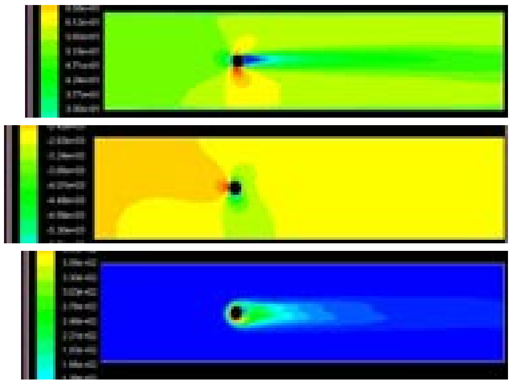


Above: Velocity vectors showing the wake pattern of the region behind the cylinder. Below: the velocity streamline plot shows the stagnation and separation points more clearly.





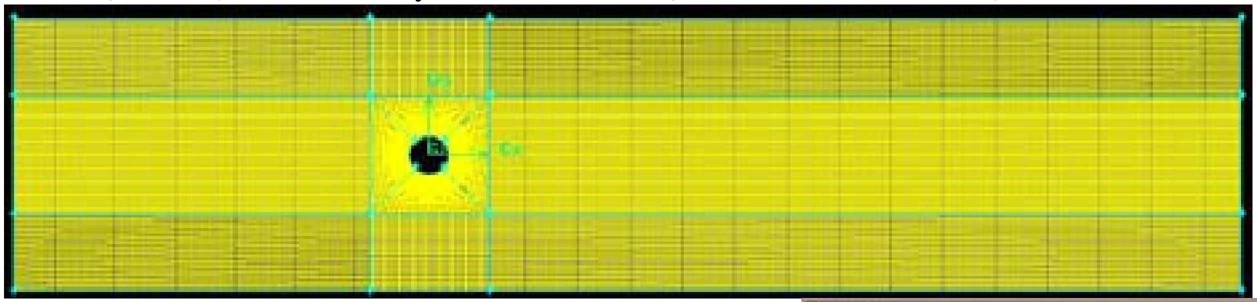
Above: Pressure distribution on the wall of the cylinder. The forward stagnation is the high point of the curve, while the two low points are the top and bottom pressure. The flat area is the wake.

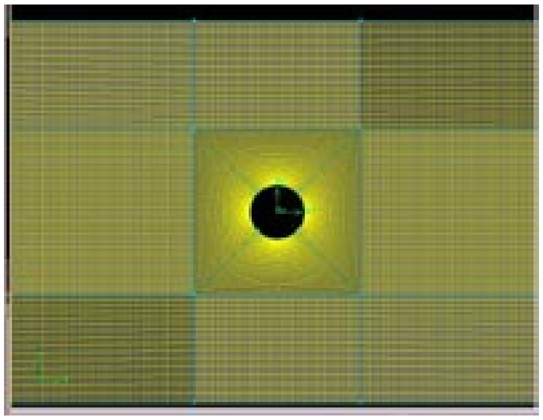


Above right: Contour plots of velocity, pressure, and turbulence. The velocity shows the upper/lower differential, while the pressure plot shows that the gradient reaches the wind tunnel walls. Turbulence in this model diminishes considerably after about 5 diameters of the cylinder.

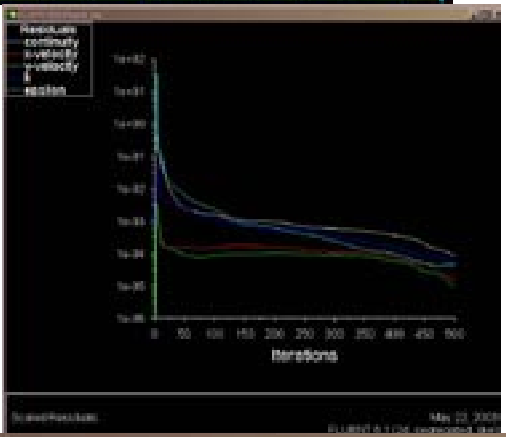
RESULTS Page 1

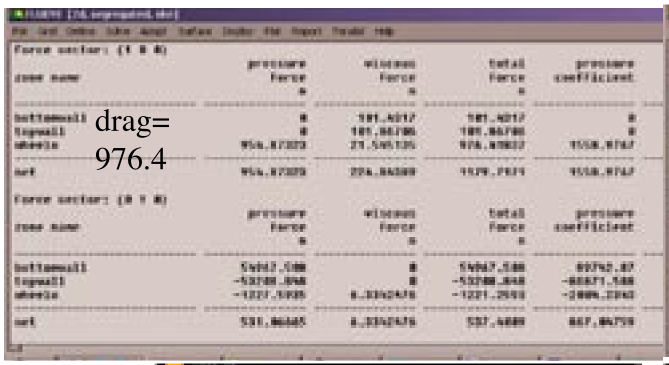
Initial Runs with Quad Mesh and standard K e model and second order discretiation. This is where I learned about the proper way to check for convergence.. These tests were run with my initial medium quad mesh (12 faces), which initially had 57073 nodes (200 mesh seeds on circle).





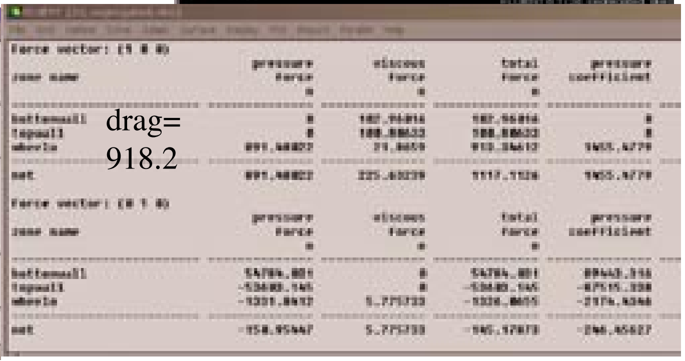
First run converged to .001 residuals after 140 iterations, second with 0.001 residual level at 500 iterations. Left side is 0.001 convergence, right side is 0.0001 convergence. After this I decided to check convergence by examining the residual plat and checking when the residuals level off.





```
Force vector: (1 0 0)
                              pressure        viscous          total       pressure
zone name                        force          force          force    coefficient
                                     n              n              n
---------------------------- -------------- -------------- -------------- --------------
bottomwall                           0      101.4217       101.4217              0
topwall                              0      101.86786      101.86786             0
wheels                       954.87323      21.545135      976.41837      1558.8767
---------------------------- -------------- -------------- -------------- --------------
net                          954.87323      224.84589      1179.7131      1558.8767

Force vector: (0 1 0)
                              pressure        viscous          total       pressure
zone name                        force          force          force    coefficient
                                     n              n              n
---------------------------- -------------- -------------- -------------- --------------
bottomwall                   54962.588              0      54962.588      89742.87
topwall                     -53708.848              0     -53708.848     -86671.588
wheels                      -1227.5935      6.3342476     -1221.2593     -2004.2343
---------------------------- -------------- -------------- -------------- --------------
net                          531.86645      6.3342476      537.4089       867.84759
```

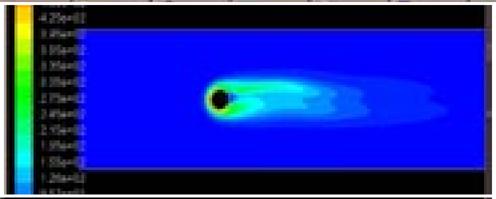
drag=
976.4

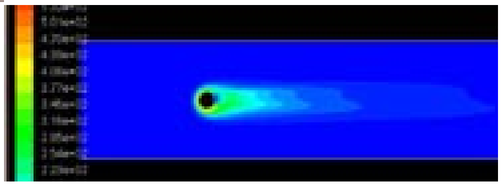


```
Force vector: (1 0 0)
                              pressure        viscous          total       pressure
zone name                        force          force          force    coefficient
                                     n              n              n
---------------------------- -------------- -------------- -------------- --------------
bottomwall                           0      107.95016      107.95016             0
topwall                              0      108.88633      108.88633             0
wheels                       891.44827      21.8659        913.34612      1455.4779
---------------------------- -------------- -------------- -------------- --------------
net                          891.44827      225.63239      1117.1126      1455.4779

Force vector: (0 1 0)
                              pressure        viscous          total       pressure
zone name                        force          force          force    coefficient
                                     n              n              n
---------------------------- -------------- -------------- -------------- --------------
bottomwall                   54784.801              0      54784.801      89443.316
topwall                     -53608.145              0     -53608.145     -87515.338
wheels                      -1331.8412      5.775733      -1326.0655     -2174.4346
---------------------------- -------------- -------------- -------------- --------------
net                         -154.95447      5.775733      -149.17873     -246.45627
```

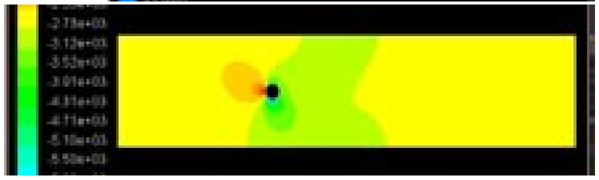
drag=
918.2

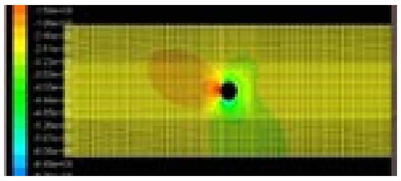
Above:
LIFT=
-1221.3



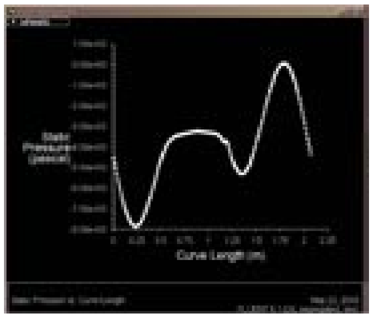


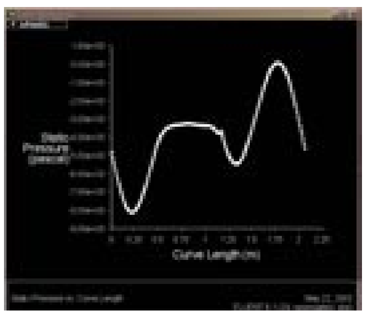
Above:
LIFT=
-1326.1





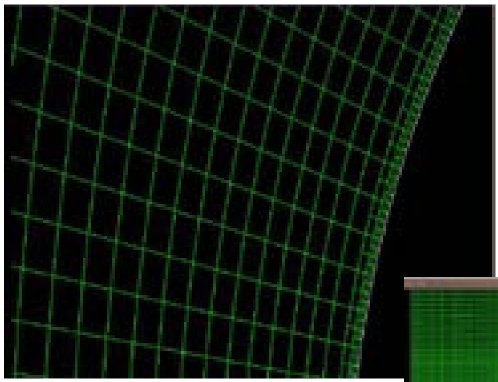
Turbulence, Pressure contour, and pressure on surface of circle for 0.001 convergence level (140 iterations)

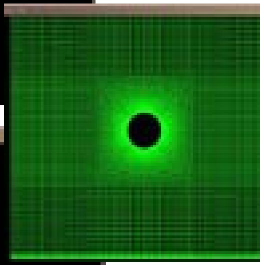




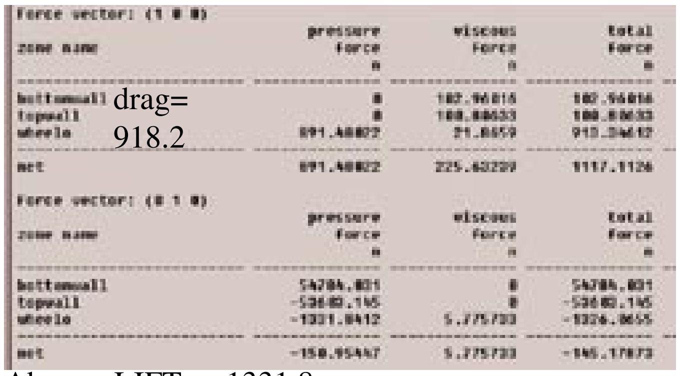
Turbulence, Pressure contour, and pressure on surface of circle for 0.0001 convergence level (>500 iterations). Note the more developed turbulent flow contour.

PAGE 2. Next I experimented with Adapted Grids, using both y+ and boundary adaptation. Later found out that the boundary adaptation process could cause undue effects due to adverse mesh gradients, but the y+ adaptation worked well (y+ adaption usually added no nodes after boundary adaption).





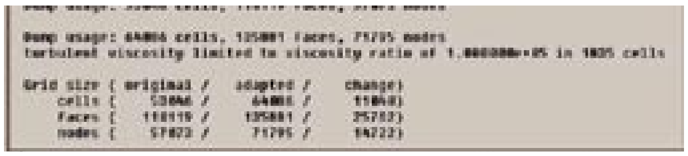
Left: Boundary Adapted Grid.
Right: Forces after adapting grid.



```
Force vector: (1 0 0)
                                     pressure         viscous           total
zone name                               force           force           force
                                            n               n               n
-------------------------------- --------------- --------------- ---------------
bottomwall                                  0      102.96016       102.96016
topwall                                     0      100.80633       100.80633
wheels                              891.40022        21.8659        913.26612
-------------------------------- --------------- --------------- ---------------
net                                 891.40022      225.62229       1117.1126

Force vector: (0 1 0)
                                     pressure         viscous           total
zone name                               force           force           force
                                            n               n               n
-------------------------------- --------------- --------------- ---------------
bottomwall                          54784.031               0       54784.031
topwall                            -53680.145               0      -53680.145
wheels                             -1331.0412        5.275733      -1326.0655
-------------------------------- --------------- --------------- ---------------
net                                -150.95447        5.275733      -145.17873
```

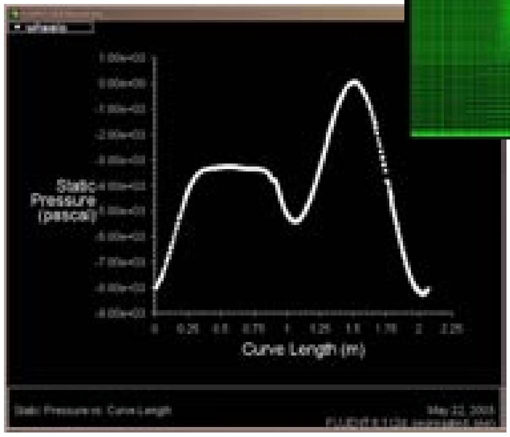
drag=
918.2

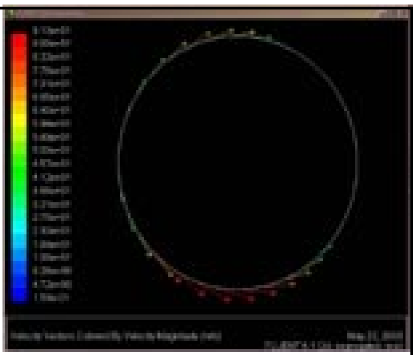
Above: LIFT = -1331.8



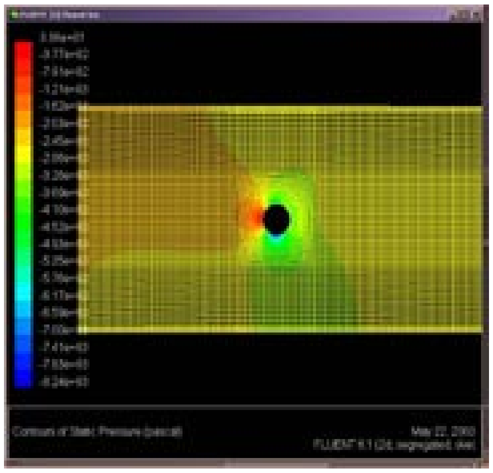
```
Dump usage: 64806 cells, 135001 faces, 71795 nodes
turbulent viscosity limited to viscosity ratio of 1.000000e+05 in 1035 cells

Grid size ( original /    adapted /    change)
    cells (    53066 /      64806 /     11040)
    faces (   110119 /     135001 /     25782)
    nodes (    57073 /      71795 /     14722)
```

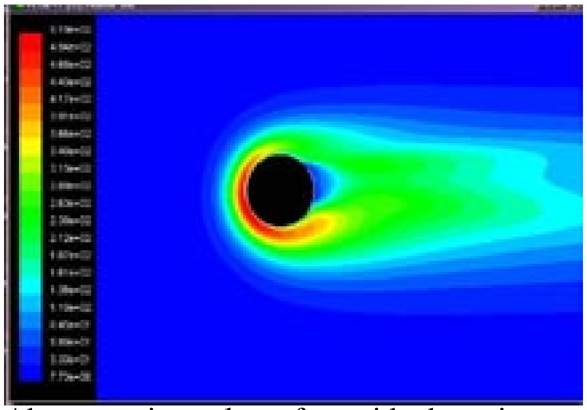
Above: 57073 nodes to 71795 nodes




Velocity Vectors Colored By Velocity Magnitude (m/s)
May 22, 2003
FLUENT 6.1 (2d, segregated, ske)

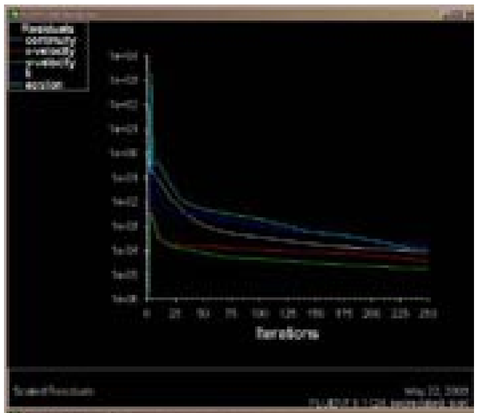
Right: velocity vectors on the spinning (168 radians/sec) circle. Note the spinning circle does not have enough rotation to counter the oncoming flow, despite the fact that it is a no-slip wall.. This seemed to be independent of the grid size, as after several boundary adaptions, the velocity vectors still did not reverse on the surface of the circle. Yet when the rotation was increased to 1680 radians/sec, the vectors on the top surface did reverse.

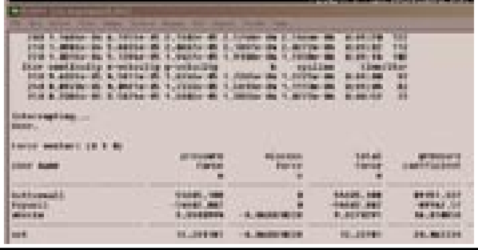

Contours of Static Pressure (pascal)
May 22, 2003
FLUENT 6.1 (2d, segregated, ske)



Above: various plots after grid adaptation.

Side Notes
(note: I also ran other models here, including RNG, Reynolds, Spalart-Allmaras, but this was prior to my knowing the correct parameters for them, so the results were bad.

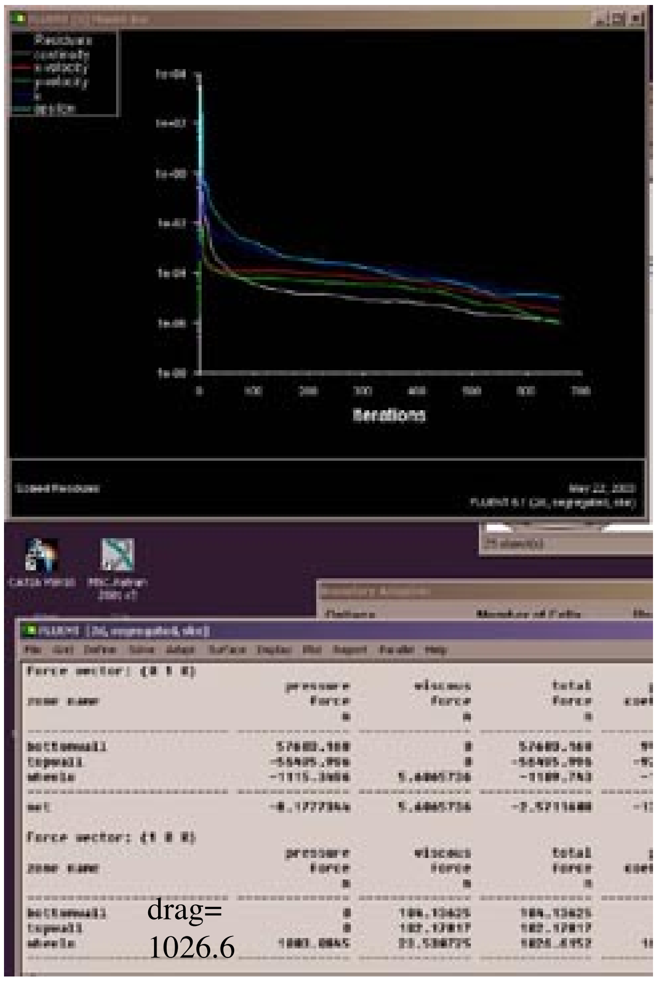




```
[illegible]

Interrupting...
Done.

Force vector: (0 1 0)
                      pressure      viscous        total     pressure
zone name                force        force        force  coefficient
                             n            n            n
bottomwall           [illegible]          0  [illegible]  [illegible]
topwall              [illegible]          0  [illegible]  [illegible]
wheels               [illegible]  [illegible]  [illegible]  [illegible]

net                  [illegible]  [illegible]  [illegible]  [illegible]
```

Left: A quick check of forces after a run with zero rotation on the circle. Note lift force error (should be zero) probably due to coarseness of grid. LIFT = -9.027

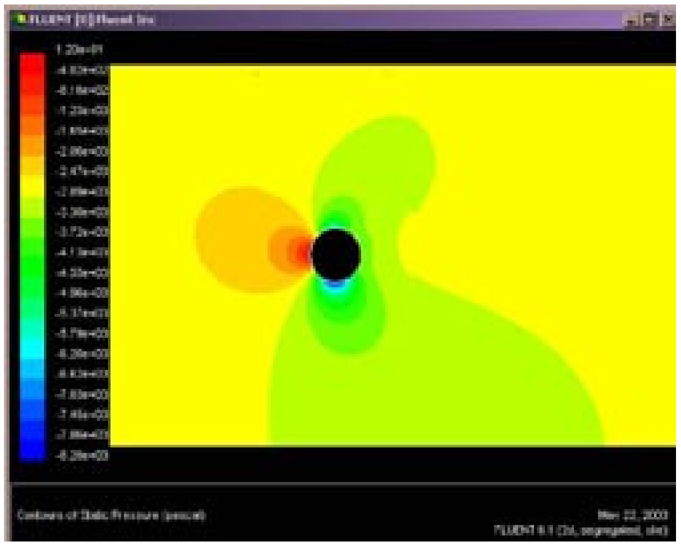
PAGE 3. Next I ran some tests with SIMPLE and first order solutions, with 0.0001 and 0.00001 residual levels. There was no difference between the residual levels. These were with the adapted grid, and the values for the lift were lower than the second order solution.

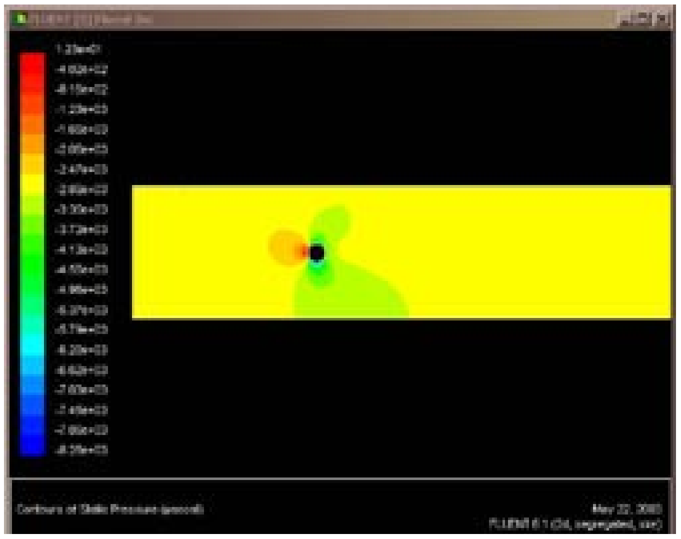


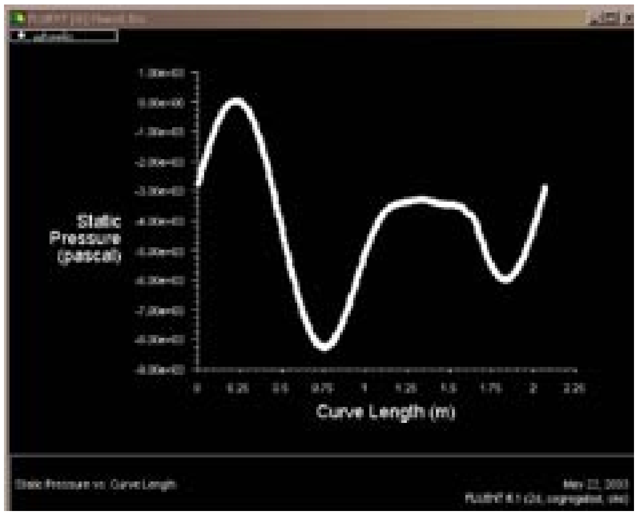
```
Force vector: (0 1 0)
                                  pressure          viscous            total
zone name                            force            force            force
                                         n                n                n
-------------------------  ---------------  ---------------  ---------------
bottomwall                       57603.168                0        57603.168
topwall                         -56405.956                0       -56405.956
wheels                          -1115.3686        5.4005736        -1109.753
-------------------------  ---------------  ---------------  ---------------
net                             -8.1777344        5.4005736       -2.5711608

Force vector: (1 0 0)
                                  pressure          viscous            total
zone name                            force            force            force
                                         n                n                n
-------------------------  ---------------  ---------------  ---------------
bottomwall                               0        104.13625        104.13625
topwall                                  0        102.17817        102.17817
wheels                           1003.0845        23.530725        1026.6152
```

drag=
1026.6

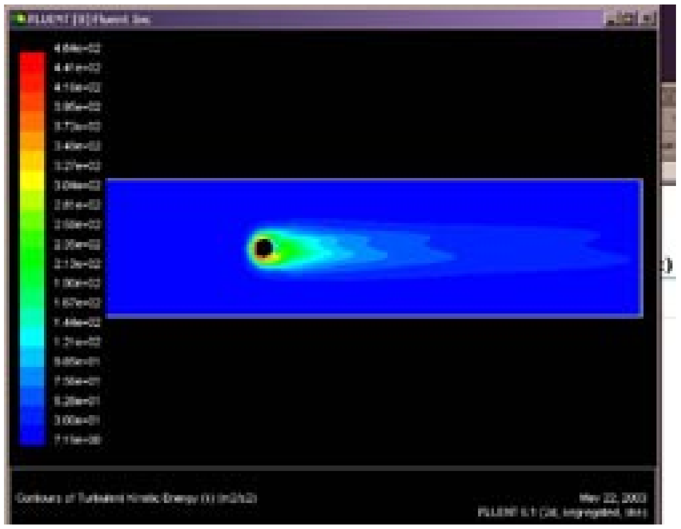
Above: LIFT= 1109.7 (first order solution)


Contours of Static Pressure (pascal)
May 22, 2003
FLUENT 6.1 (2d, segregated, ske)

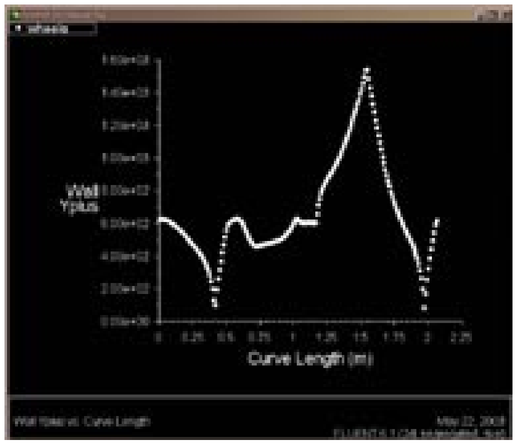

Contours of Static Pressure (pascal)
May 22, 2003
FLUENT 6.1 (2d, segregated, ske)

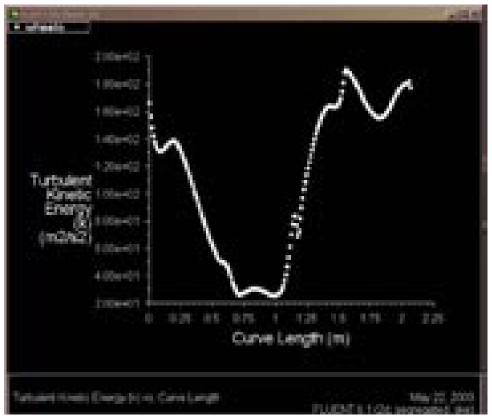


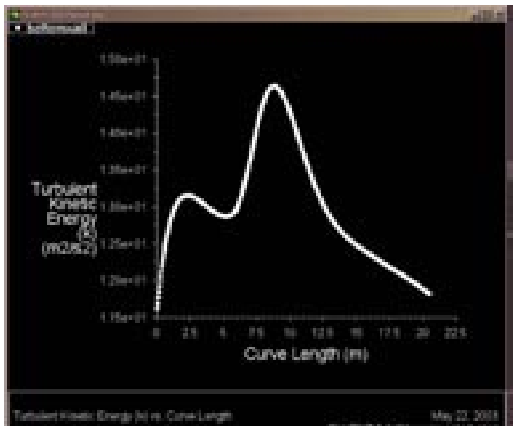
Side note: around this time I also tried a run with the PISO algorithm, but it failed to converge.

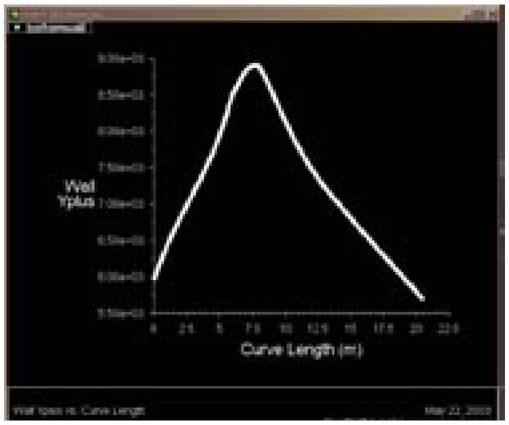

Contours of Turbulent Kinetic Energy (k) (m2/s2)
May 22, 2003
FLUENT 6.1 (2d, segregated, ske)

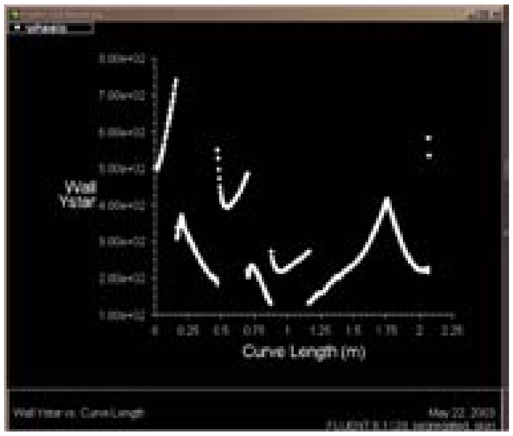
PAGE 4. Below are various other plots of the initial experimental runs with the same Medium Quad mesh.(top four before grid adaption).

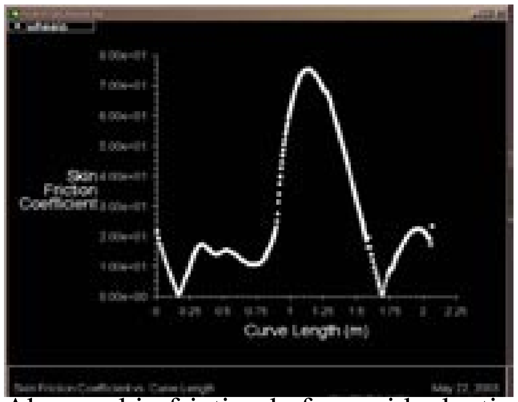




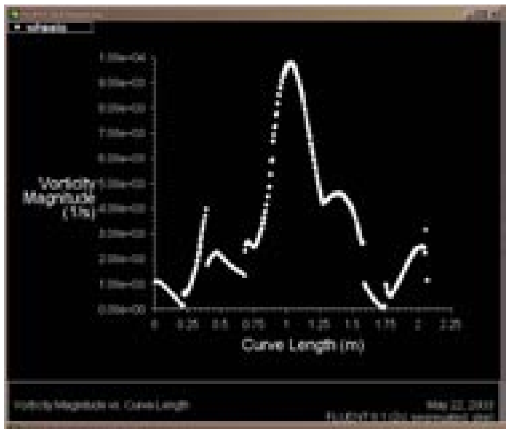


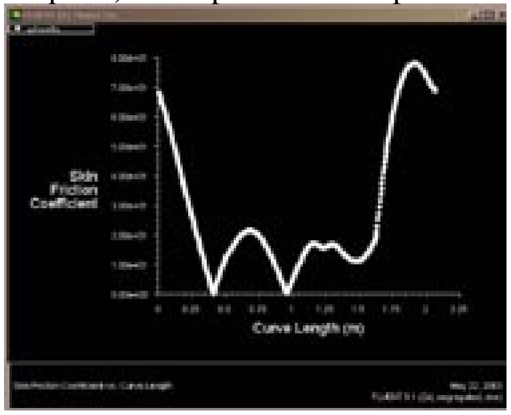




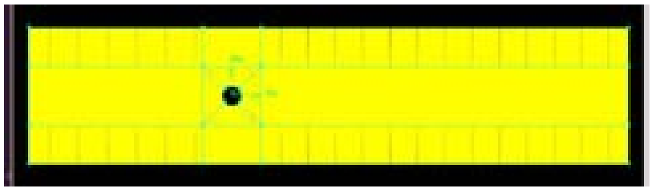


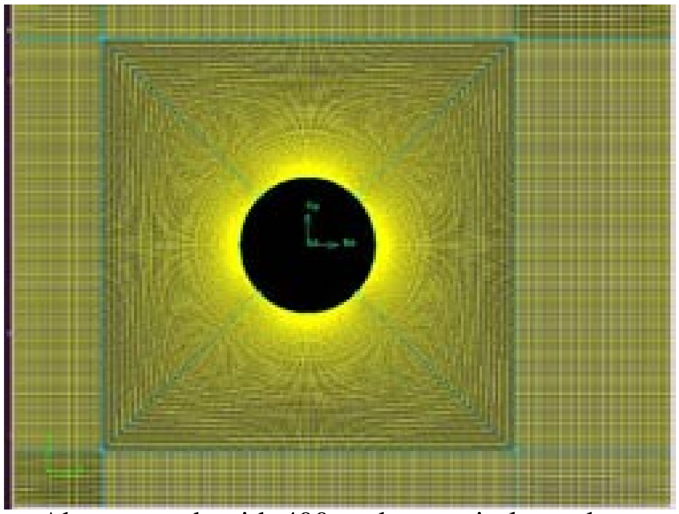
Above: skin friction before grid adaption, Below: after adaption (nearly identical except the zero point has changed with grid adaption). Left: plots after adaption.



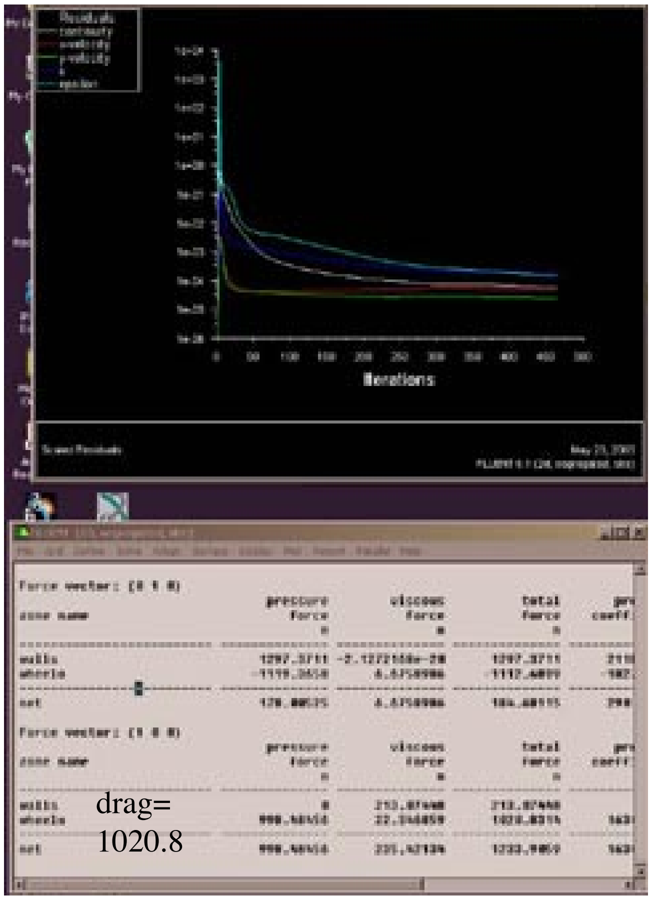


PAGE 5. Next I built and ran a fine mesh, without adaption.. This had 171,597 nodes in its 12 faces and took a ridiculous amount of time to run. (Standard Ke).





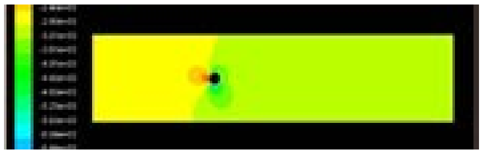
Above: mesh with 400 nodes on circle, and a bazillion elsewhere.

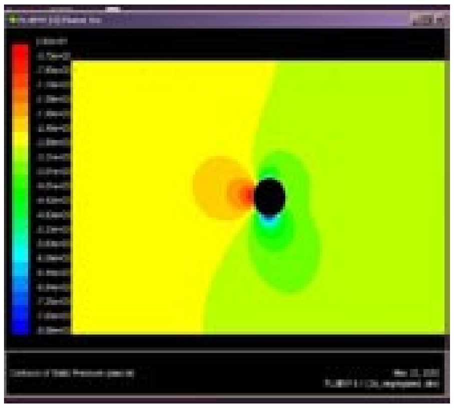


```
Force vector: (0 1 0)
                          pressure        viscous          total      pr
zone name                    force          force          force   coeff
                                 n              n              n
------------------------- -------------- -------------- -------------- -------
walls                        1297.3711 -2.1272158e-20      1297.3711    211
wheels                      -1119.3658      6.6750986     -1112.6899   -182
------------------------- -------------- -------------- -------------- -------
net                          178.00535      6.6750986      184.68115    290

Force vector: (1 0 0)
                          pressure        viscous          total      pr
zone name                    force          force          force   coeff
                                 n              n              n
------------------------- -------------- -------------- -------------- -------
walls                                0      213.07448      213.07448
wheels                       998.46434      22.346859      1020.8315    163
------------------------- -------------- -------------- -------------- -------
net                          998.46434      235.42134      1233.9059    163
```

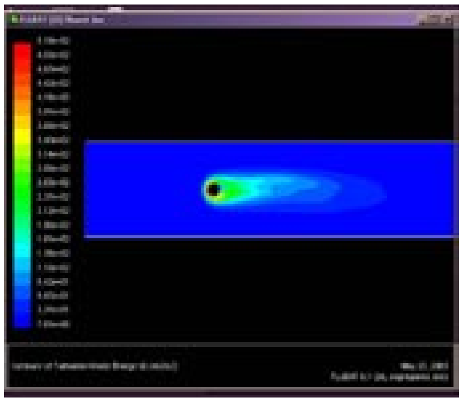
drag=
1020.8

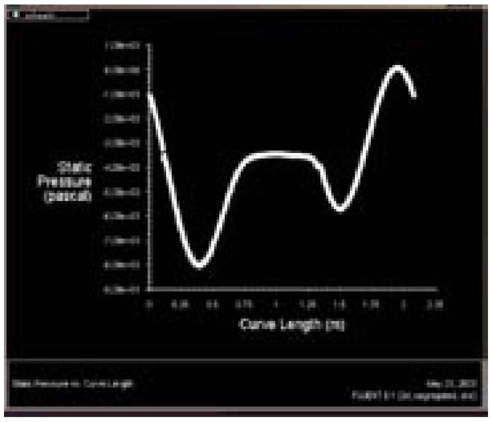
Above: LIFT= 1112.7



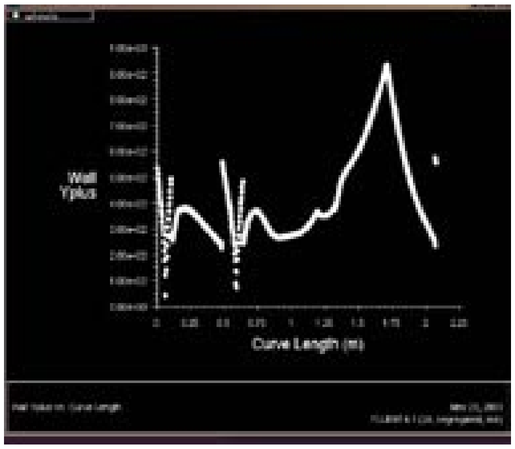


Left and below: pressure and turbulence contours.

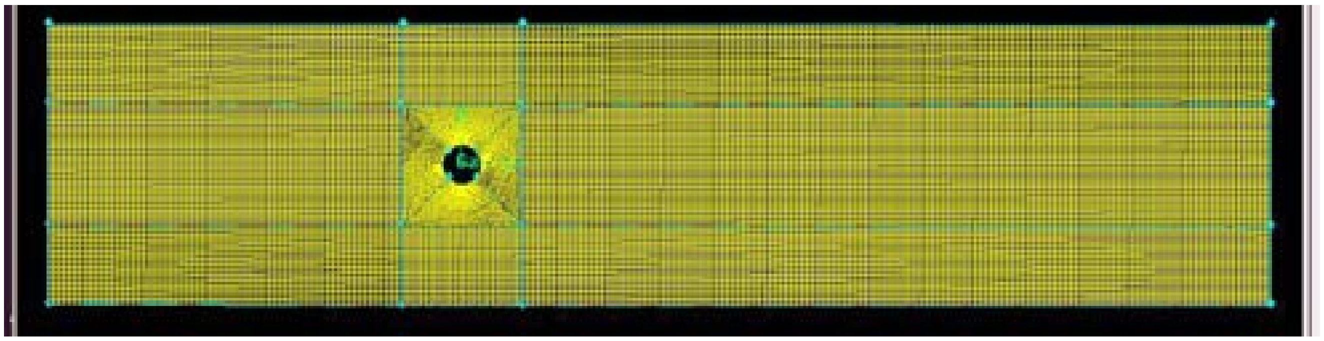


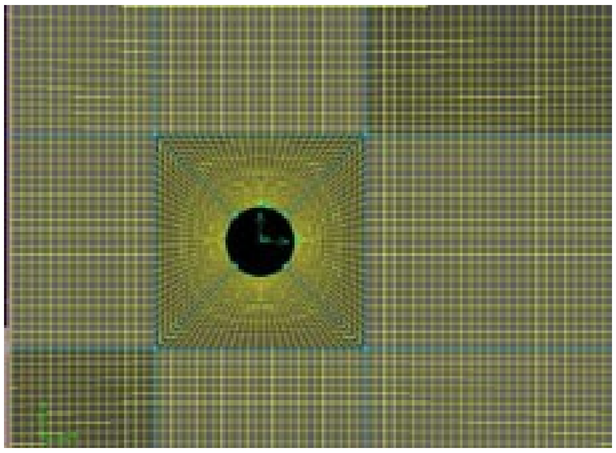


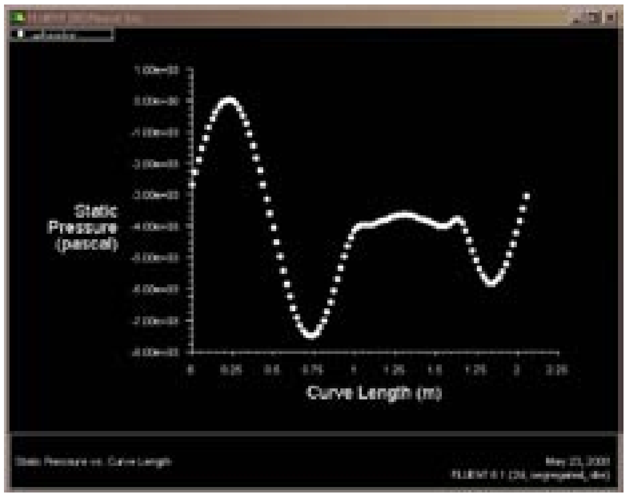
Right: Even with this excessively dense mesh, the y+ values on the wall were still out of range (peak = 1000). Clearly there had to be another way.

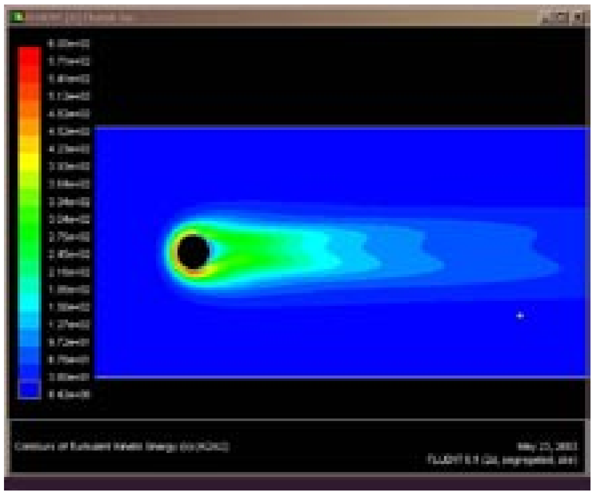


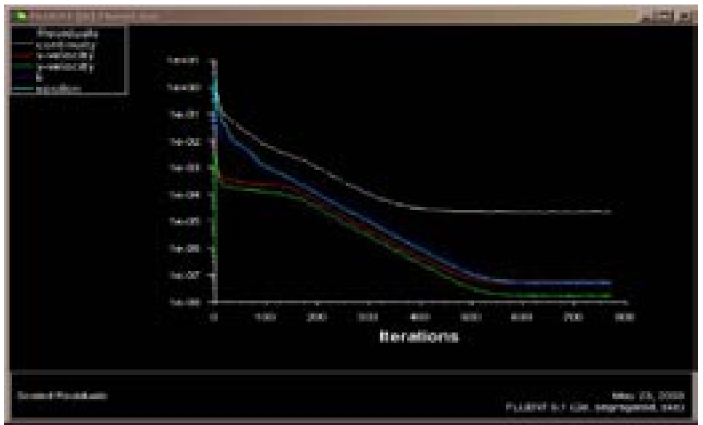
PAGE 6. Next was a coarse quad mesh to do some quick test runs(14560 nodes before adaption). 100 nodes on circle.

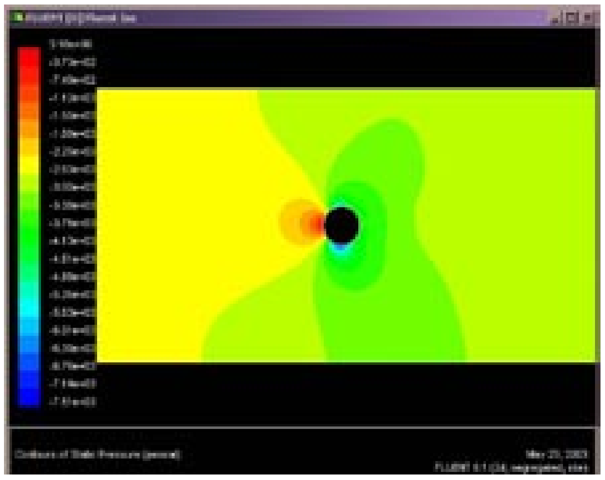


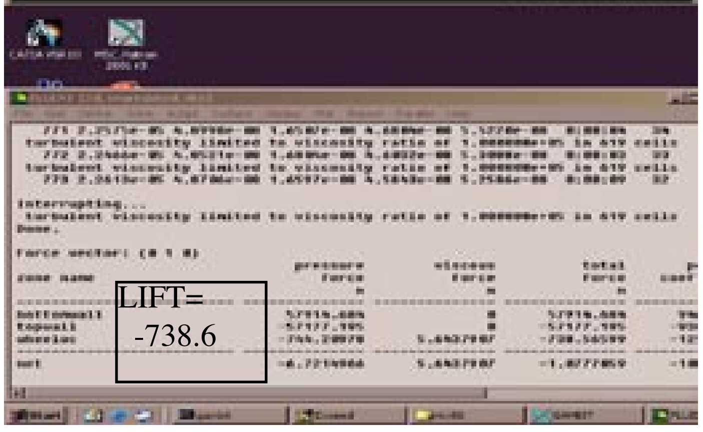





Contours of Turbulent Kinetic Energy (k) (m2/s2)
May 23, 2003
FLUENT 6.1 (2d, segregated, ske)



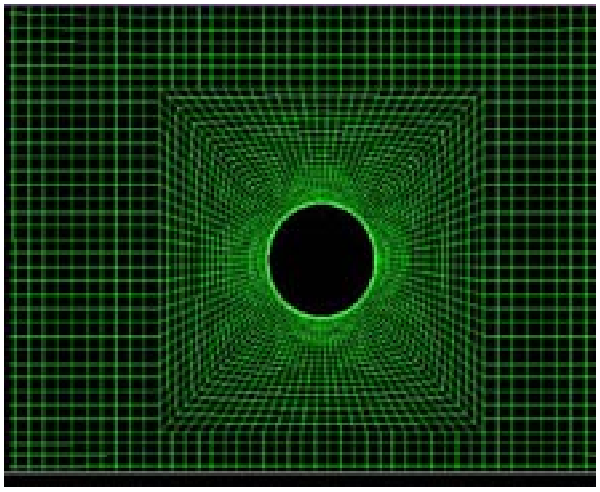

Contours of Static Pressure (pascal)
May 23, 2003
FLUENT 6.1 (2d, segregated, ske)

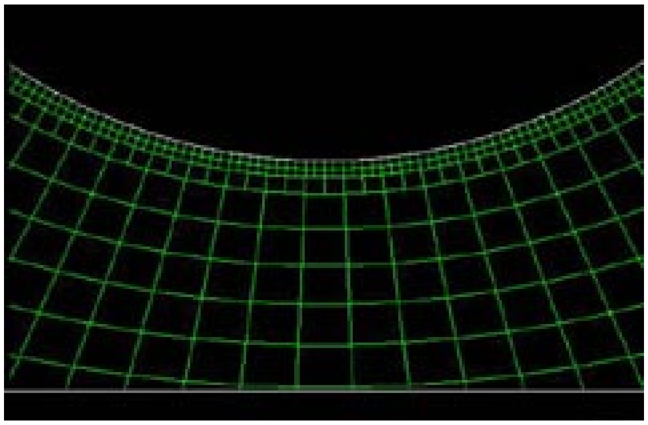


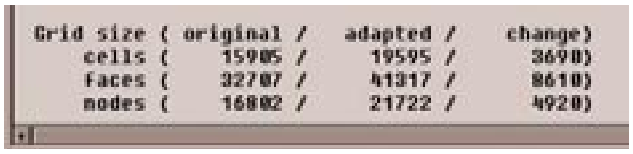
```
   771 [illegible]
 turbulent viscosity limited to viscosity ratio of 1.000000e+05 in 619 cells
   772 [illegible]
 turbulent viscosity limited to viscosity ratio of 1.000000e+05 in 619 cells
   773 [illegible]

interrupting...
 turbulent viscosity limited to viscosity ratio of 1.000000e+05 in 619 cells
Done.

Force vector: (0 1 0)
                              pressure          viscous            total
zone name                        force            force            force
                                     n                n                n
-------------------------  ---------------  ---------------  ---------------
bottomwall                       57914.686                0        57914.686
topwall                         -57177.195                0       -57177.195
[illegible]                     -744.28878        5.6827987       -738.56599
-------------------------  ---------------  ---------------  ---------------
net                             -6.7214066        5.6827987       -1.0277059
```

LIFT=
-738.6

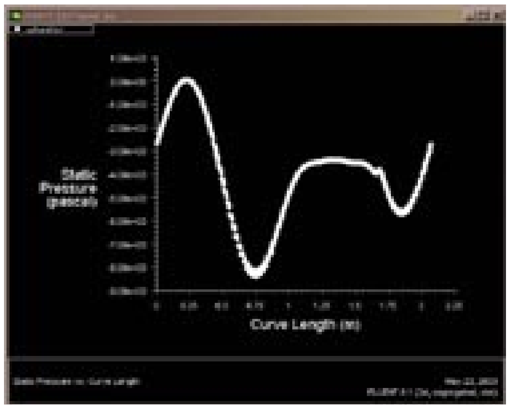
PAGE 7. Same Coarse Mesh after boundary adaption.. Again, this is before I realized that random boundary adaption didn't solve the y+ problem.

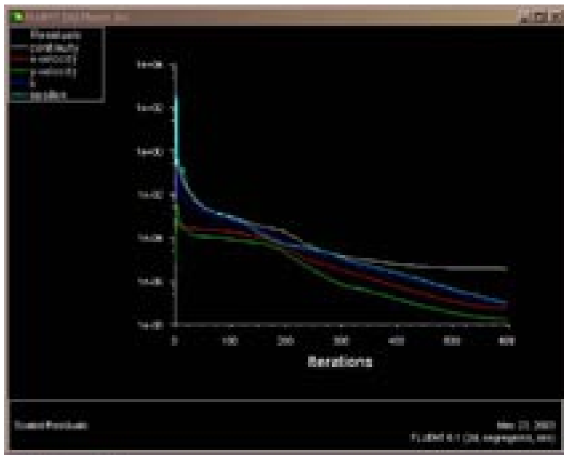


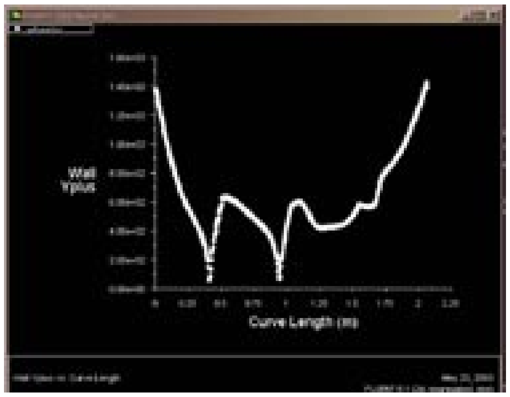




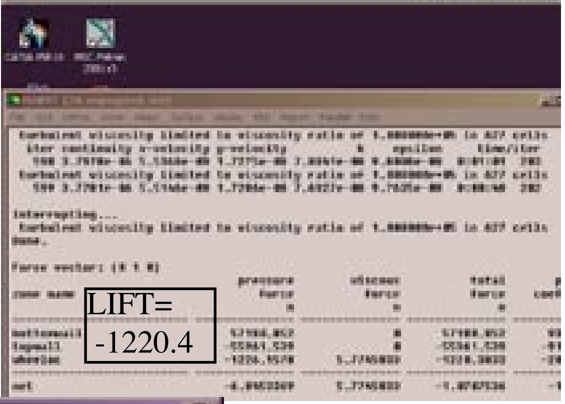
```
Grid size ( original /    adapted /     change)
    cells (     15905 /     19595 /       3690)
    faces (     32707 /     41317 /       8610)
    nodes (     16802 /     21722 /       4920)
```





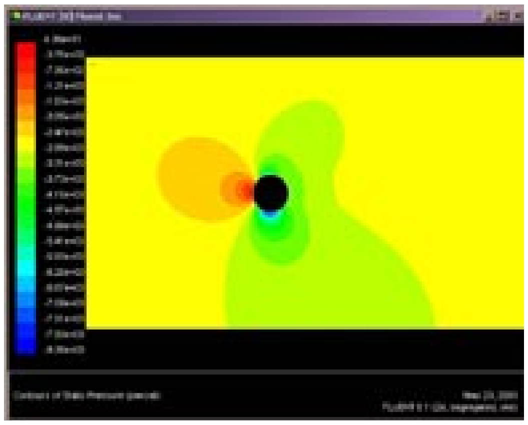


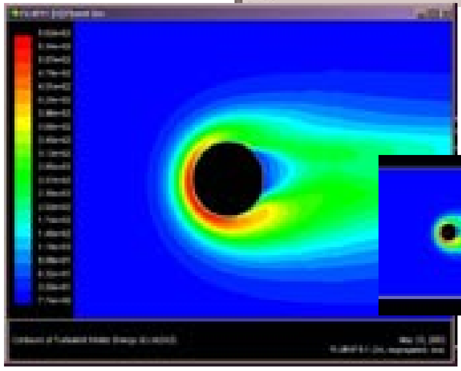
Note how the boundary adaption does not solve the y+ value problem of being too high (max =1600), yet even with the coarse mesh lift values improve after boundary adjustment.

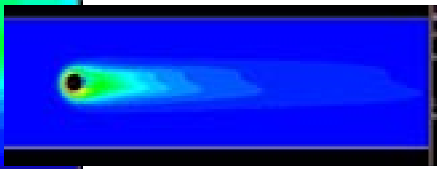


```
 turbulent viscosity limited to viscosity ratio of 1.000000e+05 in 427 cells
  iter continuity x-velocity y-velocity          k     epsilon     time/iter
   598 [illegible]
 turbulent viscosity limited to viscosity ratio of 1.000000e+05 in 427 cells
   599 [illegible]

interrupting...
 turbulent viscosity limited to viscosity ratio of 1.000000e+05 in 427 cells
Done.

Force vector: (0 1 0)
                              pressure        viscous          total
zone name                        force          force          force
                                     n              n              n
----------------------  --------------  -------------  -------------
bottomwall                   57988.852              0      57988.852
topwall                     -55961.536              0     -55961.536
wheel                       -1226.1578      5.7745833     -1220.3833
----------------------  --------------  -------------  -------------
net                         -6.8533369      5.7745833     -1.0787536
```

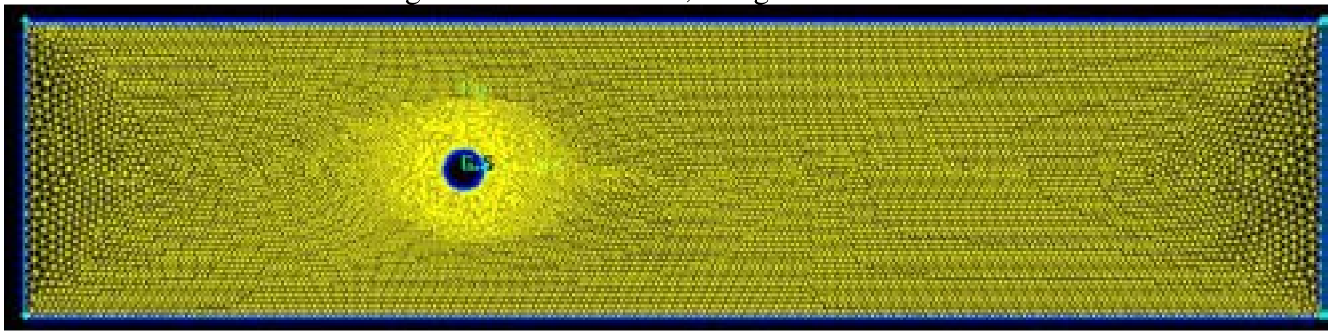
LIFT=
-1220.4



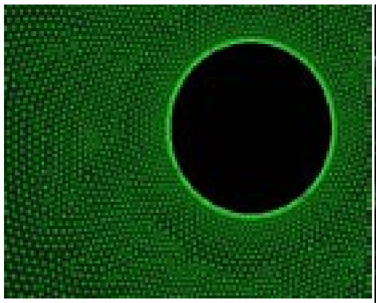


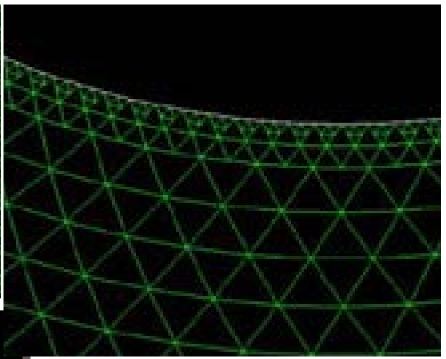


PAGE 8. Next I ran some triangle meshes .Version 1, using a standard Ke model.

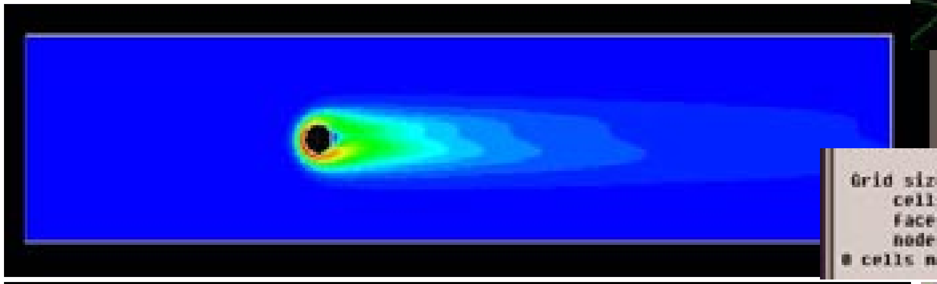


Original Gambit Mesh: 100 mesh seeds on circle, 200 bi-exponent on top and bottom walls, 80 front and back walls. 20452 nodes before boundary adaption.

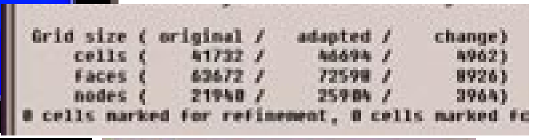




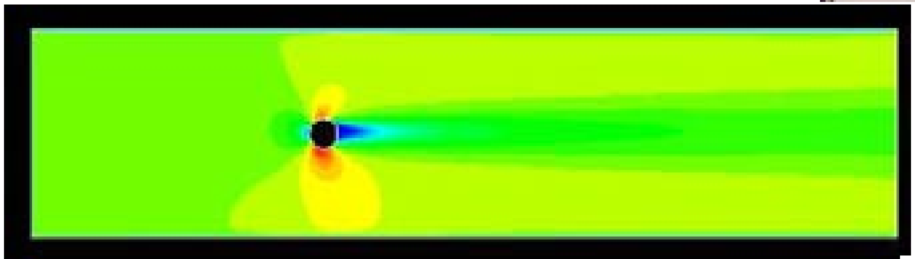
Below: Turbulence, Velocity and Pressure contours.

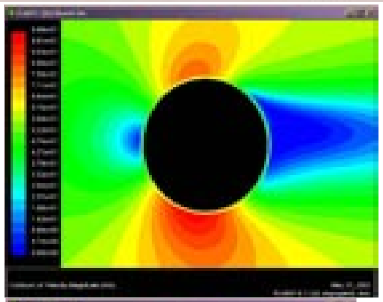


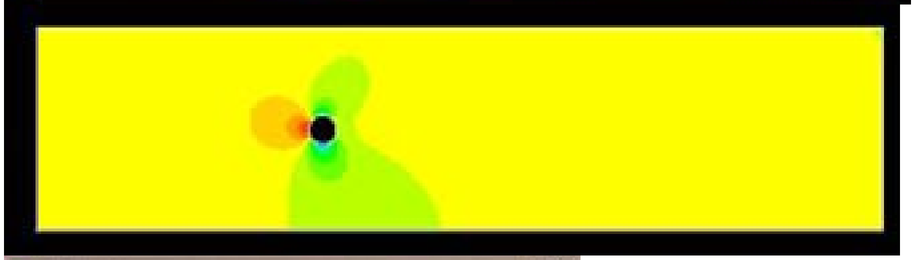
After Boundary Adaption: 25904 nodes.

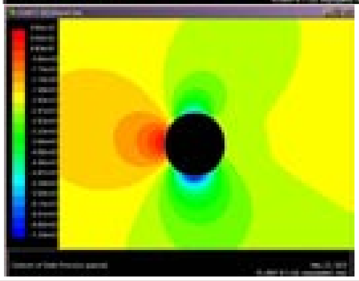


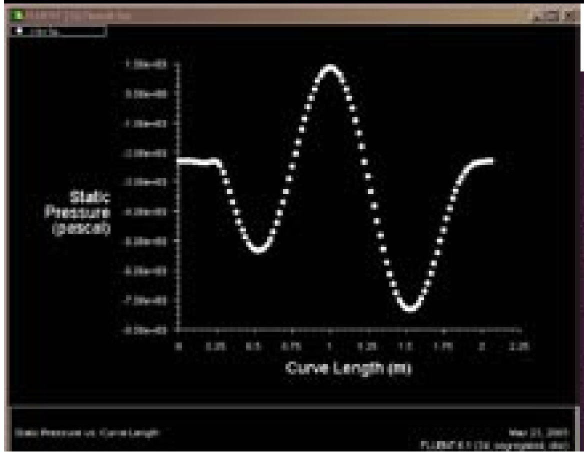
```
 Grid size ( original /    adapted /     change)
     cells (     41732 /      46694 /      4962)
     faces (     63672 /      72598 /      8926)
     nodes (     21940 /      25904 /      3964)
0 cells marked for refinement, 0 cells marked fc
```

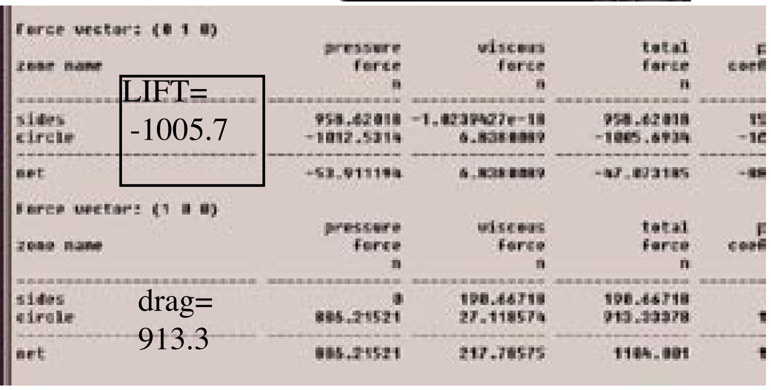










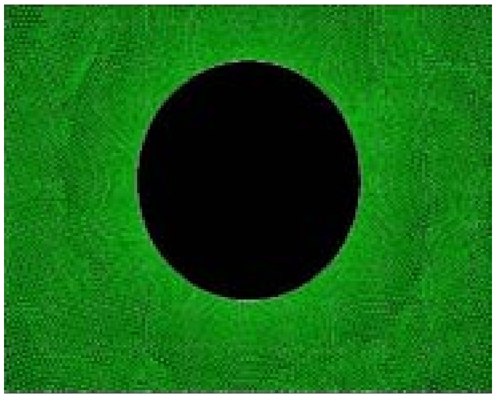


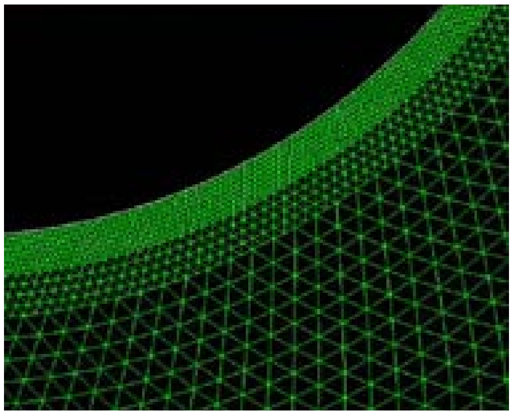
```
Force vector: (0 1 0)
                                   pressure          viscous            total       p
zone name                             force            force            force    coef
                                          n                n                n
-------------------------------- ---------------- ---------------- ---------------- --------
sides                             958.62018 -1.0239427e-18        958.62018      15
circle                           -1012.5314        6.8380089       -1005.6934     -10
-------------------------------- ---------------- ---------------- ---------------- --------
net                              -53.911194        6.8380089       -47.073185     -00

Force vector: (1 0 0)
                                   pressure          viscous            total       p
zone name                             force            force            force    coef
                                          n                n                n
-------------------------------- ---------------- ---------------- ---------------- --------
sides                                     0        190.66718        190.66718
circle                            886.21521        27.118574        913.33378       1
-------------------------------- ---------------- ---------------- ---------------- --------
net                               886.21521        217.78575         1104.001       1
```

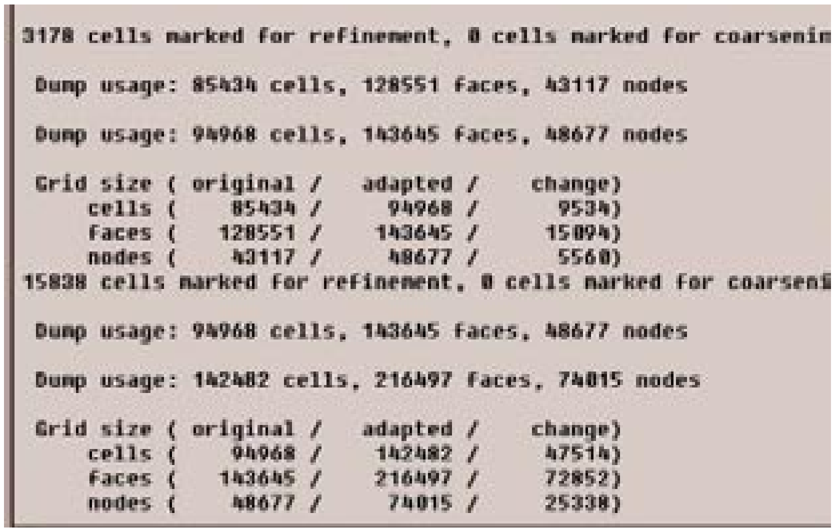
LIFT= -1005.7

drag= 913.3

PAGE 9. Triangle Mesh Version 2. 200 nodes on circle, 225 bi exponent nodes on sides, 75 front, back. Standard Ke model.

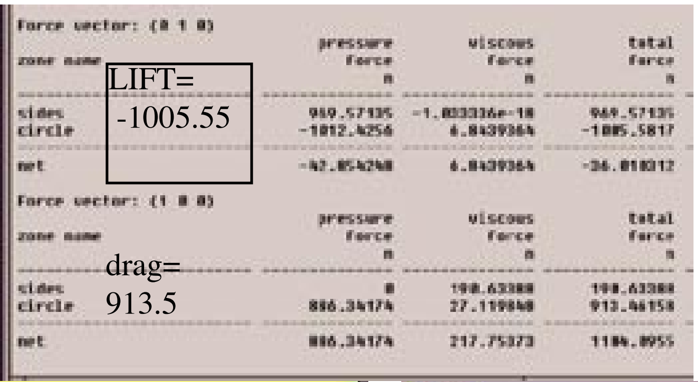






```
3178 cells marked for refinement, 0 cells marked for coarsenin

 Dump usage: 85434 cells, 128551 faces, 43117 nodes

 Dump usage: 94968 cells, 143645 faces, 48677 nodes

 Grid size ( original /   adapted /    change)
     cells (     85434 /     94968 /      9534)
     faces (    128551 /    143645 /     15094)
     nodes (     43117 /     48677 /      5560)
15838 cells marked for refinement, 0 cells marked for coarseni

 Dump usage: 94968 cells, 143645 faces, 48677 nodes

 Dump usage: 142482 cells, 216497 faces, 74015 nodes

 Grid size ( original /   adapted /    change)
     cells (     94968 /    142482 /     47514)
     faces (    143645 /    216497 /     72852)
     nodes (     48677 /     74015 /     25338)
```

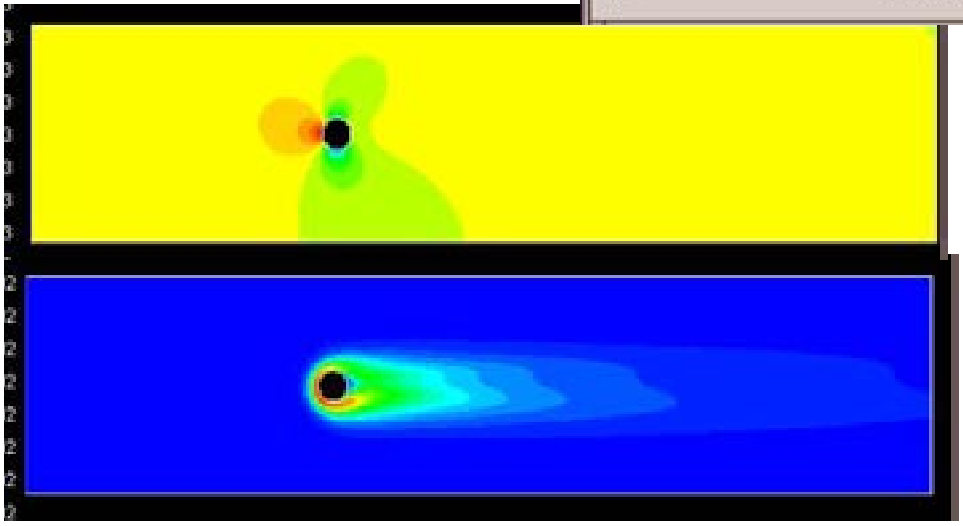
Above: experiment with double boundary adaption.

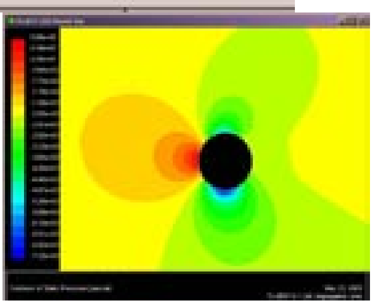


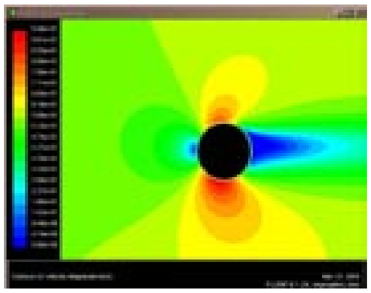
```
Force vector: (0 1 0)
                                  pressure        viscous          total
zone name                            force          force          force
                                         n              n              n
------------------------- -------------- -------------- --------------
sides                            969.57135  -1.033336e-18      969.57135
circle                          -1012.4256      6.8439364     -1005.5817
------------------------- -------------- -------------- --------------
net                             -42.854248      6.8439364     -36.010312

Force vector: (1 0 0)
                                  pressure        viscous          total
zone name                            force          force          force
                                         n              n              n
------------------------- -------------- -------------- --------------
sides                                    0      190.63388      190.63388
circle                           886.34174      27.119848      913.46158
------------------------- -------------- -------------- --------------
net                              886.34174      217.75373      1104.0955
```

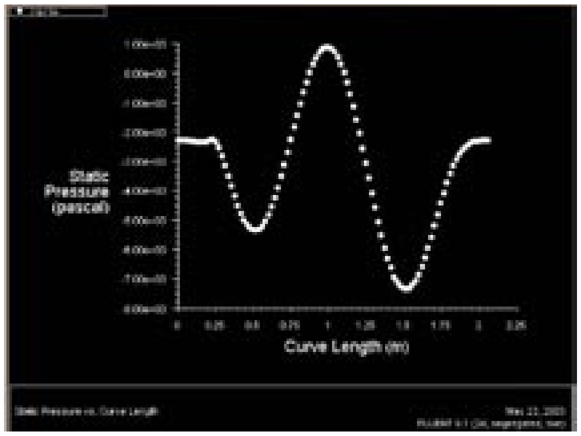
LIFT=
-1005.55

drag=
913.5

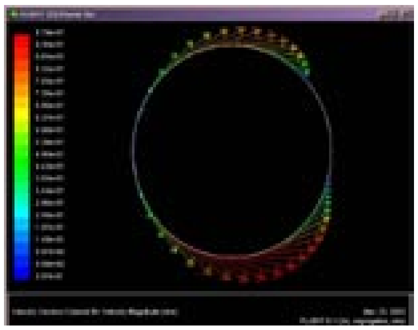




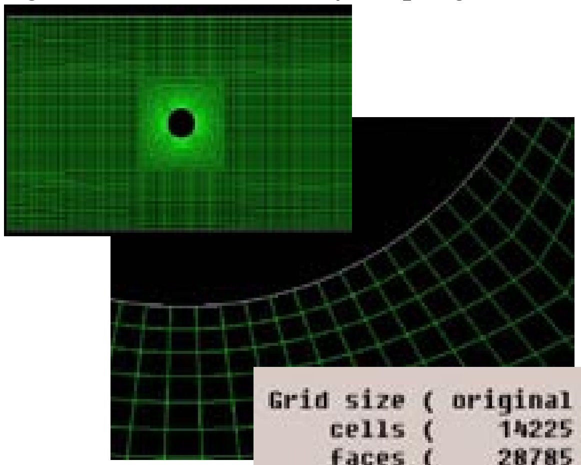


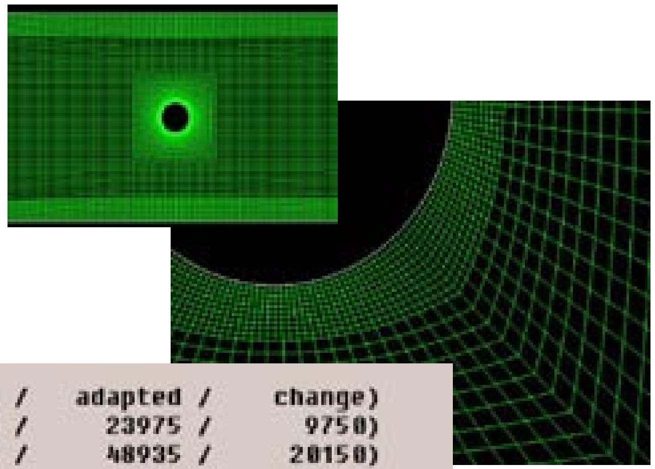


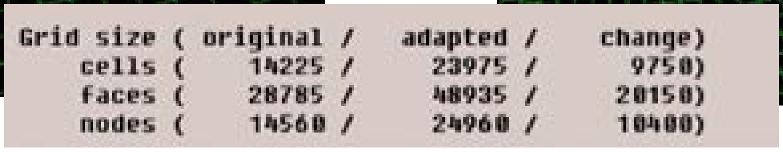
Left, above and right: Version 2 Triangular mesh plots and contours. Not much difference in the visuals from the quad mesh, though the lift is less for the equivalent mesh parameters.



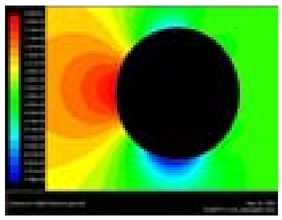
PAGE 10. More Experiments with a new Coarse Quad mesh and standard Ke model. Again we see more signs of a novice randomly adapting meshes using the boundary command.

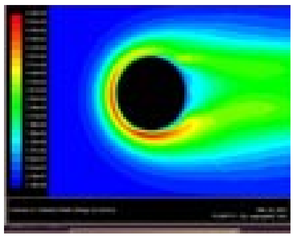


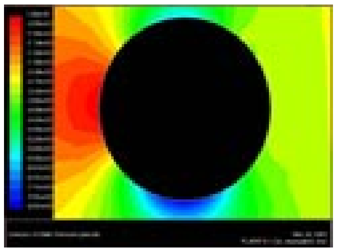


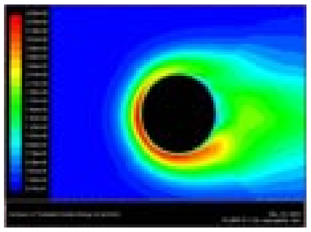


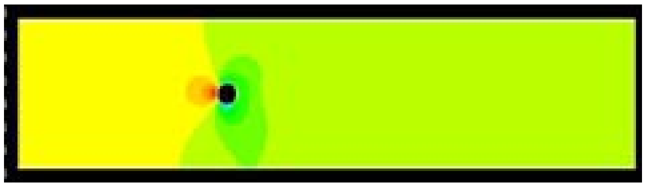
```
Grid size ( original /   adapted /    change)
    cells (     14225 /     23975 /      9750)
    faces (     28785 /     48935 /     20150)
    nodes (     14560 /     24960 /     10400)
```

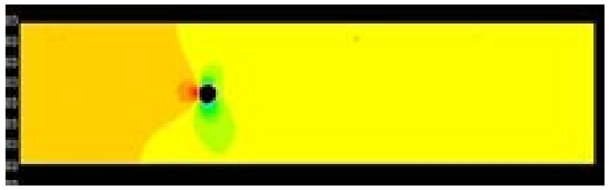


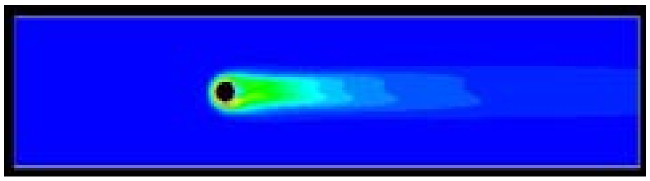


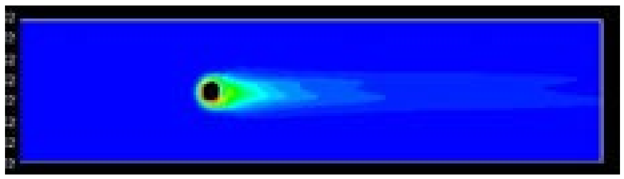


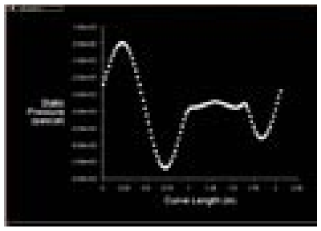


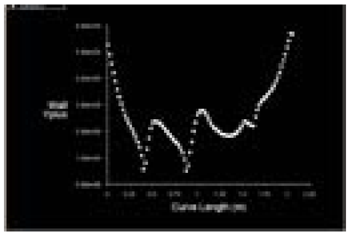


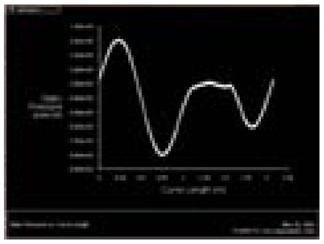


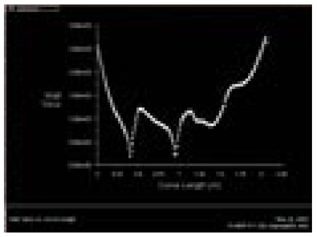






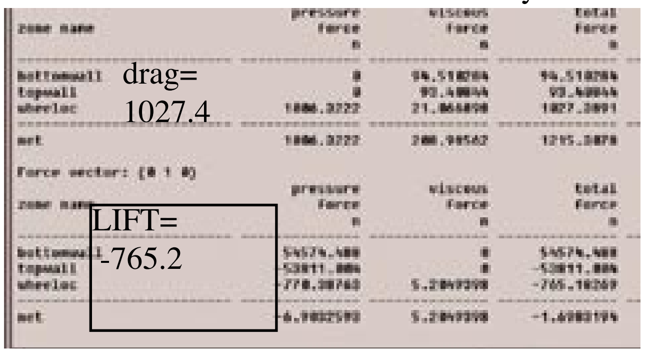






Above: max y+ = 6000

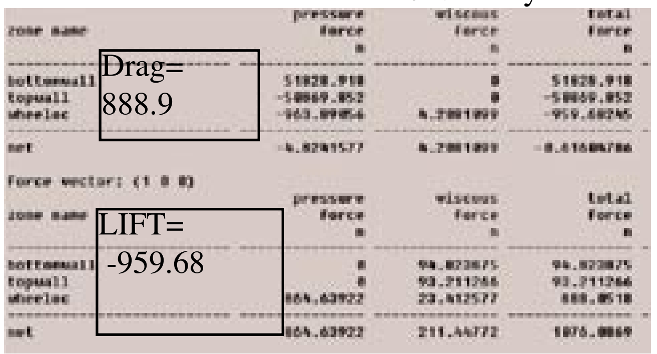
Above max y+ = 3000



drag=
1027.4

LIFT=
-765.2

```
                                   pressure         viscous           total
zone name                             force           force           force
                                          n               n               n
------------------------------ ---------------- --------------- ---------------
bottomwall                                0       94.518204       94.518204
topwall                                   0        93.40844        93.40844
wheeloc                           1006.3222       21.066898       1027.3891
------------------------------ ---------------- --------------- ---------------
net                               1006.3222       208.99562       1215.3078

Force vector: (0 1 0)
                                   pressure         viscous           total
zone name                             force           force           force
                                          n               n               n
------------------------------ ---------------- --------------- ---------------
bottomwall                        54574.488               0       54574.488
topwall                          -53811.806               0      -53811.806
wheeloc                          -770.38763       5.2049398      -765.18269
------------------------------ ---------------- --------------- ---------------
net                              -6.9032593       5.2049398      -1.6983194
```



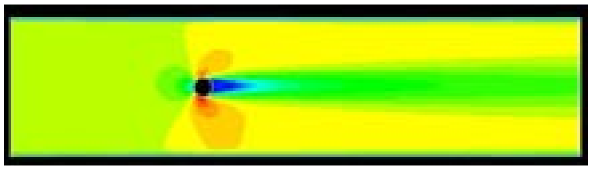
Drag=
888.9

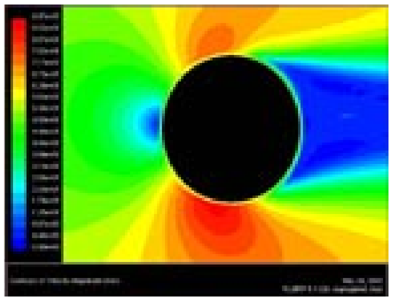
LIFT=
-959.68

```
                                   pressure         viscous           total
zone name                             force           force           force
                                          n               n               n
------------------------------ ---------------- --------------- ---------------
bottomwall                        51828.918               0       51828.918
topwall                          -50869.852               0      -50869.852
wheeloc                          -963.89856       4.2001099      -959.68245
------------------------------ ---------------- --------------- ---------------
net                              -4.8241577       4.2001099     -0.61606786

Force vector: (1 0 0)
                                   pressure         viscous           total
zone name                             force           force           force
                                          n               n               n
------------------------------ ---------------- --------------- ---------------
bottomwall                                0       94.823875       94.823875
topwall                                   0       93.211266       93.211266
wheeloc                           864.63922       23.412577        888.0518
------------------------------ ---------------- --------------- ---------------
net                               864.63922       211.44772       1076.0869
```

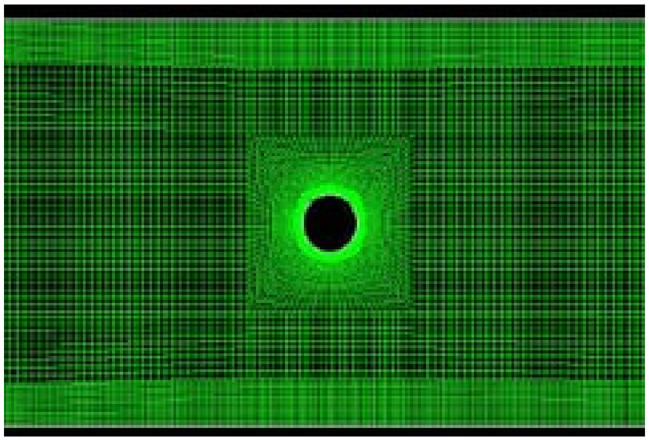
Later I did the exact same boundary adaption comparison with my final medium quad meshes. Lift went from 1024.7 to 1314.7, maximum y+ values decreased from 3500 to 1700, and number of nodes increased from 31368 to 33736. The y+ adaption proved to work better in the end.

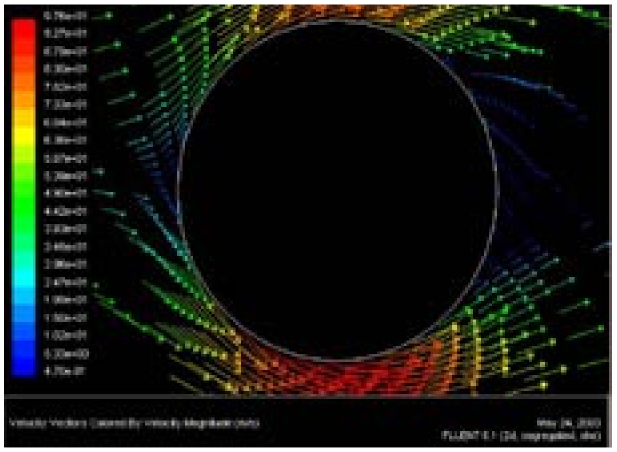
PAGE 11: With the same enhanced coarse quad grid model, I tried the wall function and k omega models and compared results. About this time, I realized that the mesh adaptions were probably adversely affecting my overall results because of the sudden element gradient change that occurs.



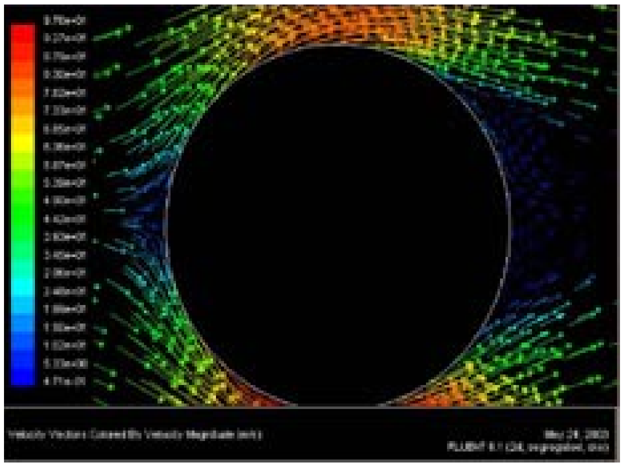


Velocity contours with standard k-e model

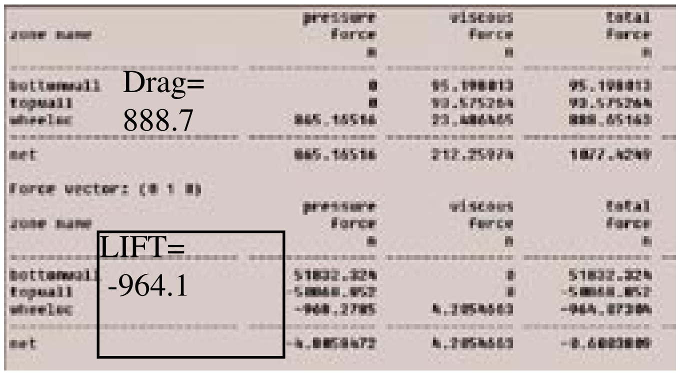




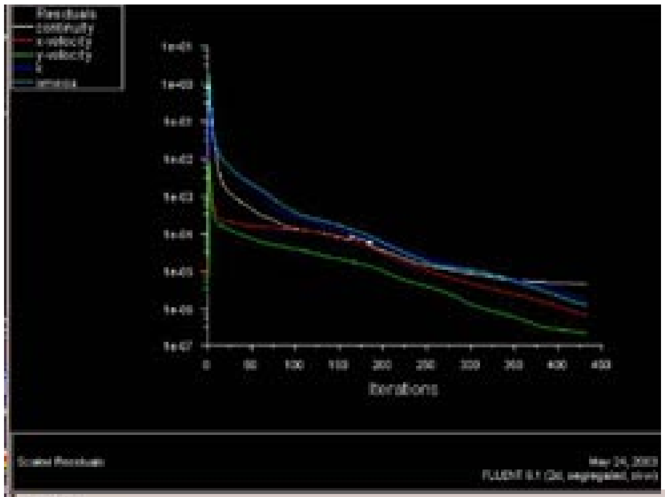
Standard K-e model velocity vectors

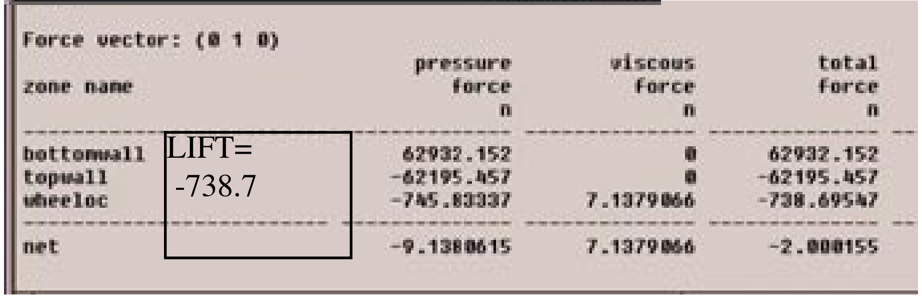


Enhanced Wall Function velocity vectors. Below: forces from K-e model with enhanced wall function.



```
                                pressure        viscous          total
zone name                          force          force          force
                                       n              n              n
-------------------------  -------------  -------------  -------------
bottomwall   Drag=                     0      95.198813      95.198813
topwall                                0      93.575264      93.575264
wheeloc      888.7             865.16516      23.486465      888.65163
-------------------------  -------------  -------------  -------------
net                            865.16516      212.25974      1077.4249

Force vector: (0 1 0)
                                pressure        viscous          total
zone name                          force          force          force
                 LIFT=                 n              n              n
-------------------------  -------------  -------------  -------------
bottomwall                     51832.324              0      51832.324
topwall          -964.1       -50860.852              0     -50860.852
wheeloc                        -968.2785      4.2054663     -964.07304
-------------------------  -------------  -------------  -------------
net                           -4.8058472      4.2054663     -0.6003809
```

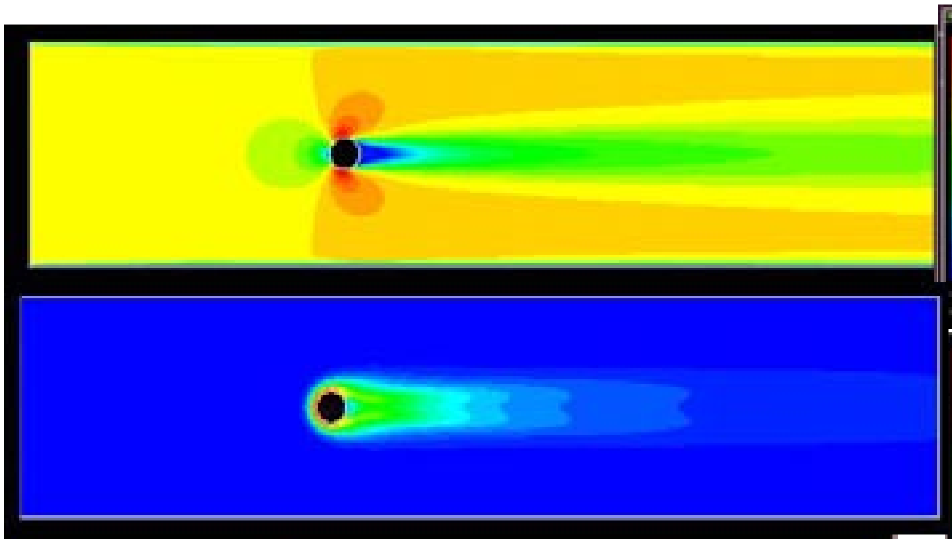


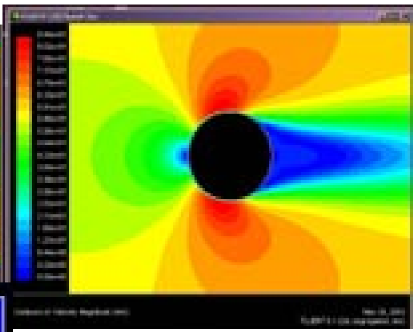


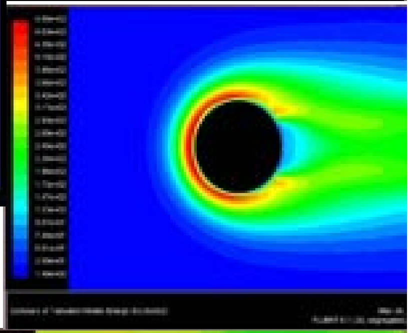
```
Force vector: (0 1 0)
                                pressure        viscous          total
zone name                          force          force          force
                                       n              n              n
-------------------------  -------------  -------------  -------------
bottomwall   LIFT=             62932.152              0      62932.152
topwall       -738.7          -62195.457              0     -62195.457
wheeloc                       -745.83337      7.1379066     -738.69547
-------------------------  -------------  -------------  -------------
net                           -9.1380615      7.1379066      -2.000155
```

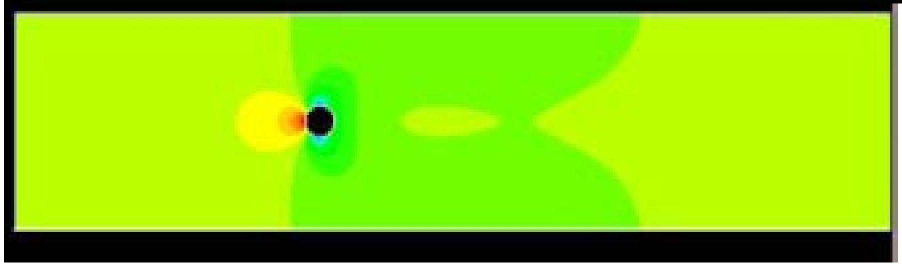
Left: Iteration plot and K Omega results with boundary adapted coarse quad grid.

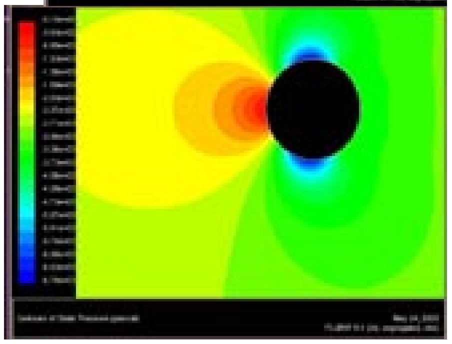
Page 12:For reference, another run with zero rotation on the circle. This is with the original medium quad mesh, using standard K-e second order solution.



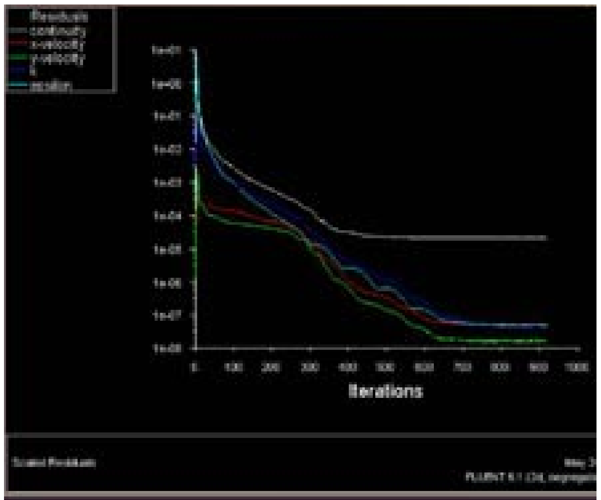


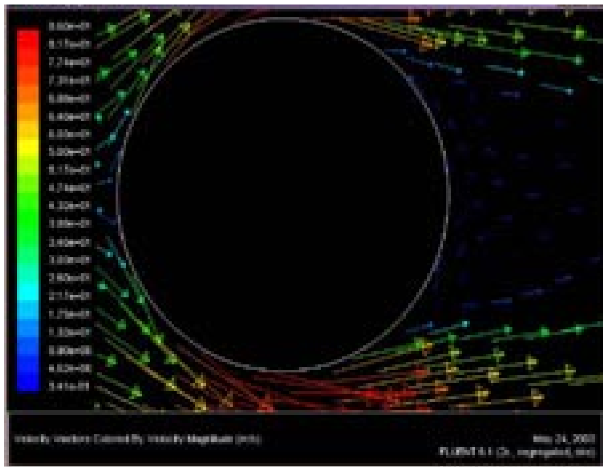


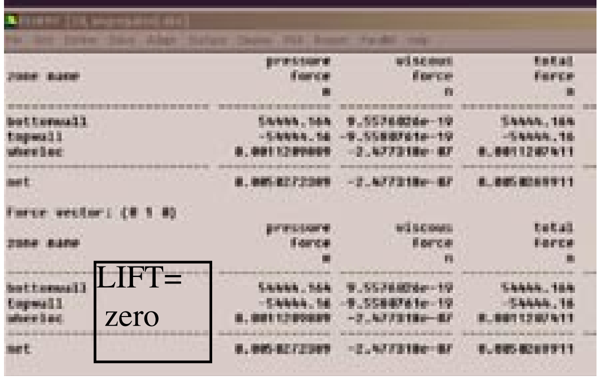




An odd pressure pattern above. Perhaps an image of a turbulent wake (zero rotation on circle).

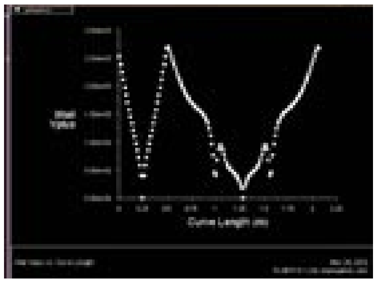


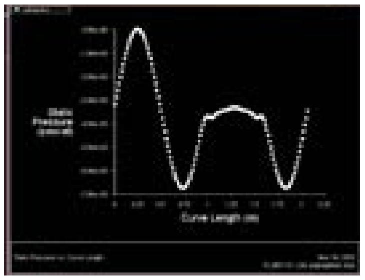




```
                               pressure      viscous        total
zone name                         force        force        force
                                      n            n            n
------------------------------ ------------ ------------ ------------
bottomwall                    54444.164  9.5576020e-19    54444.164
topwall                       -54444.16 -9.5580761e-19    -54444.16
wheelac                    0.0011200009 -2.4773180e-07 0.0011207411
------------------------------ ------------ ------------ ------------
net                        0.0050272309 -2.4773180e-07 0.0050269911

force vector: (0 1 0)
                               pressure      viscous        total
zone name                         force        force        force
                                      n            n            n
------------------------------ ------------ ------------ ------------
bottomwall                    54444.164  9.5576020e-19    54444.164
topwall                       -54444.16 -9.5580761e-19    -54444.16
wheelac                    0.0011200009 -2.4773180e-07 0.0011207411
------------------------------ ------------ ------------ ------------
net                        0.0050272309 -2.4773180e-07 0.0050269911
```

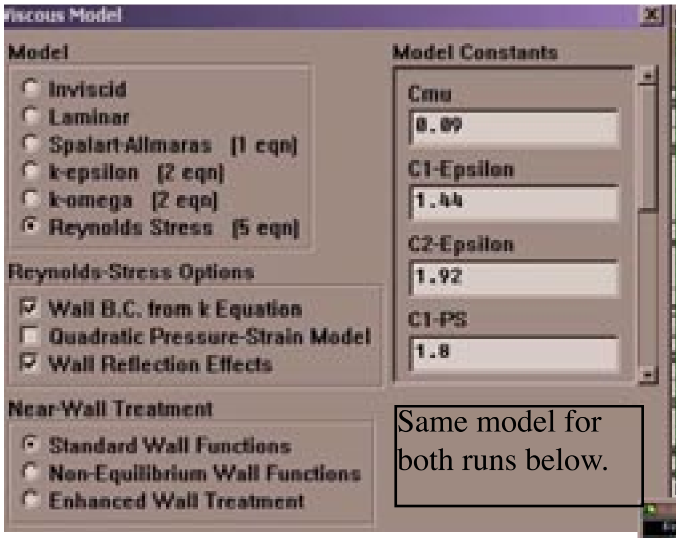
LIFT= zero





Max y+ = 3000

PAGE 13: Here's something interesting. I used a boundary adapted medium quad mesh for the first example, then a non-adapted grid for the second. The differences using a Reynold's Stress model were significant.


Viscous Model

Model
- ( ) Inviscid
- ( ) Laminar
- ( ) Spalart-Allmaras [1 eqn]
- ( ) k-epsilon [2 eqn]
- ( ) k-omega [2 eqn]
- [✓ Reynolds Stress [5 eqn]]

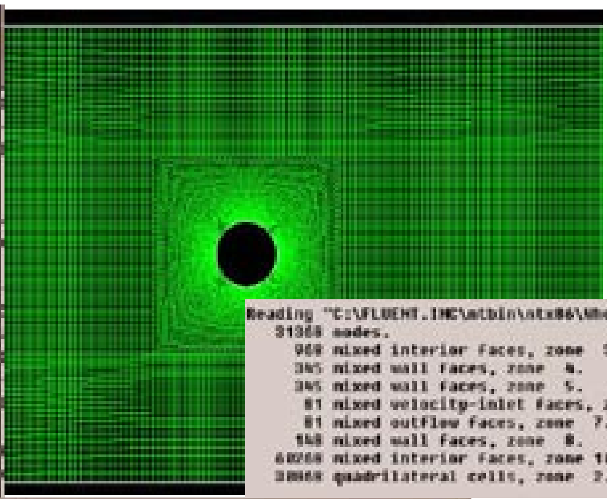
Reynolds-Stress Options
- [✓ Wall B.C. from k Equation]
- Quadratic Pressure-Strain Model
- [✓ Wall Reflection Effects]

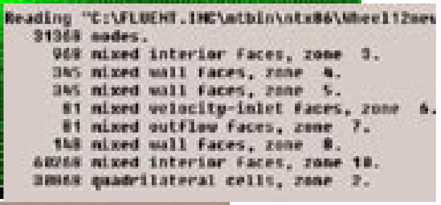
Near-Wall Treatment
- [✓ Standard Wall Functions]
- ( ) Non-Equilibrium Wall Functions
- ( ) Enhanced Wall Treatment

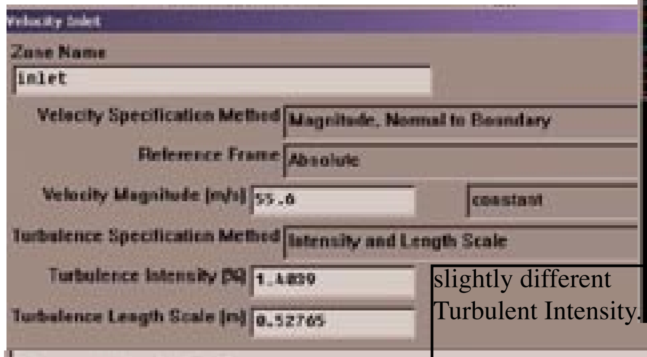
Model Constants

Cmu	0.09
C1-Epsilon	1.44
C2-Epsilon	1.92
C1-PS	1.8

Same model for both runs below.

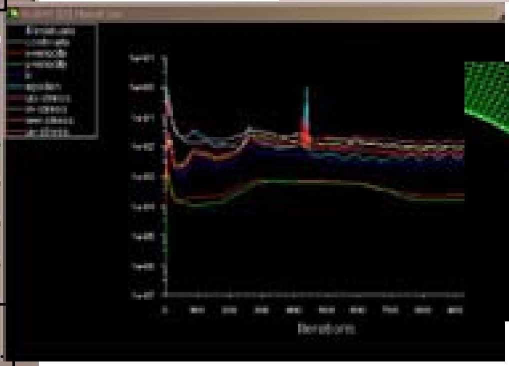


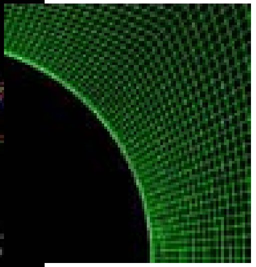

```
Reading "C:\FLUENT.INC\ntbin\ntx86\Wheel12new
   31368 nodes.
     960 mixed interior faces, zone  3.
     345 mixed wall faces, zone  4.
     345 mixed wall faces, zone  5.
      81 mixed velocity-inlet faces, zone  6.
      81 mixed outflow faces, zone  7.
     148 mixed wall faces, zone  8.
   60260 mixed interior faces, zone 10.
   30060 quadrilateral cells, zone  2.
```


Velocity Inlet

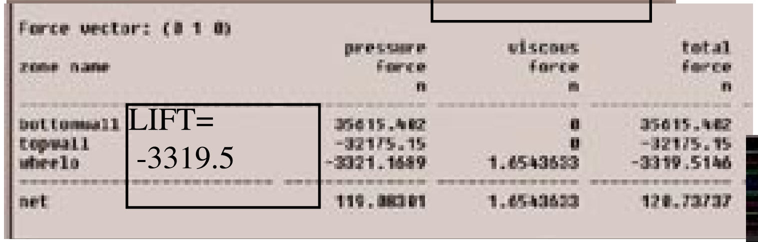
Zone Name	inlet
Velocity Specification Method	Magnitude, Normal to Boundary
Reference Frame	Absolute
Velocity Magnitude (m/s)	55.6 constant
Turbulence Specification Method	Intensity and Length Scale
Turbulence Intensity (%)	1.4839
Turbulence Length Scale (m)	0.52765

slightly different Turbulent Intensity.

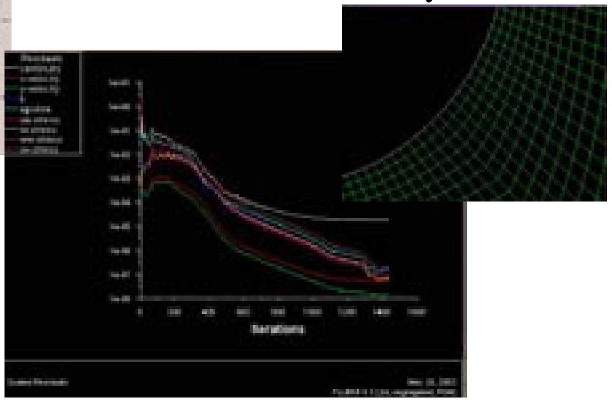


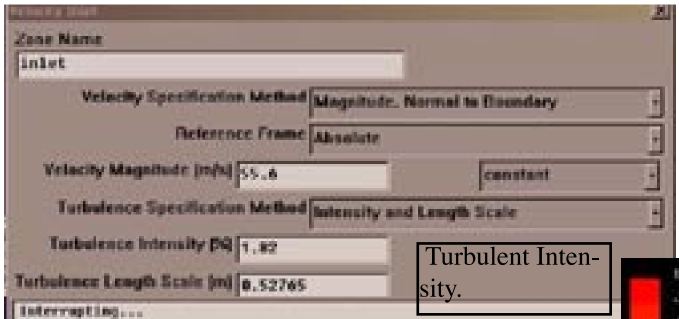


Above run with boundary adapted mesh.. Below with unadapted mesh and correct turbulence boundary conditions.


```
Force vector: (0 1 0)
                                 pressure          viscous            total
zone name                           force            force            force
                                        n                n                n
-------------------------- ---------------- ---------------- ----------------
bottomwall                      35615.402                0        35615.402
topwall                         -32175.15                0        -32175.15
wheelo                         -3321.1689        1.6543633       -3319.5146
-------------------------- ---------------- ---------------- ----------------
net                             119.08301        1.6543633        120.73737
```

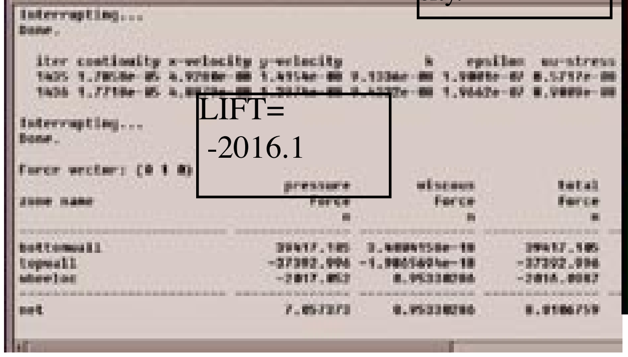
LIFT= -3319.5



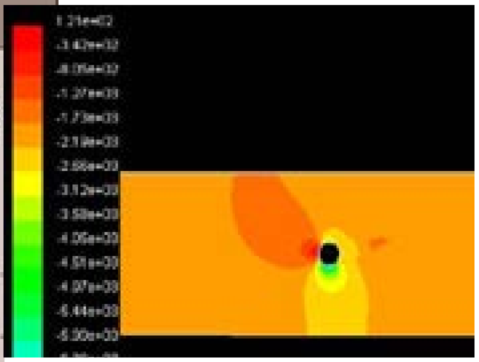


Zone Name	inlet
Velocity Specification Method	Magnitude, Normal to Boundary
Reference Frame	Absolute
Velocity Magnitude (m/s)	55.6 constant
Turbulence Specification Method	Intensity and Length Scale
Turbulence Intensity (%)	1.82
Turbulence Length Scale (m)	0.52765

Turbulent Intensity.


```
Interrupting...
Done.

  iter continuity x-velocity y-velocity          k    epsilon  uu-stress
  1635 1.7050e-05 4.9700e-08 1.4154e-08 9.1330e-08 1.9005e-07 8.5757e-08
  1636 1.7710e-05 [illegible] [illegible] [illegible] 1.9042e-07 8.9009e-08

Interrupting...
Done.

Force vector: (0 1 0)
                                 pressure          viscous            total
zone name                           force            force            force
                                        n                n                n
-------------------------- ---------------- ---------------- ----------------
bottomwall                      39417.105  3.4004150e-10        39417.105
topwall                        -37392.096 -1.9005409e-10       -37392.096
wheelos                        -2017.053       0.95330206       -2016.0987
-------------------------- ---------------- ---------------- ----------------
net                             7.057373       0.95330206        8.0106759
```

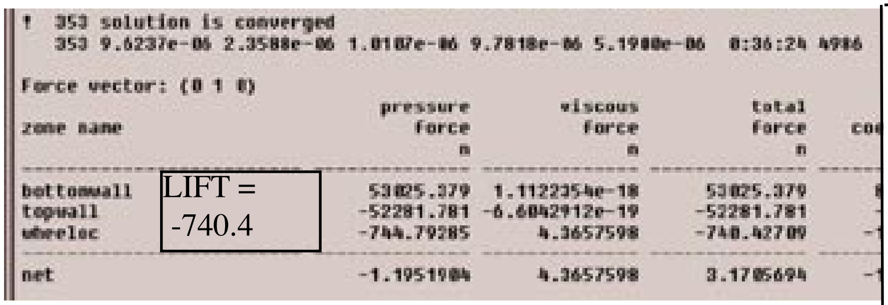
LIFT= -2016.1



From this it became clear how sensitive some models were to initial conditions and mesh parameters.

Left: an interesting bubble behind the cylinder in the pressure contour from this run.

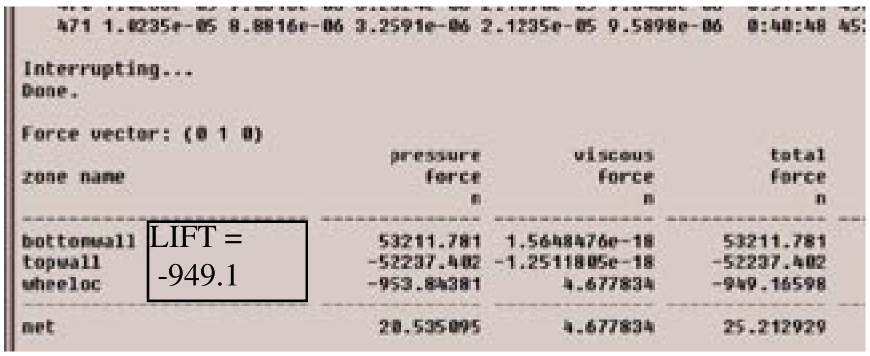
PAGE 14: At this point I redid all my meshes and ended up with semi-final versions which were much more consistent in terms of grid size (more square around circle), and pattern in the adjoining faces (see main results). With these meshes, I performed more iterations.



```
!  353 solution is converged
   353 9.6237e-06 2.3588e-06 1.0107e-06 9.7818e-06 5.1900e-06  0:36:24 4986

Force vector: (0 1 0)
                              pressure          viscous            total
zone name                        force            force            force     co
                                     n                n                n
------------------------------ --------------- ---------------- ---------------- -----
bottomwall                     53825.379  1.1122354e-18        53825.379
topwall                       -52281.781 -6.6042912e-19       -52281.781
wheeloc                       -744.79285      4.3657598       -740.42709
------------------------------ --------------- ---------------- ---------------- -----
net                           -1.1951904      4.3657598        3.1705694
```

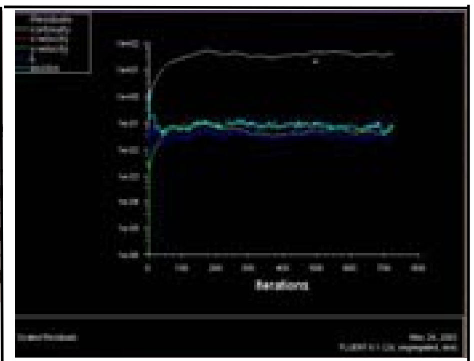
LIFT =
-740.4

Results of runs with unadapted final quad meshes w/ first order SIMPLE discretiation. Above: Run with medium mesh. Below: First order SIMPLE run with fine mesh.

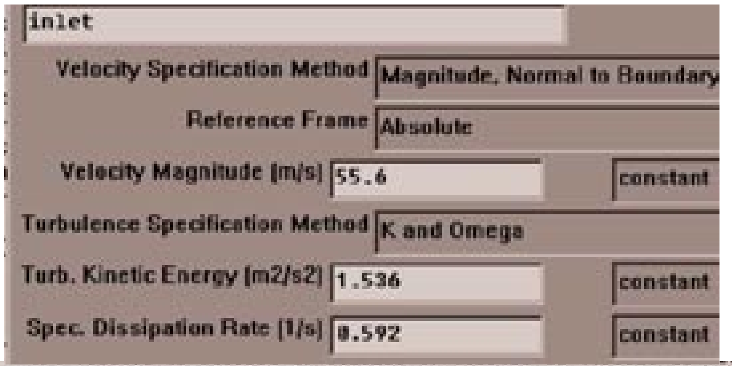


```
   471 1.0235e-05 8.8816e-06 3.2591e-06 2.1235e-05 9.5898e-06  0:40:48 45

Interrupting...
Done.

Force vector: (0 1 0)
                              pressure          viscous            total
zone name                        force            force            force
                                     n                n                n
------------------------------ --------------- ---------------- ---------------- ---
bottomwall                     53211.781  1.5648476e-18        53211.781
topwall                       -52237.402 -1.2511805e-18       -52237.402
wheeloc                       -953.84381       4.677834       -949.16598
------------------------------ --------------- ---------------- ---------------- ---
net                            20.535095       4.677834        25.212929
```

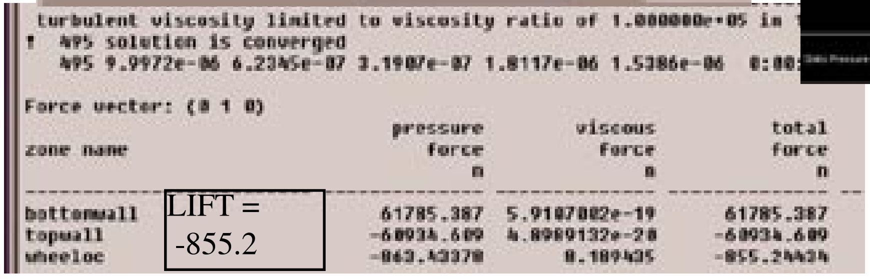
LIFT =
-949.1



Above: Another botched attempt using the PISO discretiation. I'm not sure what went wrong here. PISO is recommended for transient flow calculations with a large time step, so that could be the problem (although I did not check the under relaxation factors, which I assumed would be set a recommended default level).

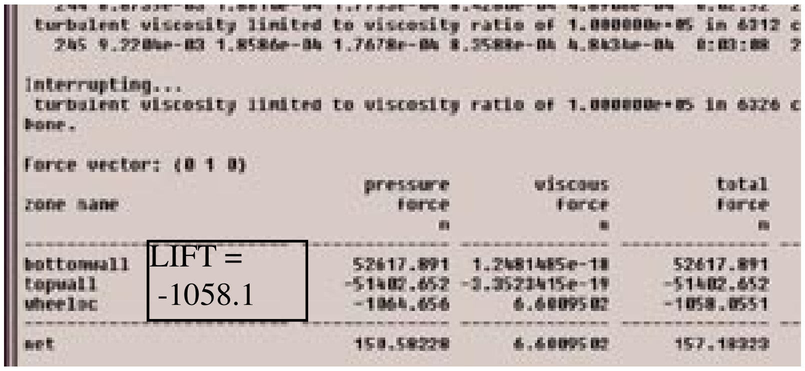


```
inlet
Velocity Specification Method   Magnitude, Normal to Boundary
Reference Frame                 Absolute
Velocity Magnitude (m/s)        55.6       constant
Turbulence Specification Method K and Omega
Turb. Kinetic Energy (m2/s2)    1.536      constant
Spec. Dissipation Rate (1/s)    0.592      constant
```



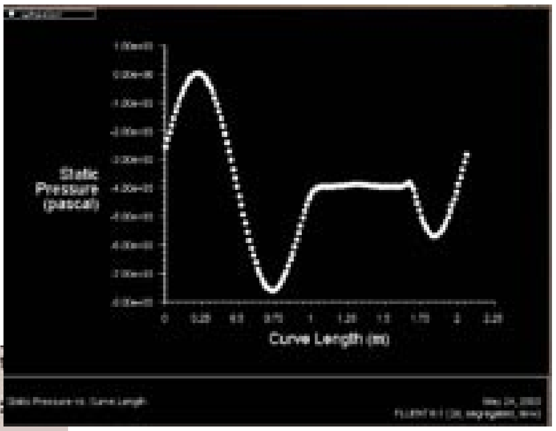
```
 turbulent viscosity limited to viscosity ratio of 1.000000e+05 in
!  495 solution is converged
   495 9.9972e-06 6.2345e-07 3.1907e-07 1.0117e-06 1.5386e-06  0:00

Force vector: (0 1 0)
                              pressure          viscous            total
zone name                        force            force            force
                                     n                n                n
------------------------------ --------------- ---------------- ---------------- --
bottomwall                     61785.387  5.9107082e-19        61785.387
topwall                       -60934.609  4.0989132e-20       -60934.609
wheeloc                       -863.43378       0.189435       -855.24434
```

LIFT =
-855.2



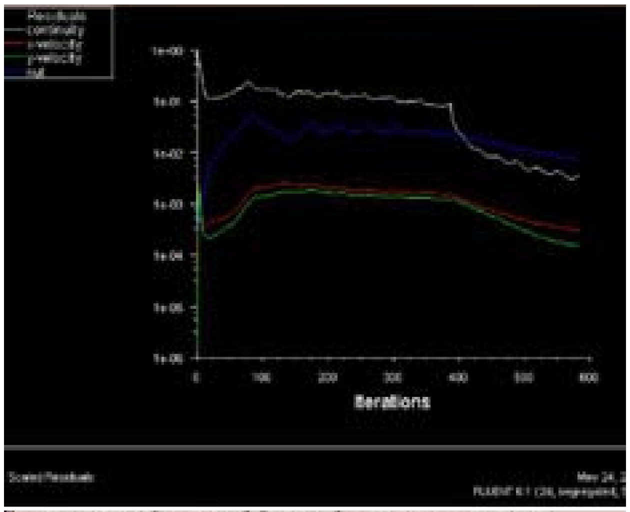
```
 turbulent viscosity limited to viscosity ratio of 1.000000e+05 in 6312 c
   245 9.2200e-03 1.8586e-04 1.7678e-04 8.3588e-04 4.8634e-04  0:03:08  2

Interrupting...
 turbulent viscosity limited to viscosity ratio of 1.000000e+05 in 6326 c
Done.

Force vector: (0 1 0)
                              pressure          viscous            total
zone name                        force            force            force
                                     n                n                n
------------------------------ --------------- ---------------- ---------------- ---
bottomwall                     52617.891  1.2481485e-18        52617.891
topwall                       -51402.652 -3.3523415e-19       -51402.652
wheeloc                        -1064.656      6.6009502       -1058.0551
------------------------------ --------------- ---------------- ---------------- ---
net                            150.58228      6.6009502        157.18323
```

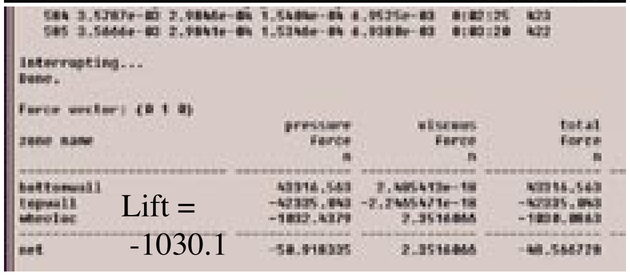
LIFT =
-1058.1



Left and above: Results from using a standard (left) and SST (left and (bottom left) k-omega models. Note the message (which seems to be dependent on initial boundary <inlet> conditions) "turbulent viscosity limited to a viscosity ratio of..." in both results, something which I made sure to avoid in my other models. The number of cells this occurred in decreased in the SST model. The K-omega is a low Reynold's number model in any case.

PAGE 15: More runs, this time with various turbulence models. Again all runs were with the final quad medium coarse mesh (31368 nodes, no adaption). These runs compare to the lift from the Standard K-E model at -1024.8.

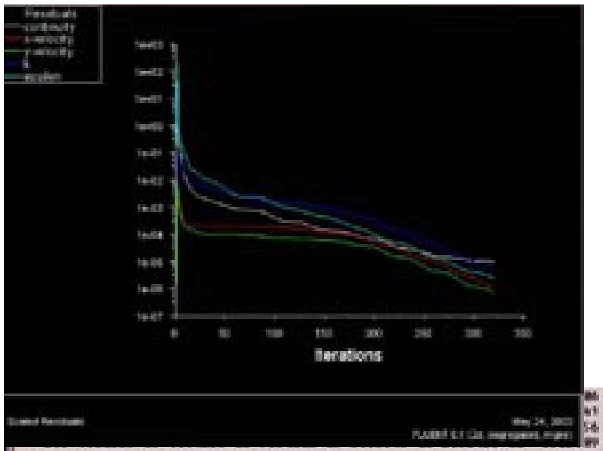


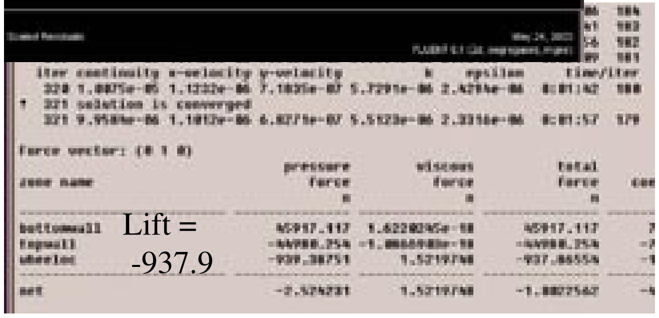

```
   584 3.5787e-03 2.9866e-06 1.5480e-06 6.9535e-03  0:02:25  423
   585 3.5666e-03 2.9861e-06 1.5346e-06 6.9388e-03  0:02:20  422

interrupting...
Done.

Force vector: (0 1 0)
                                   pressure         viscous           total
zone name                             force           force           force
                                          n               n               n
------------------------------ --------------- --------------- ---------------
bottomwall                        43316.543   7.4054913e-18       43316.543
topwall                          -42335.043  -2.2465471e-18      -42335.043
wheeloc                          -1032.4379       2.3516066      -1030.0863
------------------------------ --------------- --------------- ---------------
net                              -50.918335       2.3516066      -48.566728
```

Lift =
-1030.1

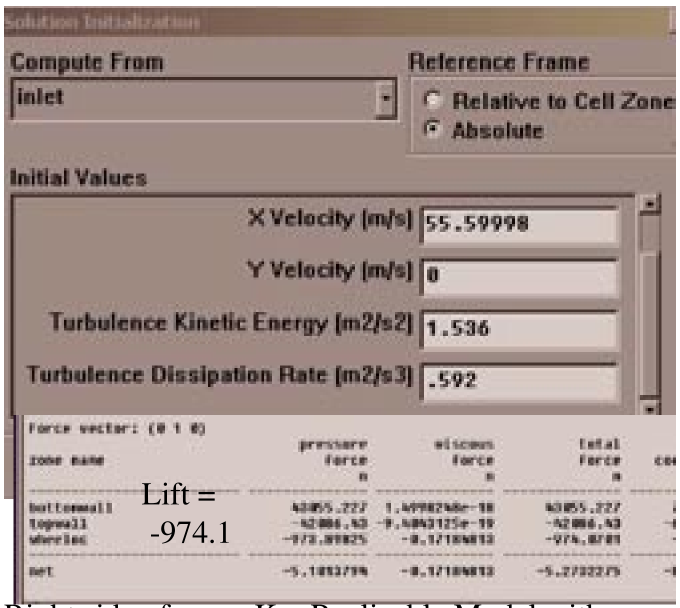
Above: Spalart Allmaras model run. The sudden drop in continuity after a certain number of iterations was from a change in under relaxation factors. The model ran quickly but residuals did not drop below 0.001

Below: K-e RNG model run. Converged to 0.0001 residual level after 321 iterations.




```
  iter continuity x-velocity y-velocity          k    epsilon     time/iter
   320 1.0875e-05 1.1232e-06 7.1835e-07 5.7291e-06 2.4294e-06  0:01:42   180
!  321 solution is converged
   321 9.9584e-06 1.1012e-06 6.8271e-07 5.5123e-06 2.3316e-06  0:01:57   179

Force vector: (0 1 0)
                                   pressure         viscous           total
zone name                             force           force           force    coe
                                          n               n               n
------------------------------ --------------- --------------- --------------- ----
bottomwall                        45917.117   1.6220245e-18       45917.117      7
topwall                          -44980.254  -1.0868980e-18      -44980.254     -7
wheeloc                          -939.38751       1.5219748      -937.86554     -1
------------------------------ --------------- --------------- --------------- ----
net                              -2.524231       1.5219748      -1.0022562     -4
```

Lift =
-937.9


Solution Initialization

Compute From: inlet

Reference Frame: ( ) Relative to Cell Zone [✓ Absolute]

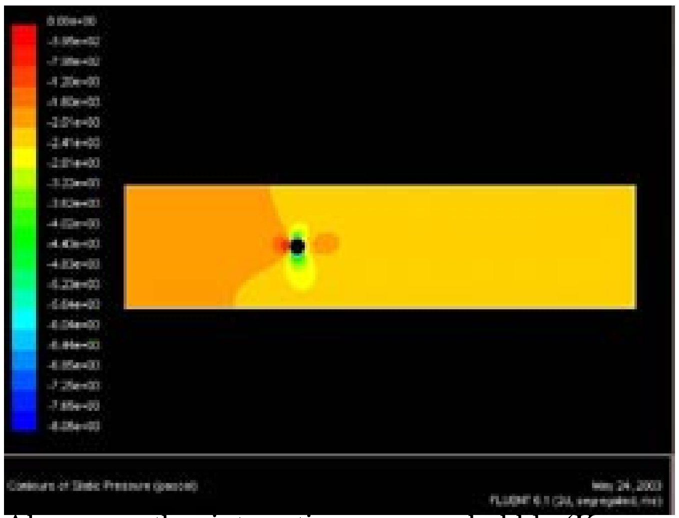
Initial Values

X Velocity (m/s)	55.59998
Y Velocity (m/s)	0
Turbulence Kinetic Energy (m2/s2)	1.536
Turbulence Dissipation Rate (m2/s3)	.592

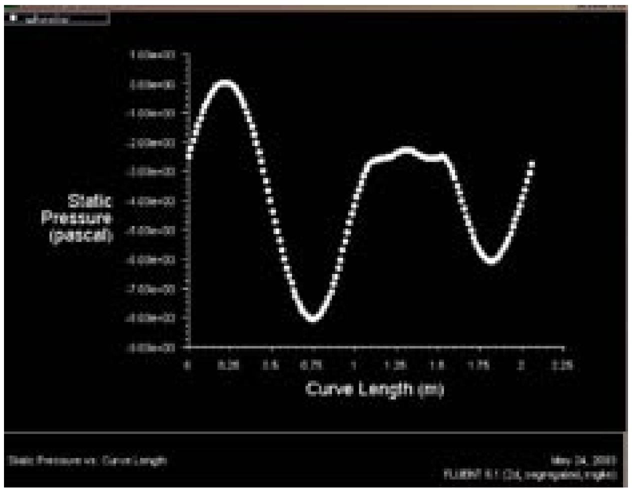
```
Force vector: (0 1 0)
                                   pressure         viscous           total
zone name                             force           force           force    coe
                                          n               n               n
------------------------------ --------------- --------------- --------------- ----
bottomwall                        43055.227   1.4998248e-18       43055.227
topwall                          -42086.43   -9.4843125e-19      -42086.43
wheeloc                          -973.89825      -0.17184813      -974.0701
------------------------------ --------------- --------------- --------------- ----
net                              -5.1093794      -0.17184813      -5.2732275
```

Lift =
-974.1

Right side of page: K-e Realizable Model with reworked turbulent dissipation rate plugged in. It converged to 0.0001 residual level after 335 iterations.



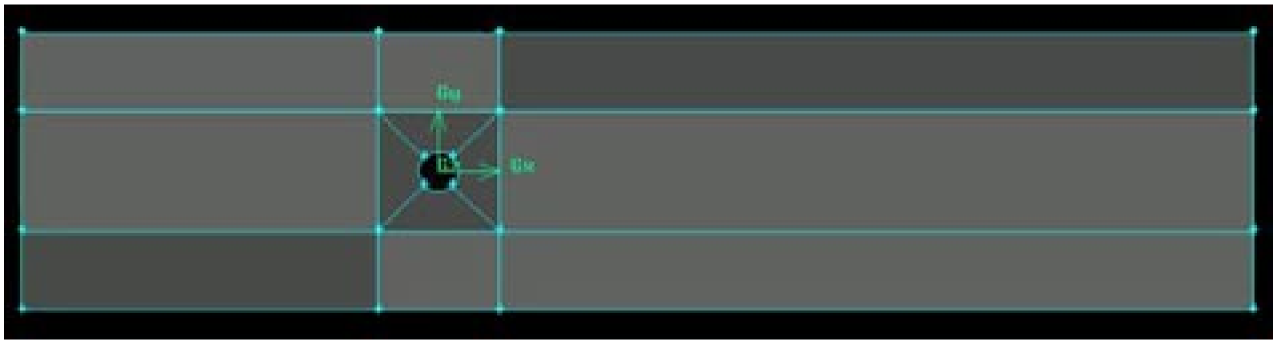
Above: another interesting pressure bubble (K-e Realizable solution).



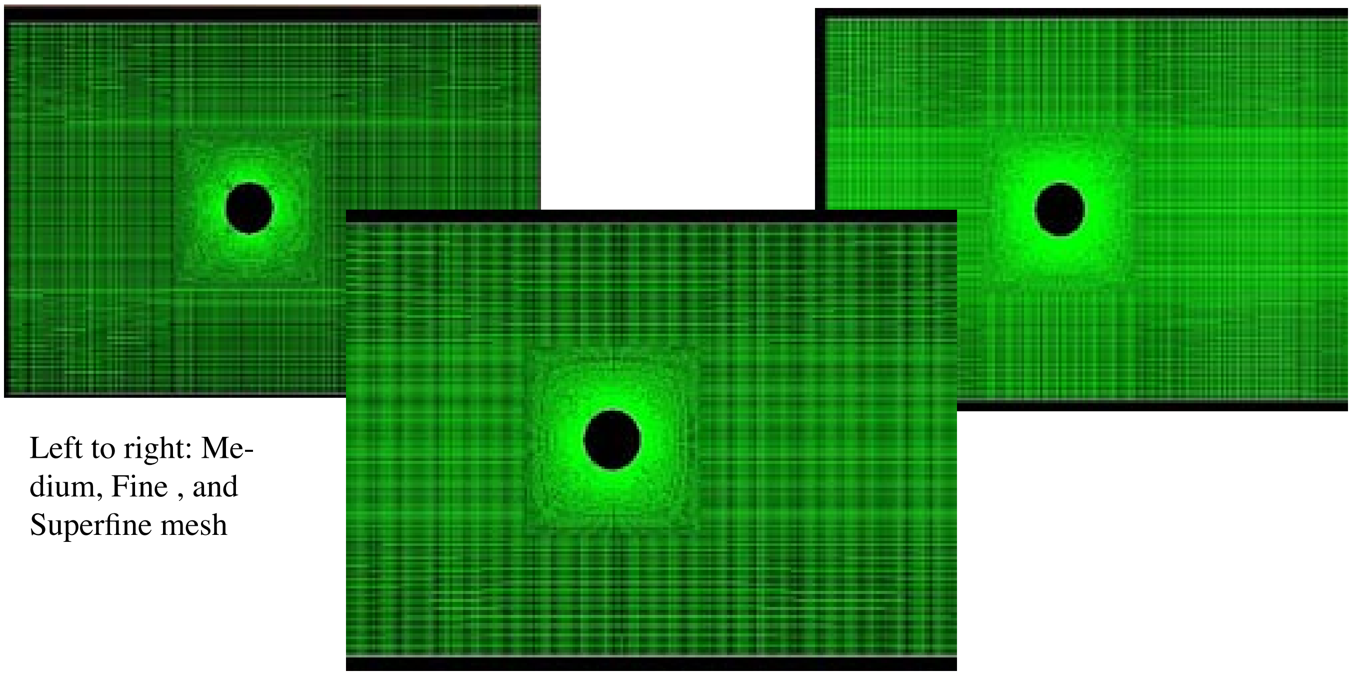
PAGE 16: At this point, thinking that I was on the right track, I ran a complete set of tests on "new improved" versions the medium, fine, and superfine quad meshes (12 faces). I focused on keeping the grid shape square with less than 5% gradient change from cell to cell. Initially all these runs showed high y+ values, but after as series of adaptions and iterations I was able to bring the y+ values down to the 350 range, which I believed acceptable for this higher Reynold's number flow.

These tests were rerun with a standard K-epsilon model with standard wall functions, second order SimpleC discretiation with standard pressure discretiation. The inlet turbulent boundary conditions were initialized with the intensity and length scale values based on a hydraulic diameter of 0.411 (which assumed a unit width of one, inlet 4.62m high). Length scale calculated as l=0.07 (hydraulic diameter, and intensity using the formula of $I = 0.16(Re_{Dh})^{-1/8}$.

The following pages contain the results. These were all based on the following meshes, consistent in their scale:



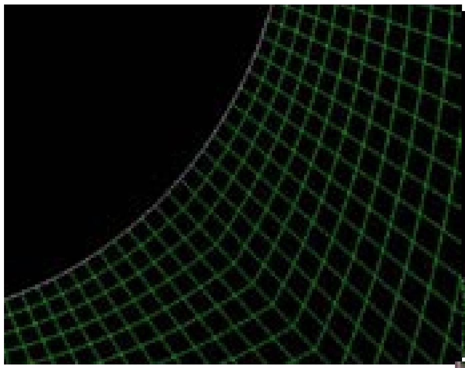
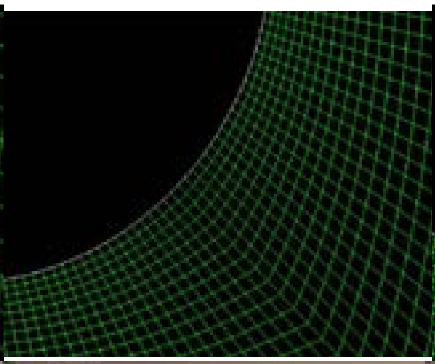
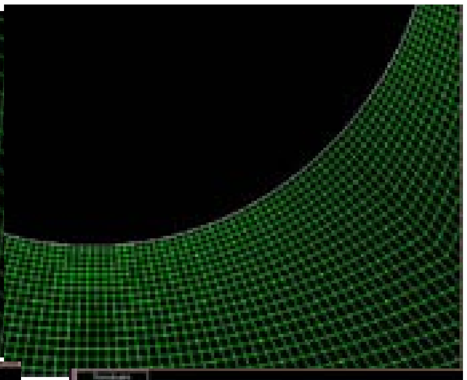
Mesh Seeds:	Circle	Inner radials	Int. horizontals	Int. verticals	Total Nodes
Medium	148	0.05 /1.05	0.06	0.06 / 1.05	31368
Fine	240	0.0375/1.05	0.0375	0.05 / 1.05	47095
SuperFine	300	0.025/1.05	0.03	0.04 / 1.05	97096
(units)	Number of	Size/Suc.Ratio	SIze	Size/ratio	

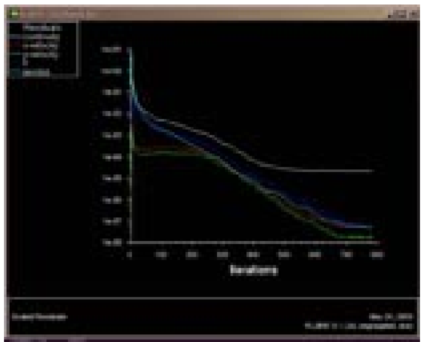


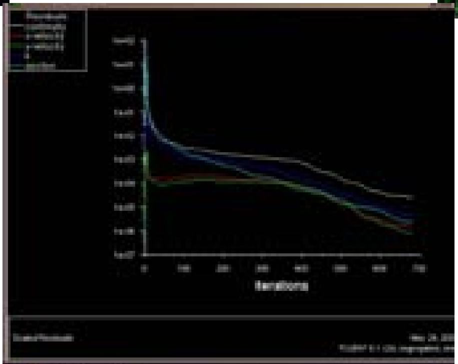
Left to right: Medium, Fine , and Superfine mesh

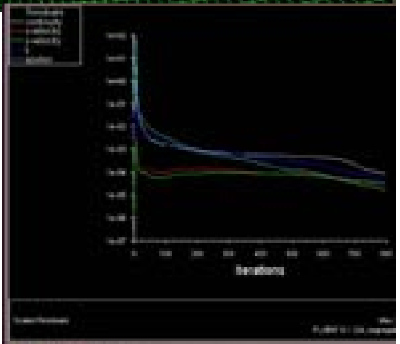
Page 17. BEFORE Y+ADAPTION PROCESS Initial Mesh runs.

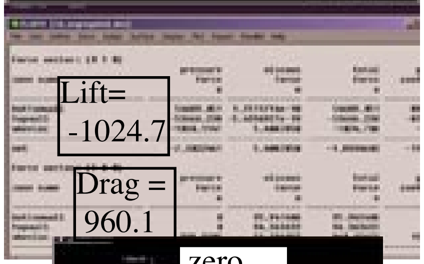
Medium Mesh Results | Fine Mesh Results | “SuperFine” Mesh Results

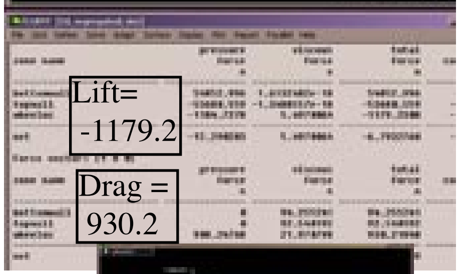




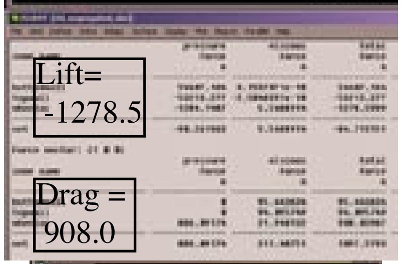


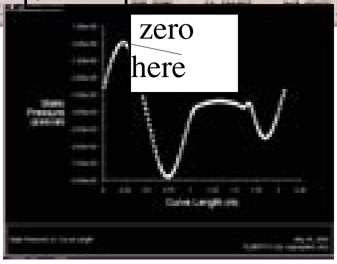




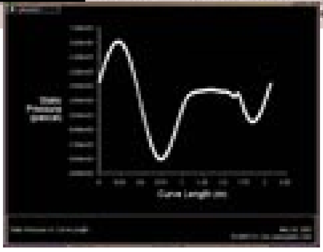


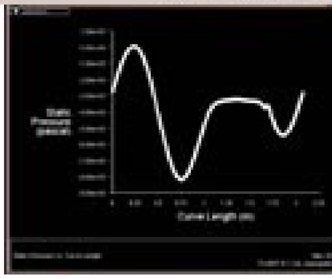


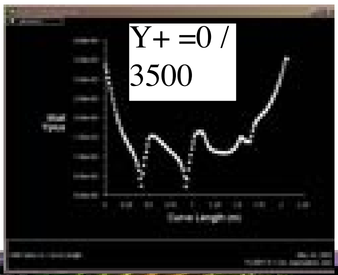


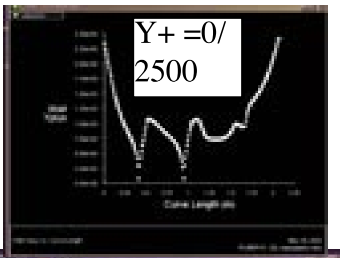


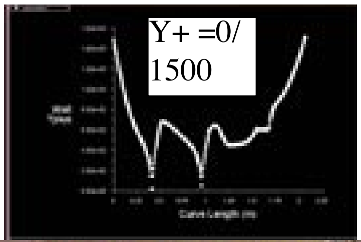
Zero pressure line normal

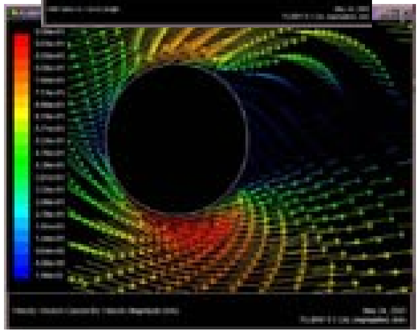
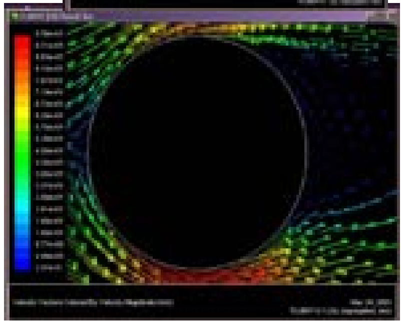
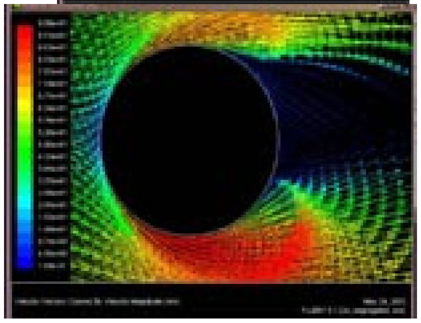




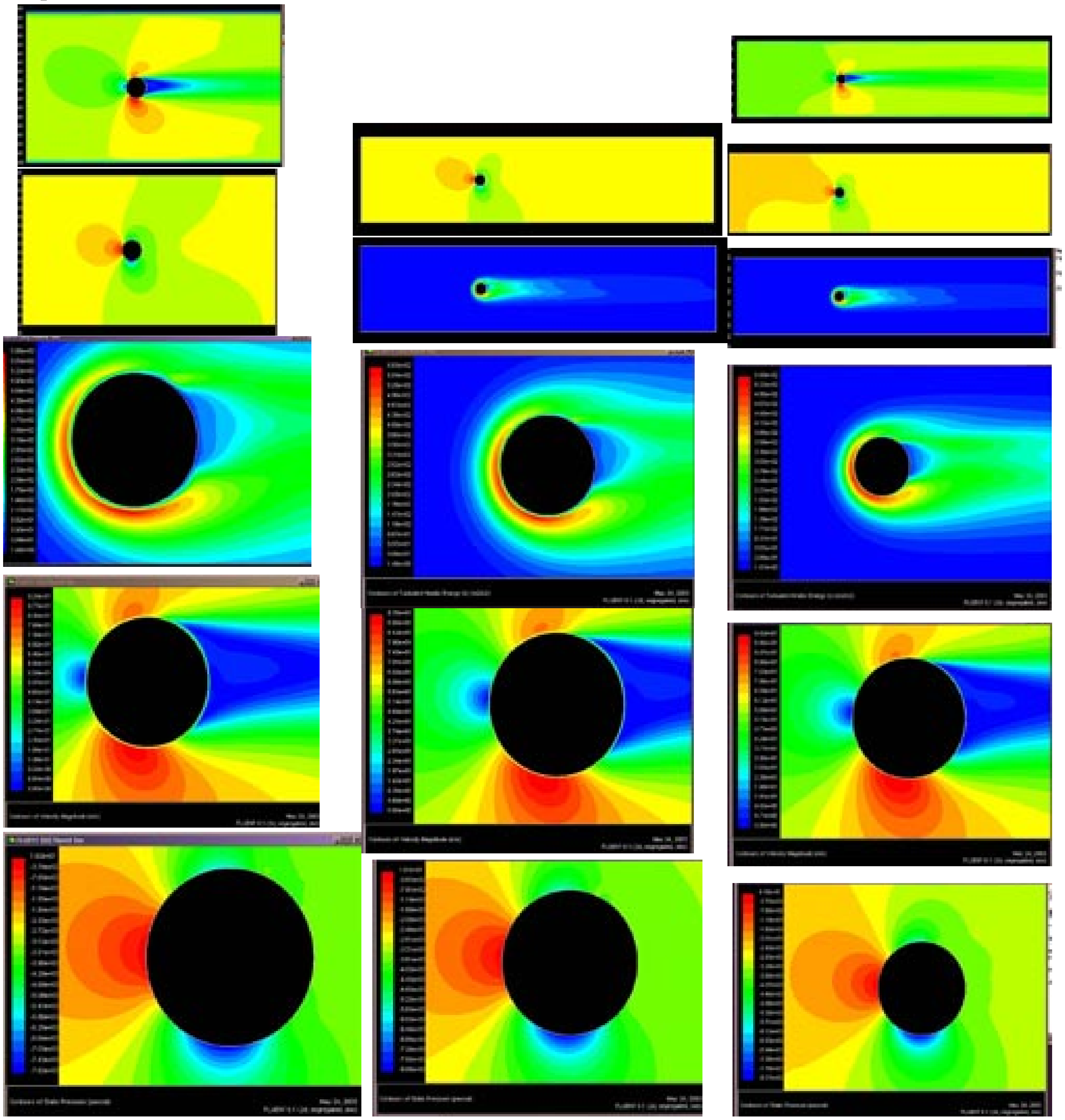






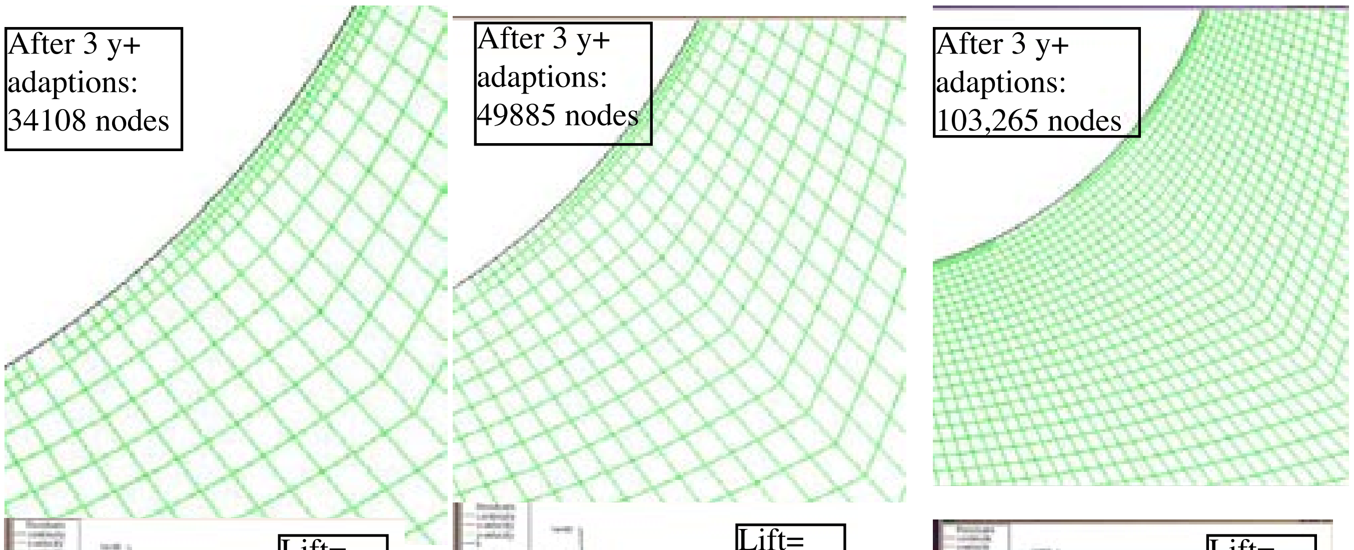




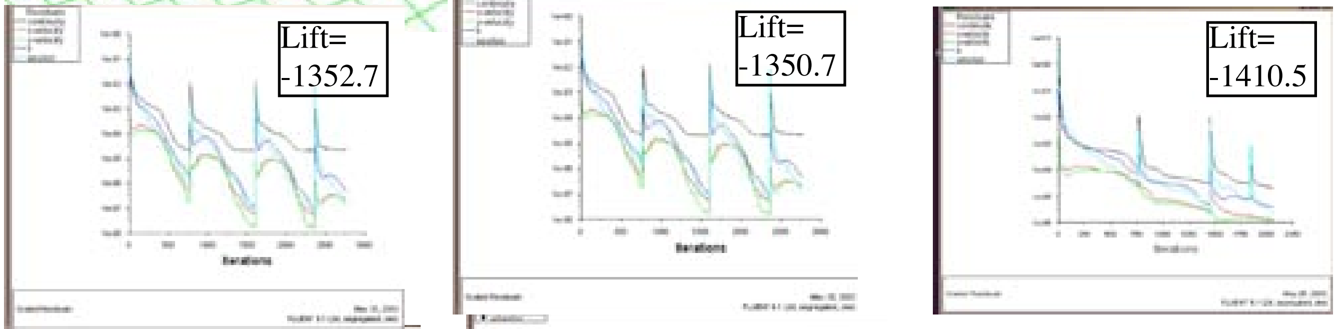
Page 18. BEFORE Y+ ADAPTION PROCESS results cont. (left: medium mesh, center: fine, right: "Superfine".)

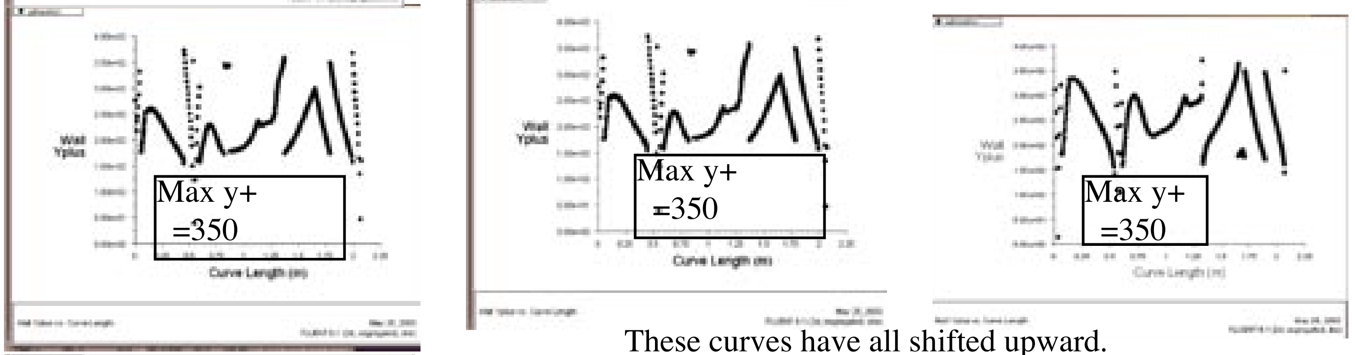


Comment and Observations: Even for the "superfine" mesh, initial y+ values are too high. This should have prompted me to create a finer mesh around the circle, but I was thinking at the time that doing so would exceed 100,000 nodes, since I was under the understanding that mesh size needed to be rectangular, and that the gradient could never exceed 5% (discussion with John Reizes, May 2003). Now I realize that more nodes in the boundary zones of interest is much more important for an accurate result, and perhaps a 8% increase in grid from cell to cell may be preferable. Another thing that becomes clear while comparing this data side by side is that the contours don't change much for a given viscous turbulence model, but the functionals do. I focused primarily on the lift coefficients, but now I realize that there is a pattern to the converging solution, namely, that lift increases, and drag decreases. Therefore, I should look in the future at the lift/drag ratios. This basic observation only came to me later, however, and I next went though an obsessive and probably useless adaption process.

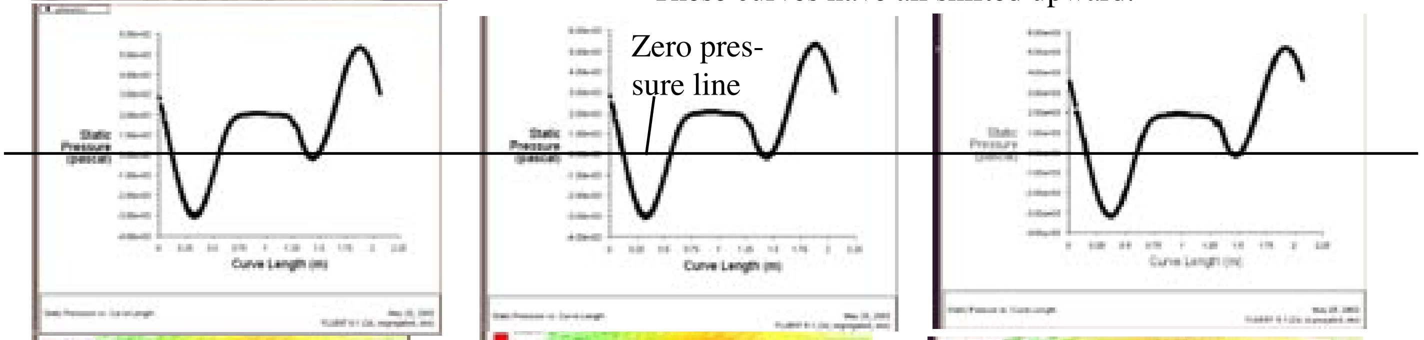
Page 19. RESULTS AFTER SUCCESSIVE Y+ ADAPTION PROCESS (standard Ke model). Note: adaptions were performed only on circle boundary.

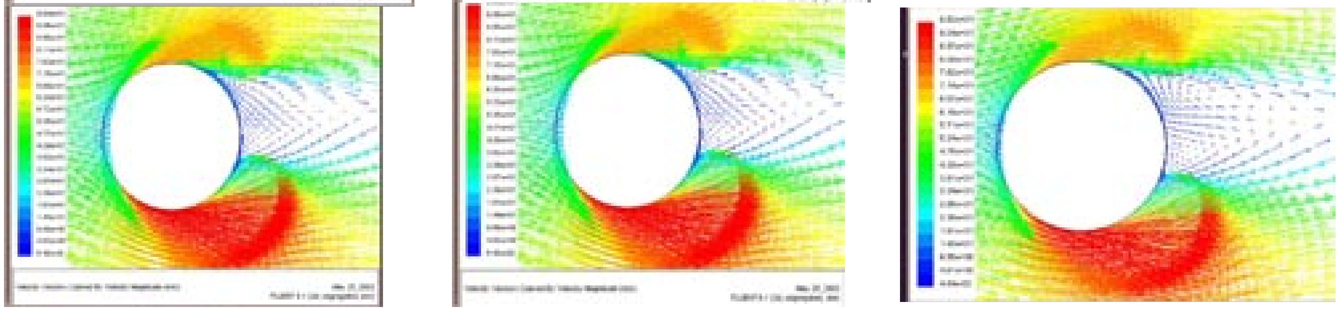

After 3 y+ adaptions: 34108 nodes
After 3 y+ adaptions: 49885 nodes
After 3 y+ adaptions: 103,265 nodes



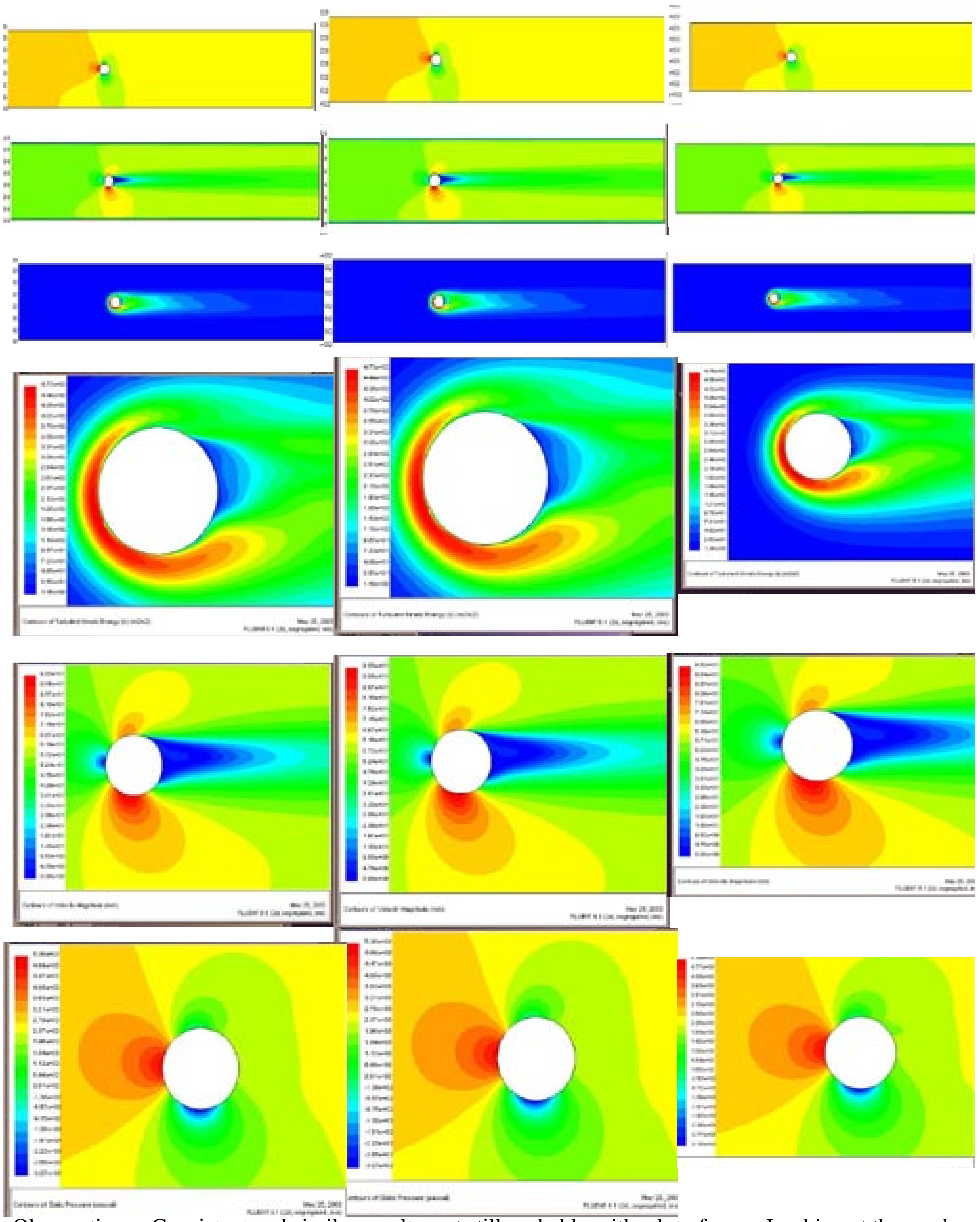


These curves have all shifted upward.



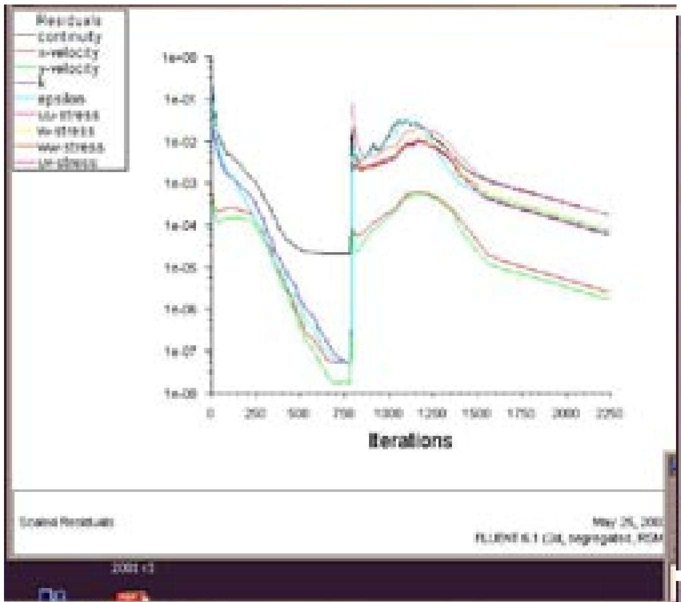


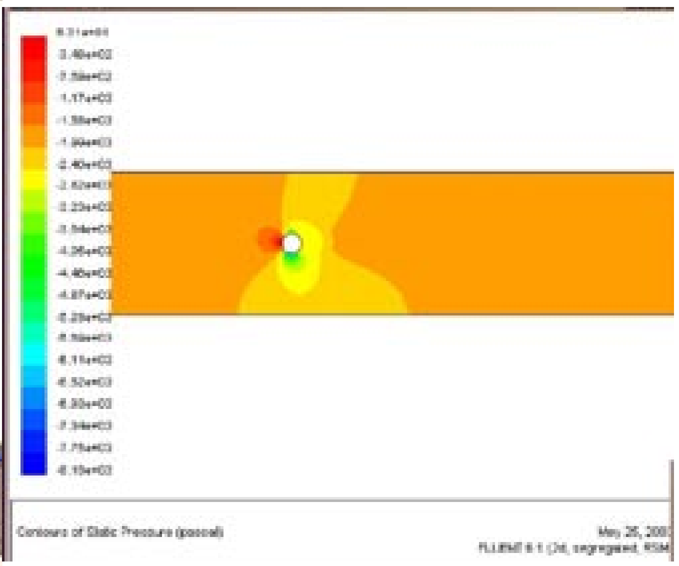
Page 20. RESULTS AFTER ADAPTION (continued). Left: medium mesh, center: fine, right:"super-fine"

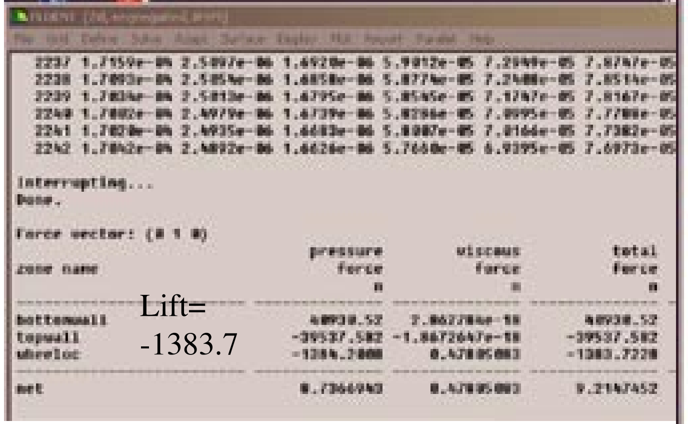


Observations: Consistent and similar results, yet still probably with a lot of error. Looking at the meshes one can see frequent gradients, probably not a good thing. Unfortunately, I neglected to record the drag values for these runs. But at least I at last learned how to plot white backgrounds.

Page 21. Reynold's Stress Model again, this time with the medium mesh (unadapted) after it had already been solved by the Ke standard turbulence model.

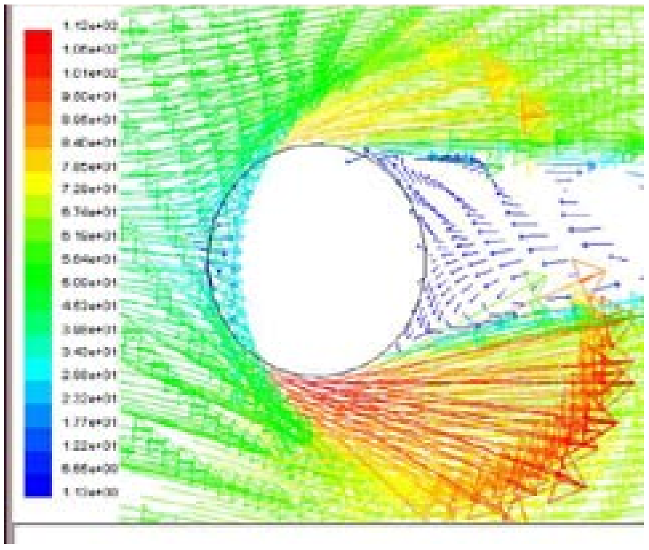


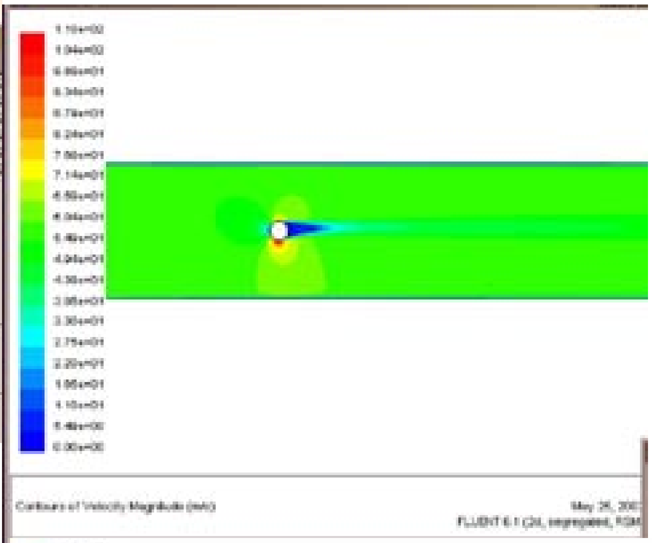

Contours of Static Pressure (pascal)
May 26, 2003
FLUENT 6.1 (2d, segregated, RSM)

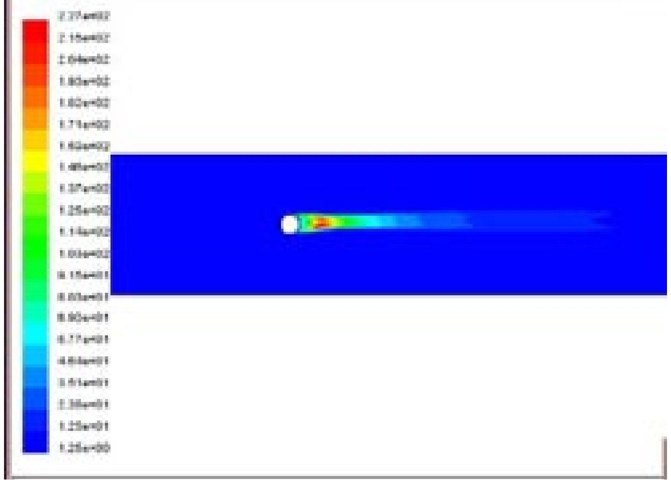


```
  2237 1.7159e-04 2.5097e-06 1.6920e-06 5.9012e-05 7.2949e-05 7.8747e-05
  2238 1.7093e-04 2.5054e-06 1.6858e-06 5.8774e-05 7.2408e-05 7.8514e-05
  2239 1.7034e-04 2.5013e-06 1.6795e-06 5.8545e-05 7.1747e-05 7.8167e-05
  2240 1.7002e-04 2.4979e-06 1.6739e-06 5.8296e-05 7.0995e-05 7.7788e-05
  2241 1.7020e-04 2.4935e-06 1.6683e-06 5.8007e-05 7.0166e-05 7.7302e-05
  2242 1.7042e-04 2.4892e-06 1.6626e-06 5.7660e-05 6.9395e-05 7.6973e-05

interrupting...
Done.

Force vector: (0 1 0)
                              pressure          viscous            total
zone name                        force            force            force
                                     n                n                n
-------------------------  ---------------  ---------------  ---------------
bottomwall                        40930.52   7.0627840e-18          40930.52
topwall                         -39537.582  -1.0672647e-18        -39537.582
wheeloc                         -1384.2000      0.47805003        -1383.7220
-------------------------  ---------------  ---------------  ---------------
net                              8.7366943      0.47805003         9.2147452
```

Lift=
-1383.7

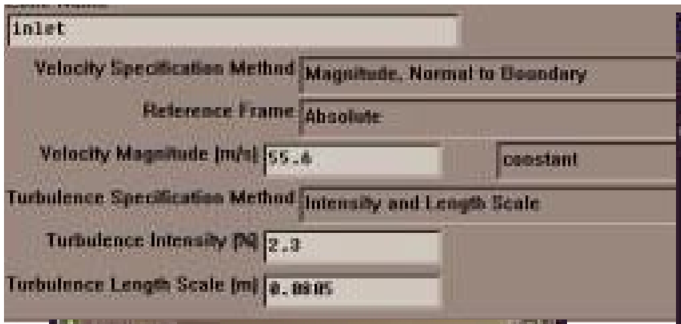



Contours of Velocity Magnitude (m/s)
May 26, 2003
FLUENT 6.1 (2d, segregated, RSM)

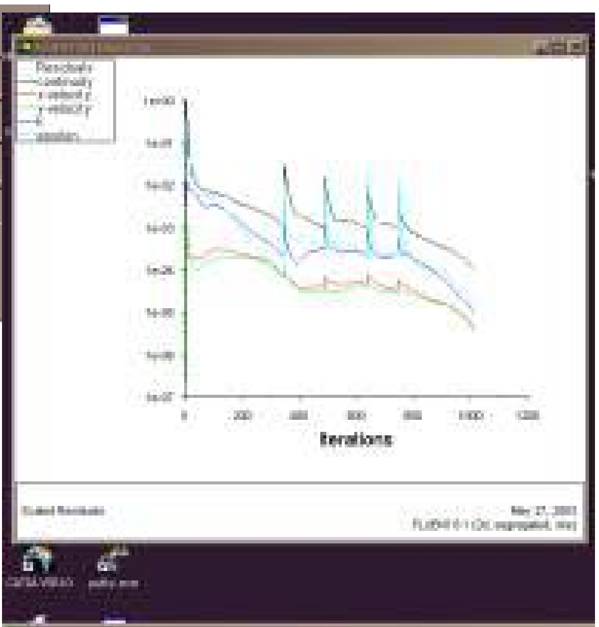


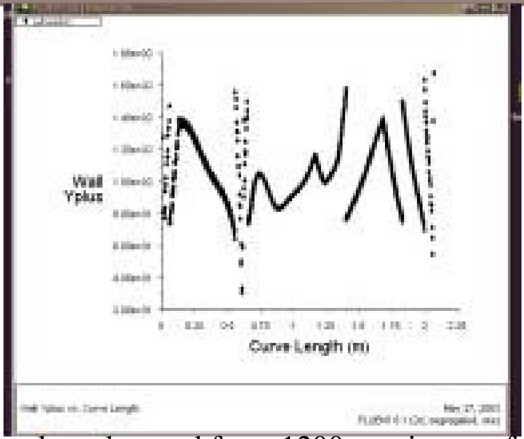
Observations: The results are quite similar to the Ke model solution, yet the contours look underdeveloped and may represent an unconverged solution.

PAGE 22: The next couple runs involve changing two parameters. The main one the use of the PRESTO pressure discretiation scheme. The other is a change to the inlet turbulence setting. Using PRESTO, I was getting the message regarding turbulence viscosity level limit in x number of cells, so I reworked the turbulence parameters based on a Hydraulic diameter of 0.411, which assumes a 4.62 meter square inlet condition, which resulted in a length scale of 0.07*0.411=0.0805, and an Intensity of 2.3%, using the standard formula as before. Below Fine mesh results.

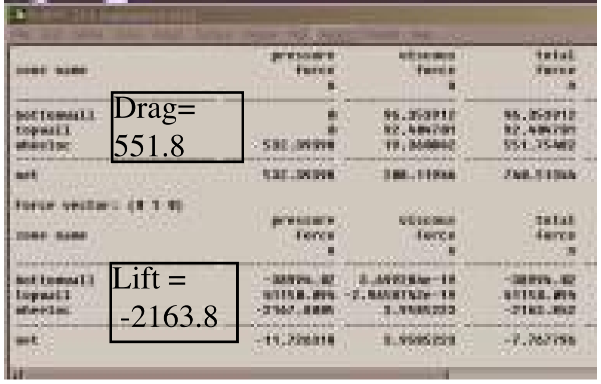


	inlet	
Velocity Specification Method	Magnitude, Normal to Boundary	
Reference Frame	Absolute	
Velocity Magnitude (m/s)	55.6	constant
Turbulence Specification Method	Intensity and Length Scale	
Turbulence Intensity (%)	2.3	
Turbulence Length Scale (m)	0.0805	





Y+ values dropped from 1200 maximum after the first run to 175 after several y+ adaption runs. The fact that the curve breaks up like this is a bit troubling, but the y+ values began smaller, and got smaller faster, using PRESTO.

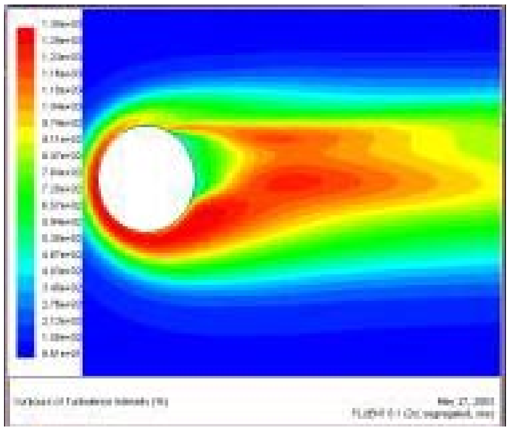


```
                      pressure        viscous          total
zone name                force          force          force
                             n              n              n
------------------------------------------------------------
bottomwall                   0      96.353912      96.353912
topwall                      0      92.406781      92.406781
wheelie            532.39998      19.360002      551.75402
------------------------------------------------------------
net                532.39998      208.11966      740.51366

force vector: (0 1 0)
                      pressure        viscous          total
zone name                force          force          force
                             n              n              n
------------------------------------------------------------
bottomwall           -38996.82   2.6993e-16       -38996.82
topwall              41158.095  -7.9450762e-16    41158.095
wheelie             -2167.8808      3.9985233     -2163.882
------------------------------------------------------------
net                 -11.226018      3.9985233     -7.262796
```

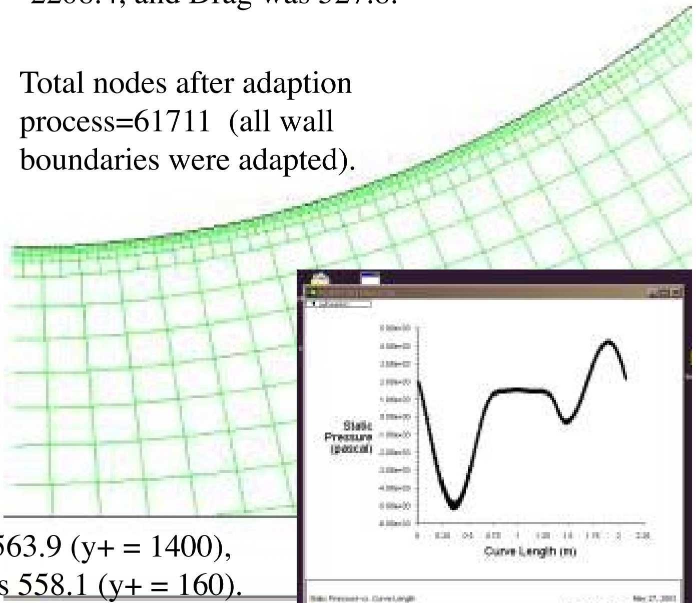
Drag= 551.8

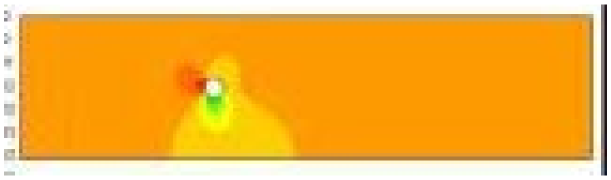
Lift = -2163.8

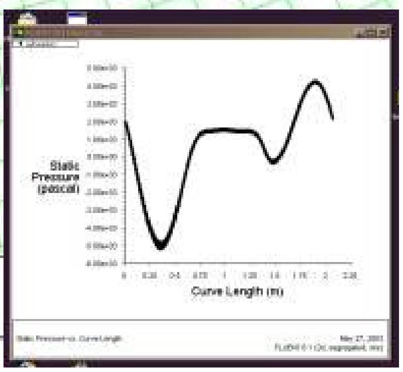
Note: after first run with PRESTO, Lift was -2208.4, and Drag was 527.8.



Total nodes after adaption process=61711 (all wall boundaries were adapted).






Static Pressure (pascal)
Curve Length (m)
Static Pressure vs. Curve Length
May 27, 2003
FLUENT 6.1 (2d, segregated, ske)

Similar results occurred for the Superfine Mesh. Initial run indicated a lift and drag of -2182.0 and 563.9 (y+ = 1400), and after 4 adaptions, lift was -2161.9 and drag was 558.1 (y+ = 160).

PAGE 23: The previous results were mostly done in a flurry of activity using the manuals to help decipher the complex program, and are mostly worthless results. However, after writing all the previous pages up I was able to take a calm look at the results thus far, and a few important things suddenly dawned. The basic principles of fluid dynamics were at last considered. In the previous tests, I had chosen mostly to look at lift as my functional, and I hoped to get a consistent, converging lift result. But as you can see, values change dramatically depending on the turbulence model. It's clear that even my finest grid was not fine enough in then area of interest. Most importantly, I realized that a random blunt approach does not work for the complex study of turbulent fluid dynamics, and that before any study it is important to consider carefully the underlying principles. The relationship between lift and drag in the CFD solutions seemed consistent: as Fluent results showed increased lift, the drag values would decrease). Perusing some established research, I uncovered the diagram below. I also used this time to write up my report, and in the ensuing wrestling with the Fluent manual, I discovered that it is possible to look at the forces (lift and drag) in a plotted version as the model solves. This was a key to the next step of understanding.

Sailing With a Rotating Cylinder Sail

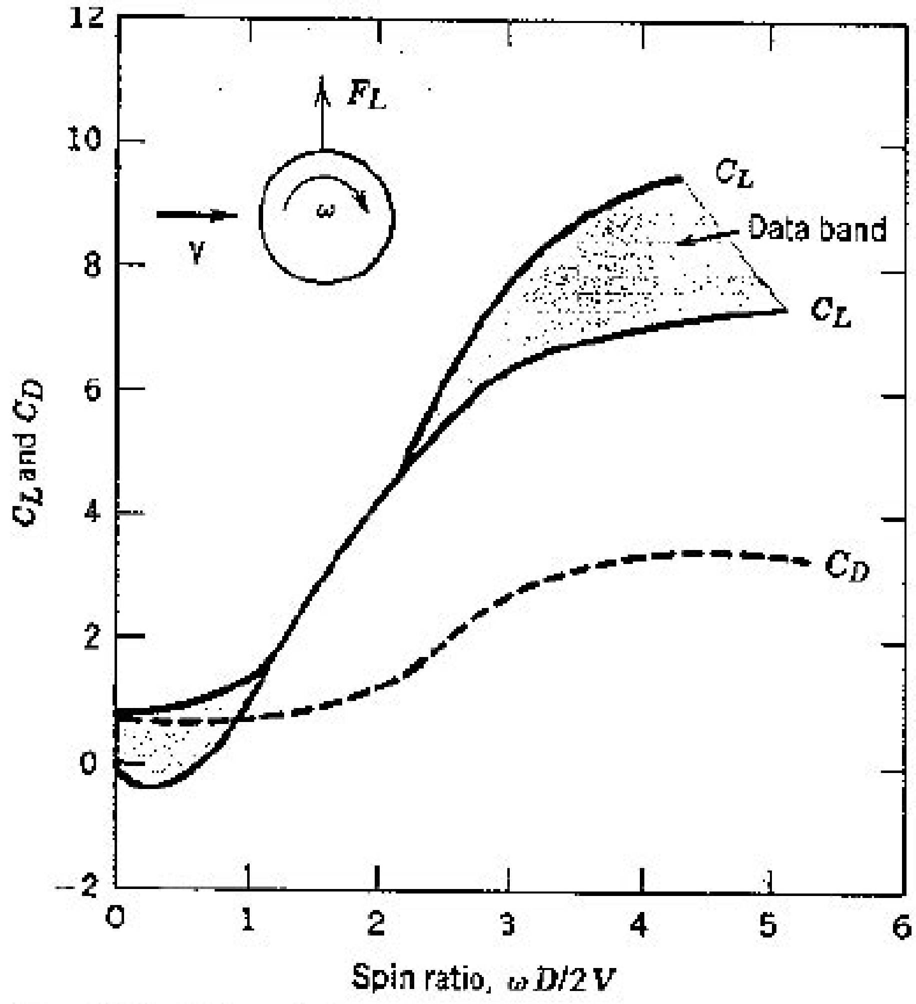


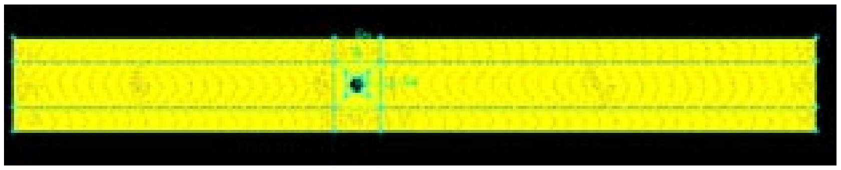
Fig. 9.29 Lift and drag of a rotating cylinder as a function of relative rotational speed: Magnus force. (Data from [32].)

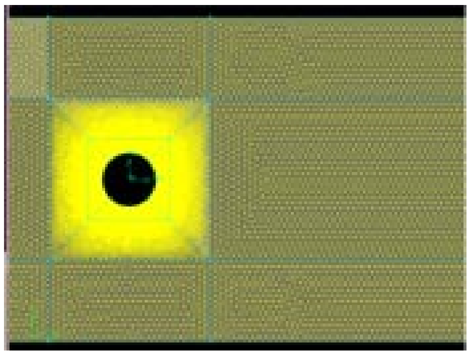
This diagram indicates that lift and drag should be on the same order, as the spin ratio of our problem is wD/2V=1 . W=168 radians/sec, V=55.6m/sec, and D=0.66m).

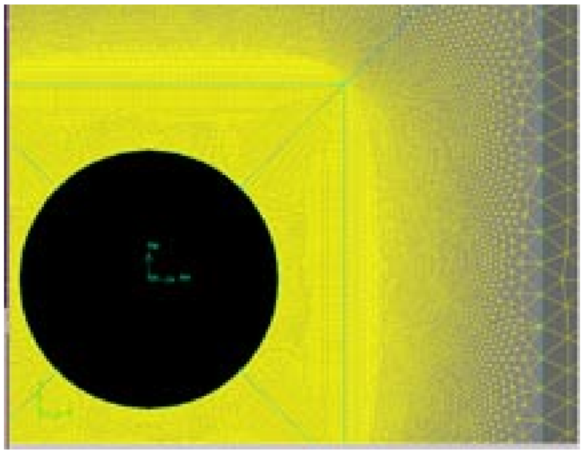
The coefficient of lift is obtained by diving the lift force by 1/2 times the density times the velocity squared times the diameter (for this 2D problem), which results in a value of 1250 for our free stream flow.

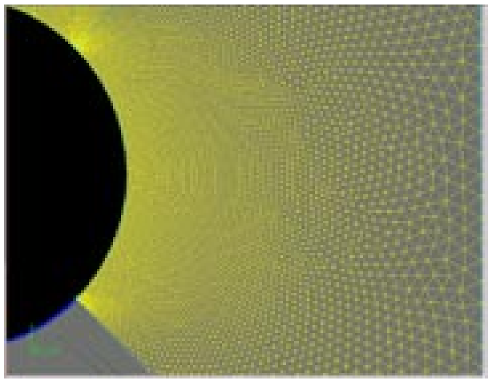
Source: Louisiana State University Mechanical Engineering website. Original source unknown.

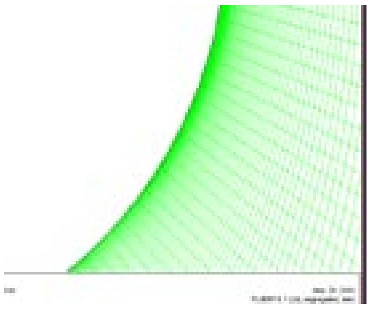
Page 25. Next I tried a variety of new grids, and although the y+ problem was sometimes solved at the wall boundary, the coarseness and grid gradients in other places caused problems.

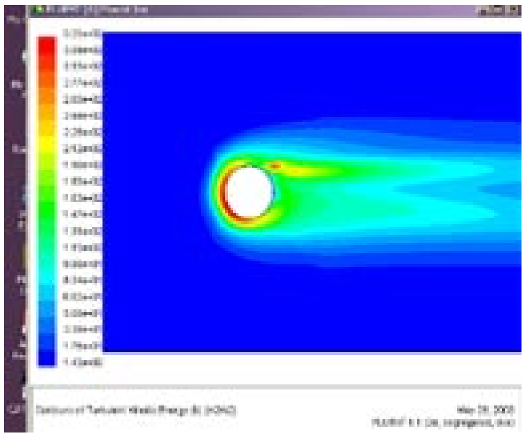

Extended inflow and outflow boundaries

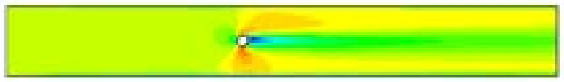

Hybrid triangular center, quad outer grids

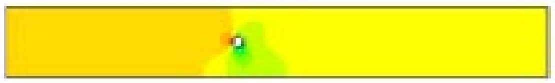


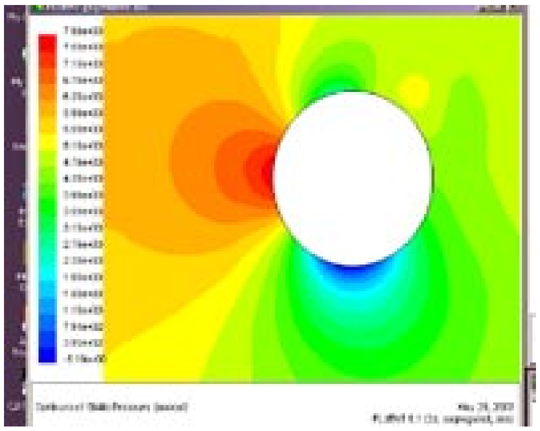


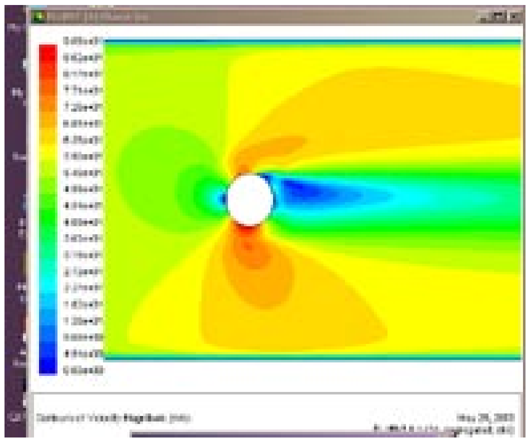




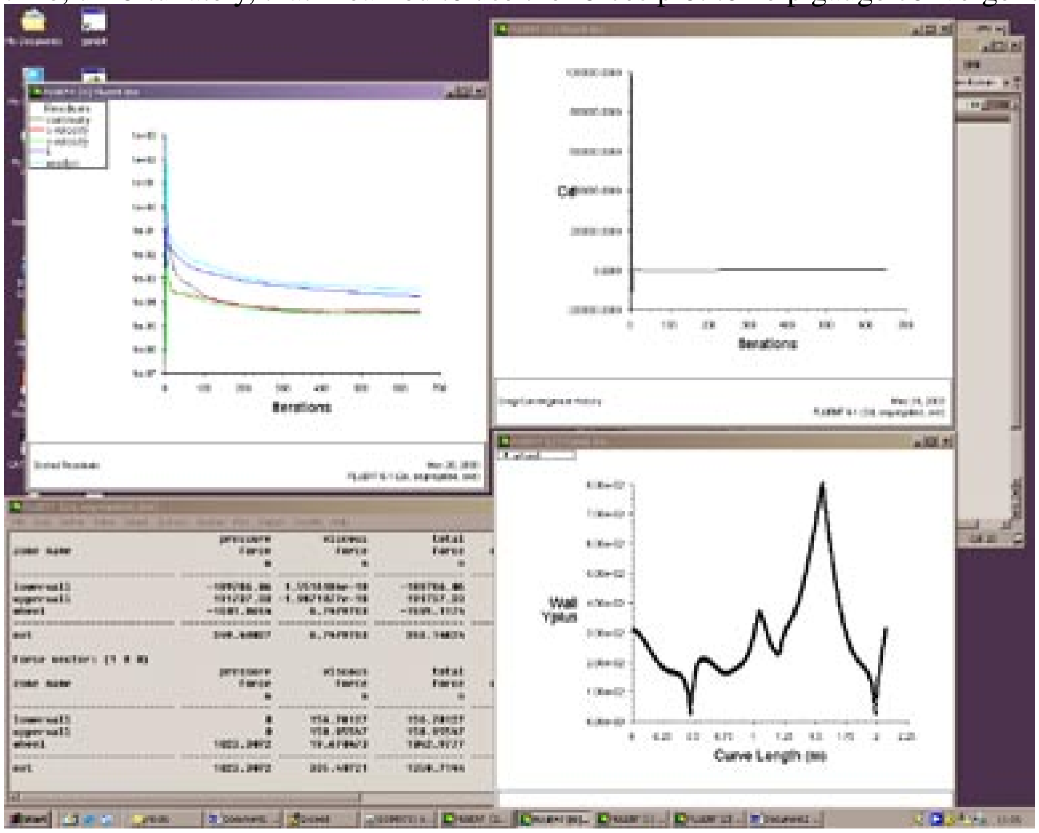






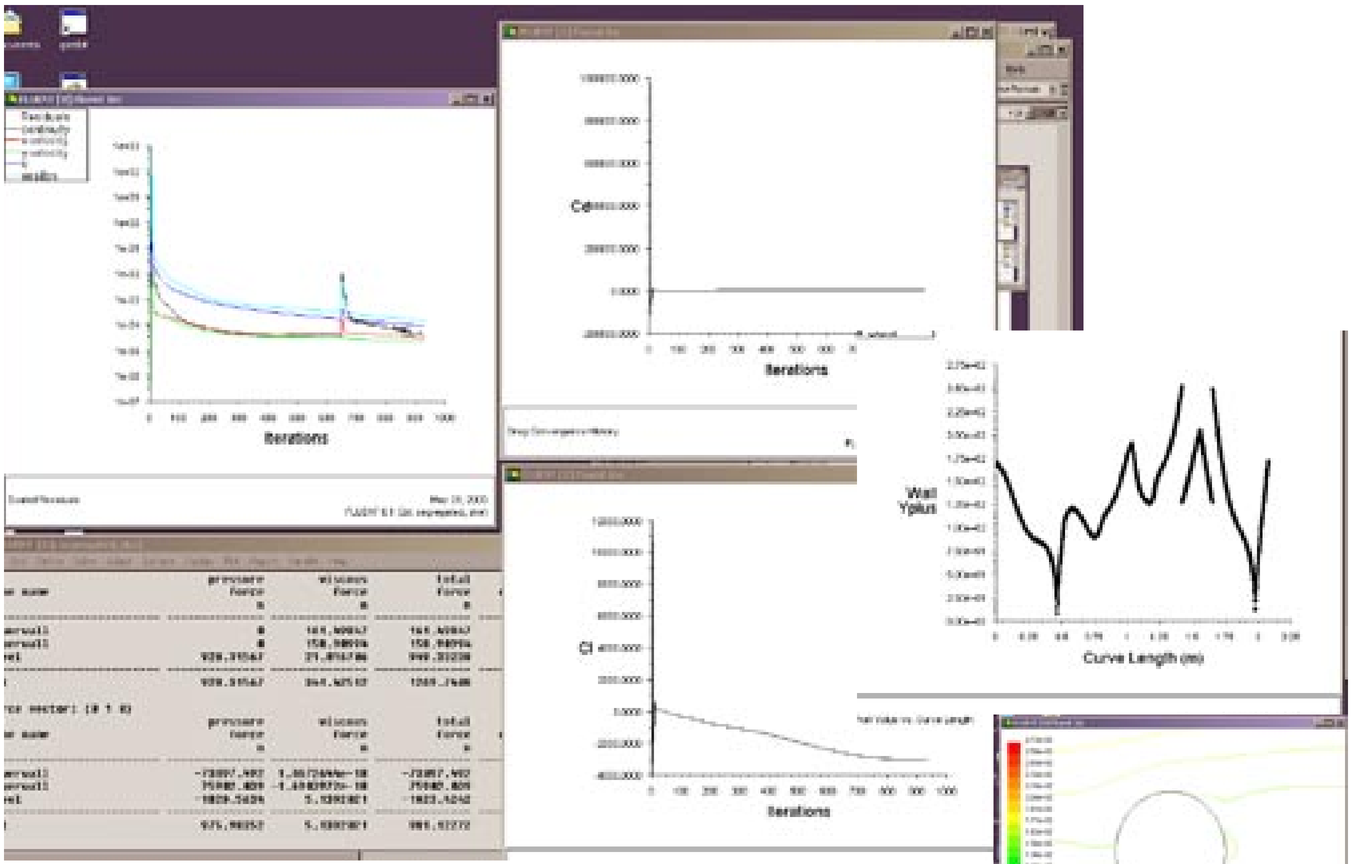

The dense mesh around the circle solved adequately for y+ but caused undue effects at the high mesh gradient boundaries

Page 26. Some results with a fine but high gradient mesh, with K-e turbulence model. It wasn't until this time, unfortunately, that I learned to use the forces plot to help gauge convergence.

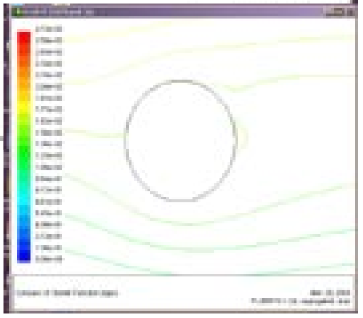


```
                        pressure        viscous          total
zone name                  force          force          force
                               n              n              n
-------------------- ------------- -------------- --------------
lowerwall             -109766.86  1.3574584e-18    -109766.86
upperwall              101727.32 -1.9827167e-18     101727.32
wheel                 -1581.8654      6.7470703    -1575.1174
-------------------- ------------- -------------- --------------
net                    340.48807      6.7470703     353.16024

Force vector: (1 0 0)
                        pressure        viscous          total
zone name                  force          force          force
                               n              n              n
-------------------- ------------- -------------- --------------
lowerwall                      0      156.70127      156.70127
upperwall                      0      158.05567      158.05567
wheel                  1823.2972      19.470423      1842.9777
-------------------- ------------- -------------- --------------
net                    1823.2972      335.44721      1258.7194
```

Two runs with very fine mesh around circle, before and after a single y+ boundary adaption.

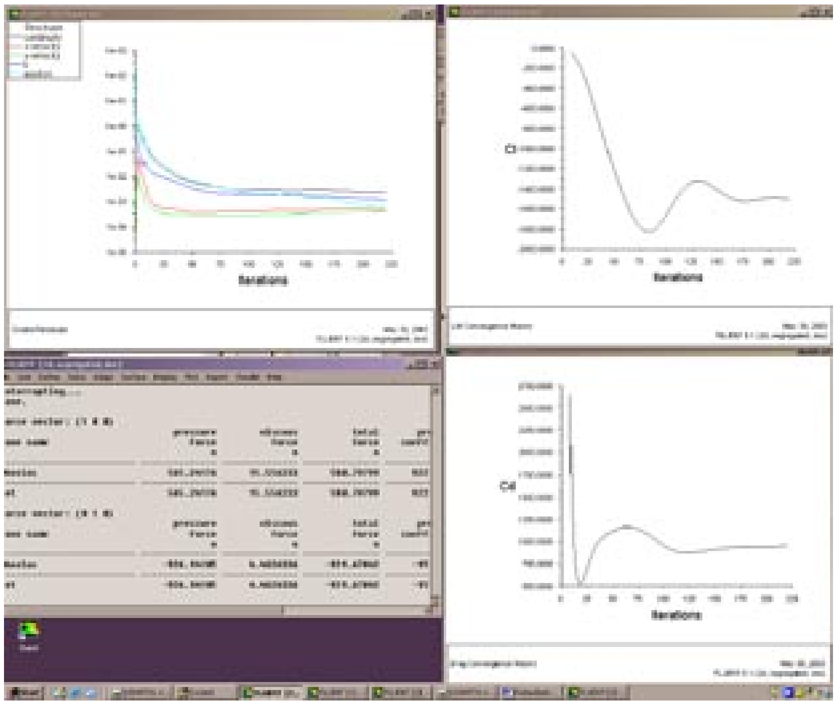


```
                        pressure        viscous          total
zone name                  force          force          force
                               n              n              n
-------------------- ------------- -------------- --------------
lowerwall                      0      161.40847      161.40847
upperwall                      0      158.98096      158.98096
wheel                  928.31567      21.816786      949.33238
-------------------- ------------- -------------- --------------
net                    928.31567      344.42512      1269.7408

Force vector: (0 1 0)
                        pressure        viscous          total
zone name                  force          force          force
                               n              n              n
-------------------- ------------- -------------- --------------
lowerwall             -73097.492  1.8672666e-18    -73097.492
upperwall              75040.409 -1.4940975e-18     75040.409
wheel                 -1029.5434      5.1093021    -1023.4242
-------------------- ------------- -------------- --------------
net                    975.98252      5.1093021     981.12272
```

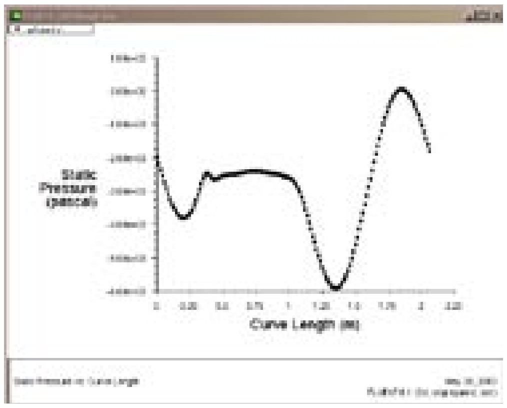


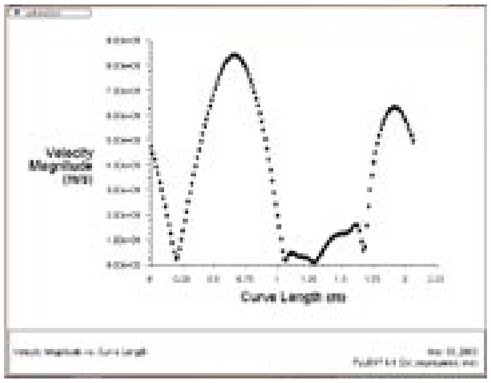
Perhaps this whole document could be entitled, "How not to learn Fluent from scratch". Most of the previous results are no good, but the process of seeing how complex the program is and the difficulty in setting the correct parameters was reinforced.

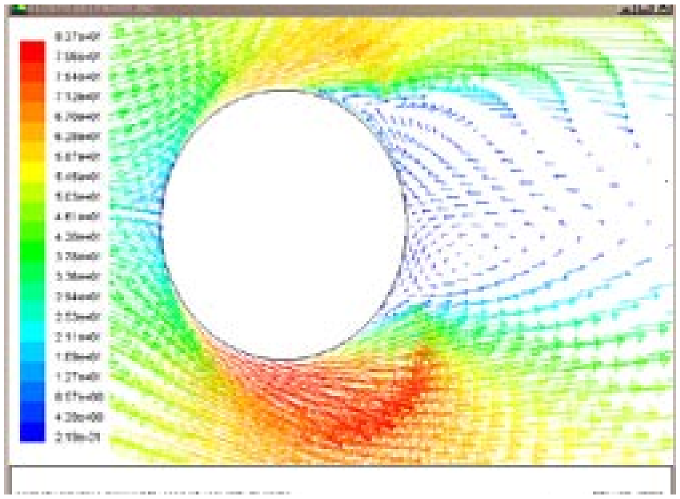
Page 26.Here, a run with the sidewalls of the wind tunnel changed from WALL to OUTFLOW are presented here.

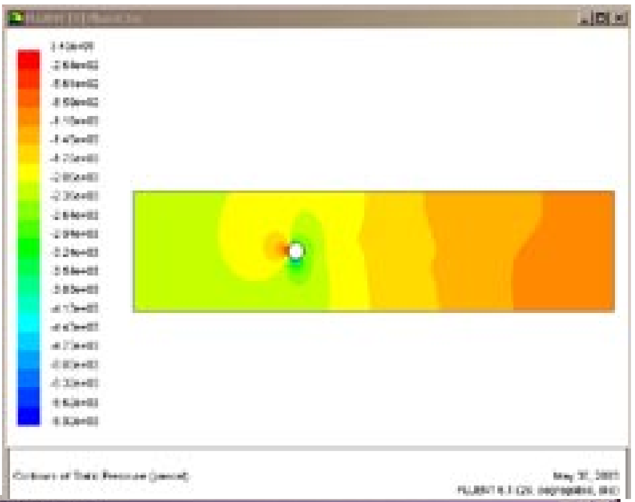


```
[illegible]
force vector: (1 0 0)
                 pressure     viscous      total
zone name        force        force        force
                 n            n            n
[illegible]
force vector: (0 1 0)
                 pressure     viscous      total
zone name        force        force        force
                 n            n            n
[illegible]
```

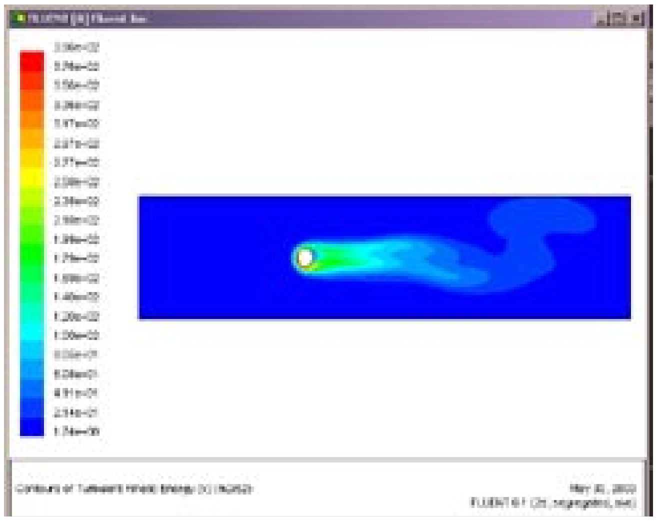


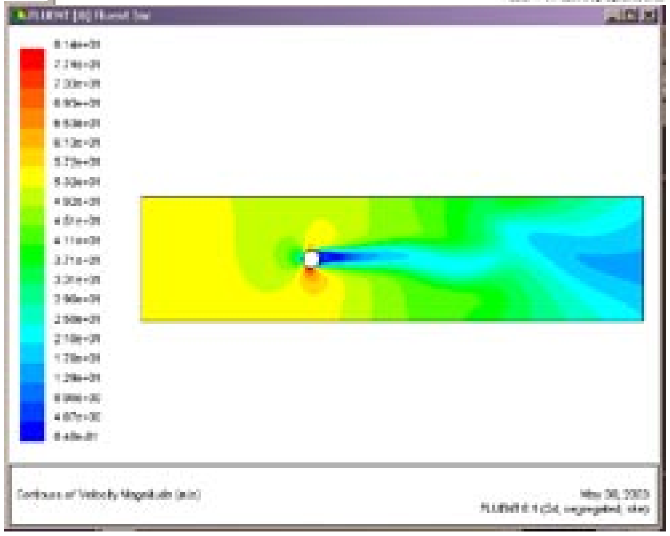




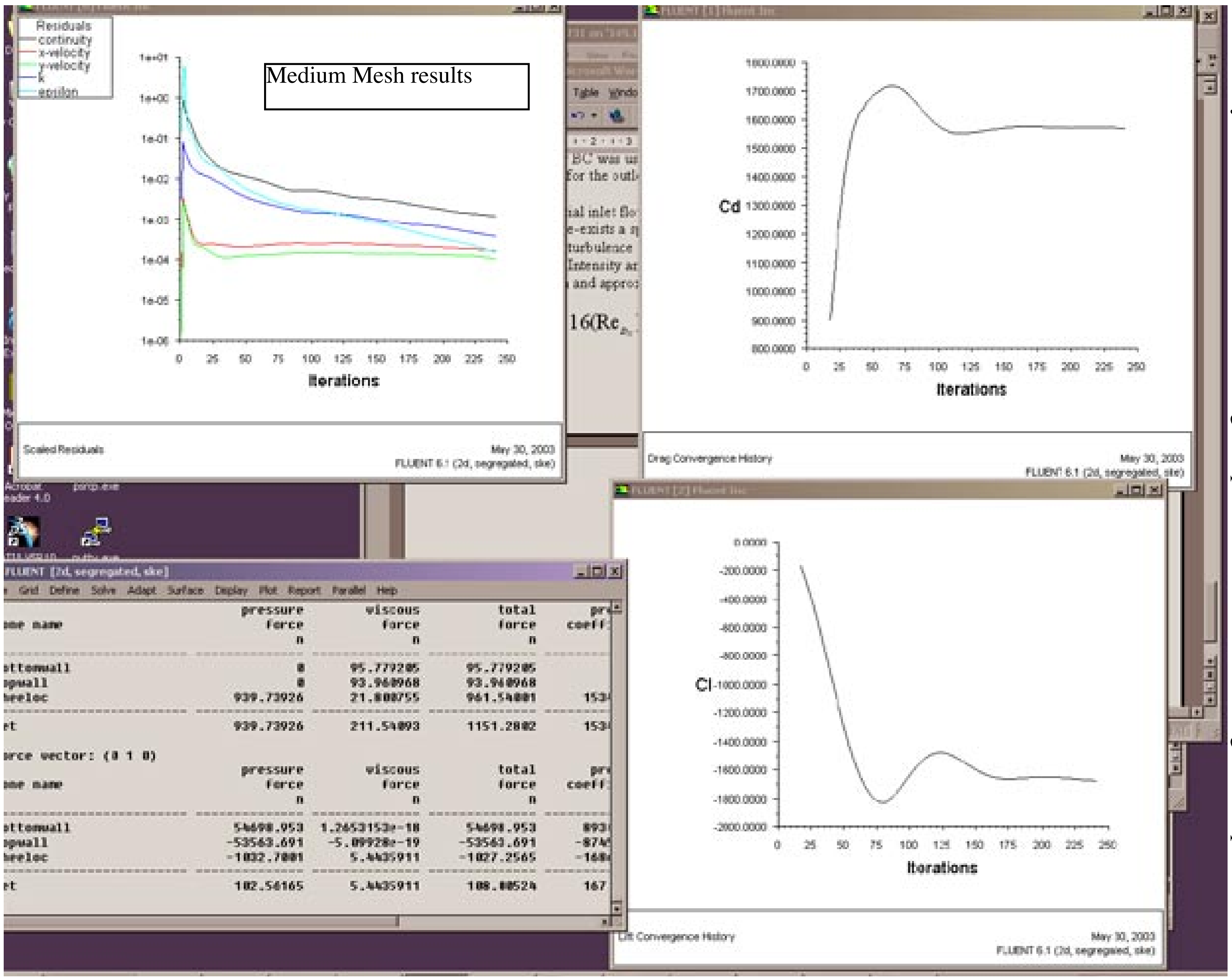


The contours show odd patterns in the rear perhaps due to the constant pressure assumption.



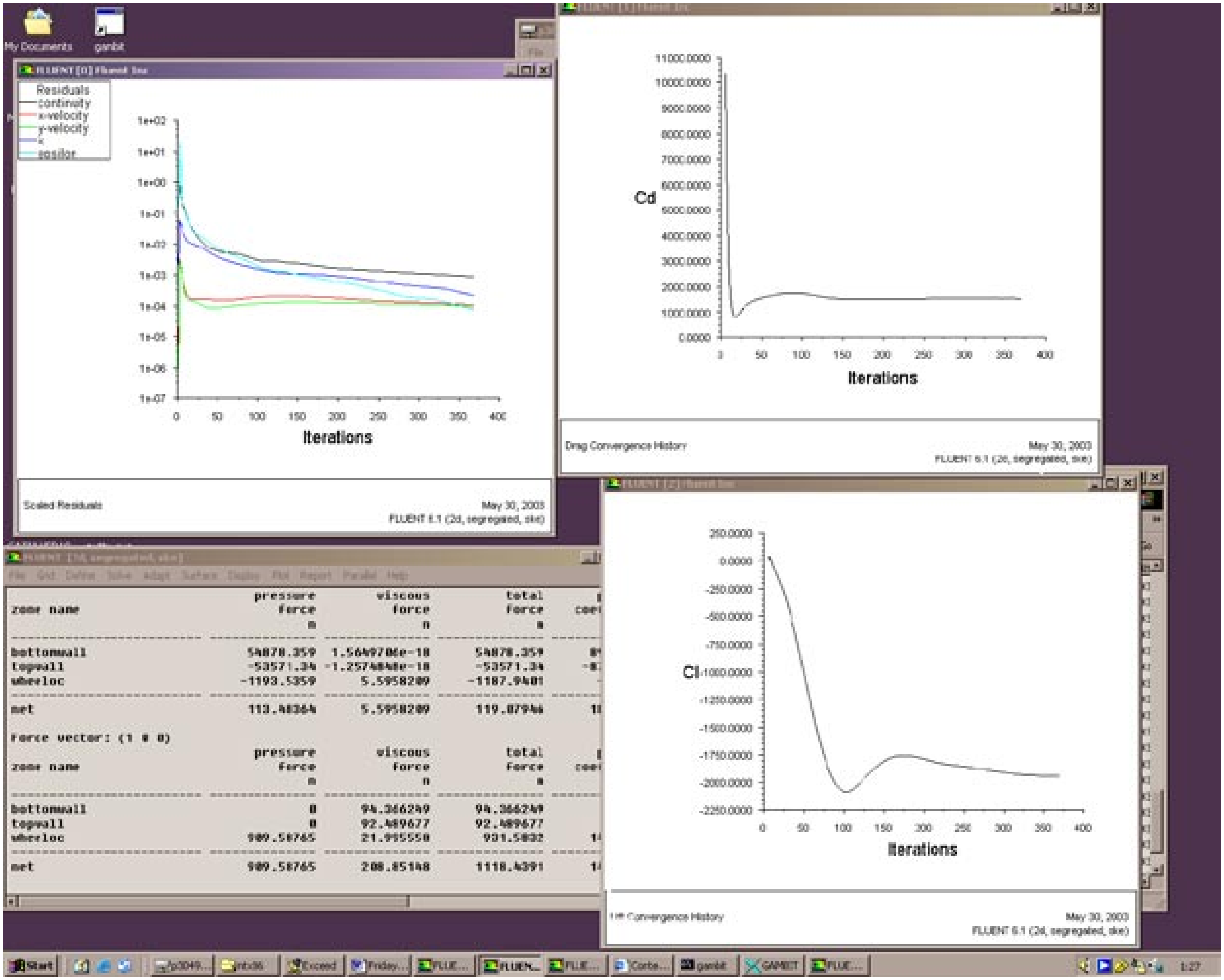


FINAL RESULTS PAGES. The next three pages represent my presentable results. These used the medium, fine, and superfine meshes described on page 16, and were run with the Standard K e model, enhanced wall treatment, initial conditions as specified in the report, and second order discretization. The y+ values for the mesh cells on the circle were still out of range (3500, 2500, and 1500), but the convergence and contour plots were consistent (see Results p.17-18). To obtain satisfactory y+ ranges, a entirely new mesh strategy will need to be devised. Additional meshes were run (with 100,000+ nodes) which had y+ values of 250, but the convergence pattern was not as reassuring as these presented below.



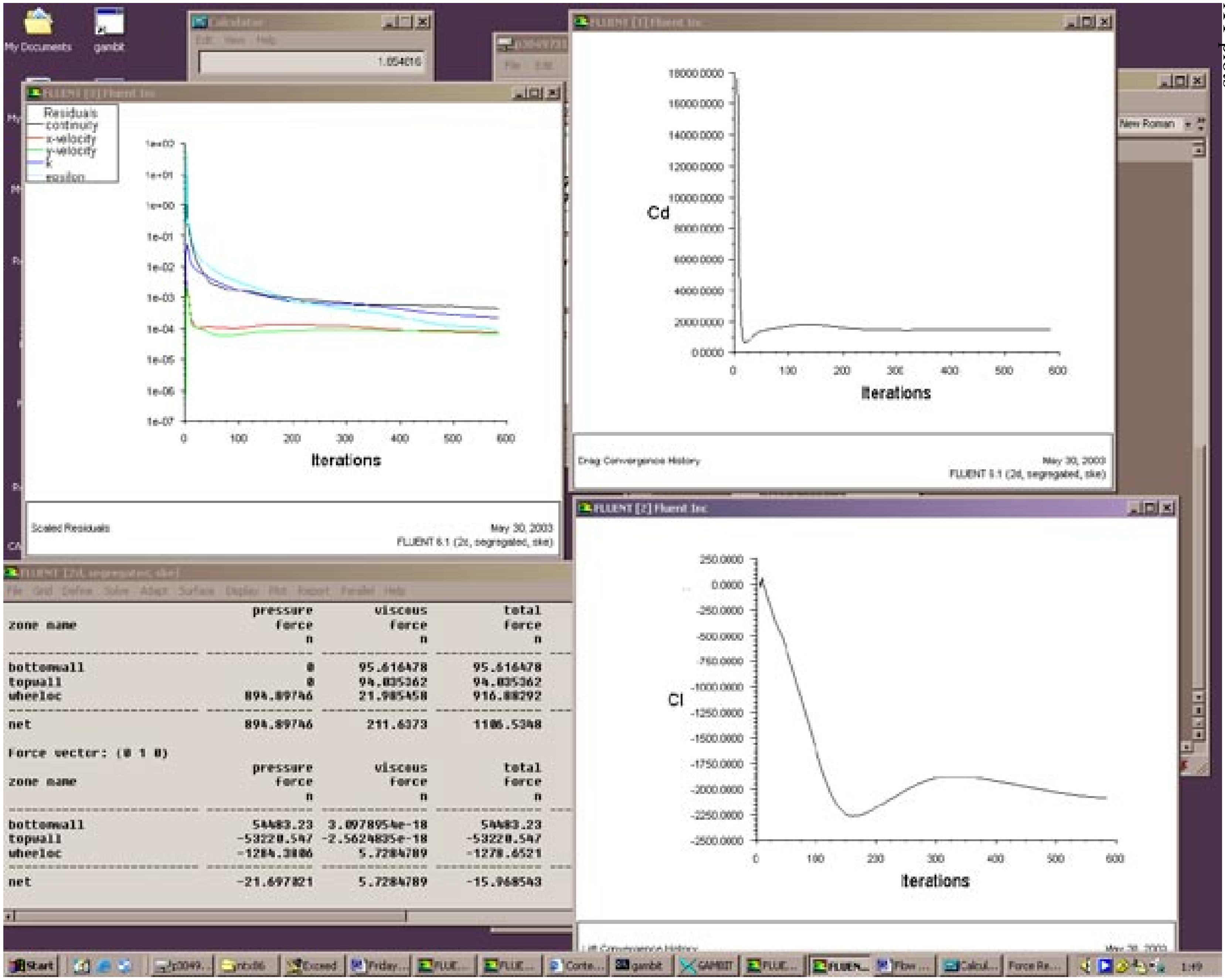
```
one name                 pressure          viscous            total       pr
                            force            force            force     coeff
                                n                n                n
-------------------------------------------------------------------------------
ottomwall                       0        95.779205        95.779205
opwall                          0        93.960968        93.960968
heeloc                  939.73926        21.800755        961.54001      153
-------------------------------------------------------------------------------
et                      939.73926        211.54093        1151.2802      153

orce vector: (0 1 0)
                         pressure          viscous            total       pr
one name                    force            force            force     coeff
                                n                n                n
-------------------------------------------------------------------------------
ottomwall               54698.953   1.2653153e-18        54698.953       893
opwall                 -53563.691   -5.09928e-19        -53563.691      -874
heeloc                 -1032.7001        5.4435911       -1027.2565      -168
-------------------------------------------------------------------------------
et                      102.56165        5.4435911        108.00524       167
```

Fine Mesh results. Standard K e model, enhanced wall treatment, initial conditions as specified in the report, and second order discretization. See Results p.16 and following for contour, vector, and XY plots.



```
                                pressure          viscous            total
zone name                          force            force            force
                                       n                n                n
------------------------------ --------------- ---------------- ----------------
bottomwall                      54878.359   1.5649706e-18        54878.359
topwall                         -53571.34  -1.2574846e-18        -53571.34
wheeloc                        -1193.5359       5.5958209       -1187.9401
------------------------------ --------------- ---------------- ----------------
net                             113.48364       5.5958209        119.07946

Force vector: (1 0 0)
                                pressure          viscous            total
zone name                          force            force            force
                                       n                n                n
------------------------------ --------------- ---------------- ----------------
bottomwall                              0       94.366249        94.366249
topwall                                 0       92.489677        92.489677
wheeloc                         909.58765       21.995558        931.5832
------------------------------ --------------- ---------------- ----------------
net                             909.58765       208.45148        1118.4391
```

“Superfine” mesh results. Standard K e model, enhanced wall treatment, initial conditions as specified in the report, and second order discretization. See Results p.16 and following for contour, vector, and XY plots



```
                                    pressure        viscous          total
zone name                              force          force          force
                                           n              n              n
-------------------------------- -------------- -------------- --------------
bottomwall                                 0      95.616478      95.616478
topwall                                    0      94.035362      94.035362
wheeloc                            894.89746      21.985458      916.88292
-------------------------------- -------------- -------------- --------------
net                                894.89746       211.6373      1106.5348

Force vector: (0 1 0)
                                    pressure        viscous          total
zone name                              force          force          force
                                           n              n              n
-------------------------------- -------------- -------------- --------------
bottomwall                          54483.23  3.0978954e-18       54483.23
topwall                           -53220.547 -2.5624835e-18     -53220.547
wheeloc                           -1284.3806      5.7284789     -1278.6521
-------------------------------- -------------- -------------- --------------
net                               -21.697021      5.7284789     -15.968543
```