Use of *Mathematica* as a Teaching Tool for (Computational) Fluid Dynamics and Transport Phenomena

Mark J. McCready

University of Notre Dame

mjm@nd.edu http://www.nd.edu/~mjm http://www.nd.edu/~chegdept

Outline

- Mixing
  - Importance of dimensionless groups
  - Experimental study of mixing of viscous materials
- Numerical solution to flow in a rectangular duct
  - *Mathematica* used to show finite difference vs finite element
- Mathematica notebooks on computational fluid flow and heat transfer problems
  - Boundary-layer flow, Falkner-Skan problem
  - Natural convection thermal boundary layer
- Mathematica notebooks for other fluid flow problems
  - Creeping flow past a sphere
  - Introduction to multiphase flows

Dimensionless groups

- A big theme throughout the Junior-Level Fluid Dynamics course is the importance of comparing competing or cooperating effects and how dimensionless groups inherently do this.
- To make the point we did a laboratory exercise on mixing and combined with dimensional analysis.

Mixing Experiment

Coloring Liquid Soap

mjm@nd.edu http://www.nd.edu/~mjm http://www.nd.edu/~chegdept

Toothpaste and Karo Syrup

mjm@nd.edu http://www.nd.edu/~mjm http://www.nd.edu/~chegdept

They can be mixed

mjm@nd.edu http://www.nd.edu/~mjm http://www.nd.edu/~chegdept

Mixing questions using Dimensional analysis

Mixing

Homework due 9/5. (There will be other questions.)

We would now like to do dimensional analysis of the mixing of two liquids. You can go to the learning center an check out some mixing instruments and some substances that can be mixed. These will be different fluids that requir different levels of effort and different mechanical actions to mix well.

1. Once you have gained some feel for mixing from the experiments, choose the variables that seem to be mos important to mixing for say, two liquids that will be mixed in a cylindrical tank with a single agitator.

2. Now do dimensional analysis, perhaps patterned after the work done above, to find the dimensionless groups tha will be important for this mixing problem.

3. Now consider that you have to design a flowing process in a 500 gallon tank to mix the same substances. Can yo convince me it will work?

Mixing Answers

- Viscous mixing

$$\frac{\mu_1}{\mu_2},\quad \frac{L}{R},\quad \frac{P}{L^3\mu\omega^2}$$

Dimensionless groups do not need to be on technical subjects

$$Cr \equiv \frac{How \quad Smart \quad You \, Are}{How \quad Smart \quad You \quad Think \quad You \quad Are}$$

mjm@nd.edu http://www.nd.edu/~mjm http://www.nd.edu/~chegdept

Dimensionless

Confucius Proverb

- He who knows not and knows he knows not is a child, teach him, Cr~1
- He who knows not and knows not he knows not is a fool, shun him, Cr<<1
- He who knows and knows not he knows is asleep, awaken him, Cr>>1
- He who knows and knows he knows is wise, follow him Cr~1

Dimensionless Proverb

- Child ~ Wise person

Dimensional Analysis NoteBook

Buckingham Pi method (dimensional analysis)

This notebook has been written in *Mathematica* by

Mark J. McCready
Professor and Chair of Chemical Engineering
University of Notre Dame
Notre Dame IN 46556
USA

mjm@nd.edu
http://www.nd.edu/~mjm/

Version: 8/24/00
More recent versions of this notebook should be available at the web site:
http://www.nd.edu/~mjm/*dimensional.analysis*.nb

Mathematica Note Books

Many of these (and other useful materials) are also available from MathSource, at the Wolfram Research website. This and other courses that use Mathematica materials can be found at the Mathematica Courseware web site.

A simple Mathematica primer,
Mathematica_primer.1.nb. (Notebook)
Mathematica Primer(html)

A basic introduction to dimensional analysis including physical motivation and how to solve pipe flow.
dimensional.analysis.nb. (Notebook)
dimensional.analysis.html (html)

A simple primer on why we use log-log plots and what they mean,
Primer on log-log and semilog plots. (Notebook)
Primer on log-log and semilog plots(html)

An exhaustive solution of the lubricated flow example ("core-annular flow") from Middleman 3.2.3, pp79-82).
It demonstrates a number of Mathematica features and several important basic ideas from this course,
lubricatedflow.nb (notebook format)
lubricatedflow.html (html, this is not as good as the Mathematica version, but you don't need MathReader.)

This one shows how to use the chain rule to nondimensionalize differential equations. It also makes a point that the Resulting dimensionless terms are of order 1.
Making a differential equation dimensionless (Notebook format)
Making a differential equation dimensionless (html)

mjm@nd.edu http://www.nd.edu/~mjm http://www.nd.edu/~chegdept

Finite Difference and Finite Element solution to flow in a Rectangular Duct

The numerical code for this problem is adapted from a Fortran program and discussion given by C. A. J. Fletcher (1991) Computational Techniques for Fluid Dynamics, Springer vol.1 p 112.

Homework problem

1. Plot the solution for different values of the the aspect ratio, ba. Explain what is happening.
2. What do you have to do to get accuate answers as ba is changed ??
3. If you use the finite element method with nx=6 and ba>=6, there will be two cells of flow.

How do you know that this is incorrect ? One of the important issues that you may face when using numerical codes is that they may not be correct !!

4. It is rather significant that the solution depends only on the geometric parameter, ba. There is no qualitative change in the flow field with flow rate. The group , (-eta/b^2/DPDZ) scales the velocity to make this possible, discuss this physical meaning of this group.

http://www.nd.edu/~mjm/RectangularDuct.nb

mjm@nd.edu http://www.nd.edu/~mjm http://www.nd.edu/~chegdept

Objectives for "CFD"

- BS engineers, whether we like it or not, will increasingly be using computational packages and analytical instruments that are "turn key" (they don't understand how they work). We need to instill in them both a <u>healthy skepticism</u> that they need to verify the answers, and <u>enough fundamental understanding</u> of the different subject so that they can.

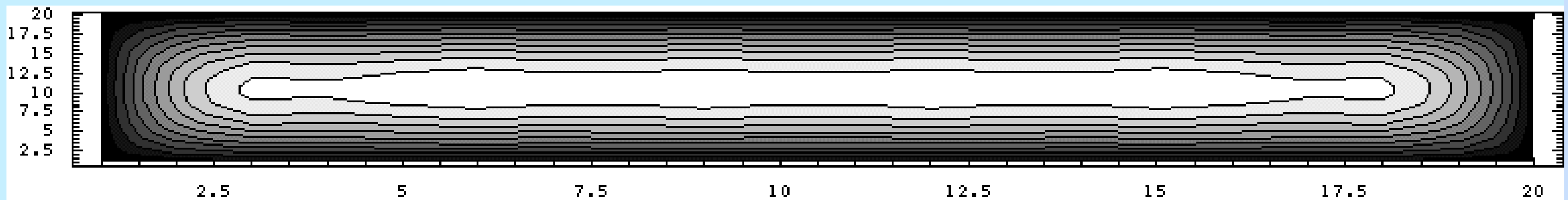
Homework problem

1. Plot the solution for different values of the the aspect ratio, ba. Explain what is happening.
2. What do you have to do to get accuate answers as ba is changed ??
3. If you use the finite element method with nx=6 and ba>=6, there will be two cells of flow.
How do you know that this is incorrect ? One of the important issues that you may face when using numerical codes is that they may not be correct !!
4. It is rather significant that the solution depends only on the geometric parameter, ba. There is no qualitative change in the flow field with flow rate. The group , (-eta/b^2/DPDZ) scales the velocity to make this possible, discuss this physical meaning of this group.

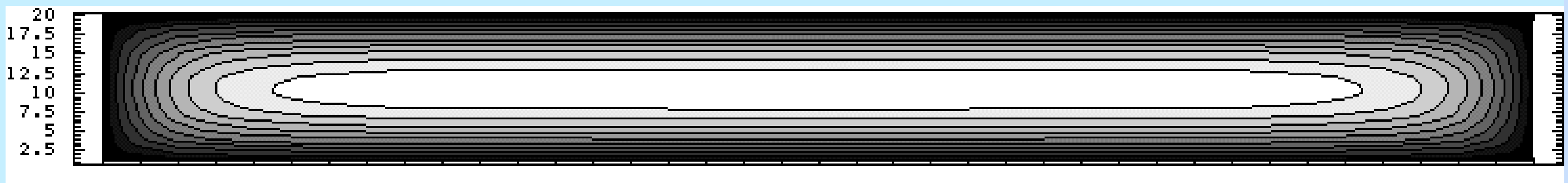
As the aspect ratio is varied from one, the solution become increasingly inaccurate. The finite different method works better

Comparison of solutions

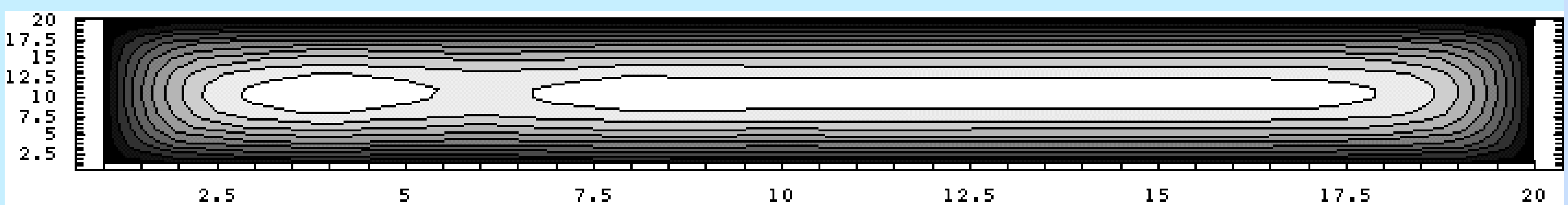
In[138]:= `ListContourPlot[Table[vex[i, j], {i, 1, nx}, {j, 1, nx}], AspectRatio → 1/bar]` Analytical



In[143]:= `ListContourPlot[Table[va[i, j], {i, 1, nx}, {j, 1, nx}], AspectRatio → 1/bar]` Finite diff.



In[151]:= `ListContourPlot[Table[va[i, j], {i, 1, nx}, {j, 1, nx}], AspectRatio → 1/bar]` Finite element



Boundary-Layer flows

Boundary-layer flow over a flat plate and wedge

This notebook has been written in *Mathematica* by

Mark J. McCready
Professor and Chair of Chemical Engineering
University of Notre Dame
Notre Dame IN 46556
USA

Mark.J.McCready.1@nd.edu
http://www.nd.edu/~mjm/

http://www.nd.edu/~mjm/boundarylayer.nb

Version: 6/19/00

Boundary-Layer Flow (cont)

Summary

This notebook examines the boundary-layer flow over a flat plate and a wedge. The solution is given using numerical scheme and several important physical aspects of boundary layers are elucidated.

Reference

Sample Homework problem for this note book

1. How does the boundary layer thickness change with Reynolds number ??
2. How does the wall shear stress change with Reynolds number ??
3. What are two physical characteristics of boundary layers ??
4. Run the code to find is the approximate location of the outer edge of the boundary layer.
5. Run the code to find the value of the stress at the wall.
6. Run the code to show how the boundary-layer thickness and shear stress change when the imposed pressur gradient changes (i.e. the wedge problem)

Boundary Layer Flow

Flow past a flat plate

- Governing equations and problem set up
- Numerical Solution
- Plots of the results

mjm@nd.edu http://www.nd.edu/~mjm http://www.nd.edu/~chegdept

Problem set-up

Governing equations and problem set up

Boundary layers are regions where two competing transport effects are about the same order of magnitude. It is expected that the gradients are also "large" in this region. This *Mathematica* program gives numerical code for the boundary velocity field for flow over a flat plate and below, the Falkner-Skan problem for flow over a wedge.

The **boundary layer equations** are

$$\frac{\partial v_x}{\partial x} + \frac{\partial v_y}{\partial y} = 0$$

$$\rho \left(v_x \frac{\partial v_x}{\partial x} + v_y \frac{\partial v_x}{\partial y} \right) = - \frac{\partial P}{\partial x} + \eta \frac{\partial^2 v_x}{\partial y^2}$$

If the plate is flat, there is no change in pressure because of a change in the fluid local velocity, say for example if the flow were converging or diverging. You might have a slight pressure *drop* in a closed system, but this can usually be neglected.

Therefore, $\frac{\partial P}{\partial x} = 0$.

Problem set-up continued

Using the variable changes given in Denn, pp292-293, the boundary layer equations become simply

f''' + f f''/2 = 0.

The primes (') denote derivatives with respect to the similarity variable. This nonlinear ode does not have a known analytical solution. However, it can be easily solved numerically.

Numerical solutions for ode's are often done by creating a system of first order odes. This is done by defining

y1 = f,
y2 = f'
y3 = f'',

These have to be related.

Thus we have
y1' = y2 (by definition)
y2' = y3 (by definition)
y3' = -1/2*y1*y3 (which is from the original ode.)

In the first part I solve the flat plate problem separately using a crude numerical scheme that usually converges, although not real fast. It is a shooting method. I have picked y =15 as the end of the integrate. The NDSolve routine uses a Runge-Kutta method with built in step size adjustment.

Flow over a flat plate solution

■ **Numerical Solution**

```
fppinit = 1;
eps = 1;

While[Abs[eps] > .000001,
 zz = NDSolve[{y1′[x] == y2[x], y2′[x] == y3[x],
     y3′[x] == - 1/2 y1[x] y3[x], y1[0] == 0, y2[0] == 0,
     y3[0] == fppinit}, {y1[x], y2[x], y3[x]}, {x, 0, 15}];
 xz = (zz /. x → 15)[[1, 2, 2]]; eps = 1 - xz; fppinit = fppinit/(1 - eps);
 Print["U = ", xz, "     error=  ",
  eps, "    f''[0]=   ", fppinit];]
```

U = 2.08541 error= −1.08541 f''[0]= 0.479523

U = 1.27761 error= −0.277607 f''[0]= 0.375329

U = 1.08509 error= −0.0850888 f''[0]= 0.345897

U = 1.02759 error= −0.0275943 f''[0]= 0.336608

U = 1.00911 error= −0.00911473 f''[0]= 0.333568

U = 1.00303 error= −0.00302905 f''[0]= 0.332561

U = 1.00101 error= −0.00100866 f''[0]= 0.332225

U = 1.00034 error= −0.000336106 f''[0]= 0.332114

U = 1.00011 error= −0.000112022 f''[0]= 0.332077

U = 1.00004 error= −0.0000373392 f''[0]= 0.332064

U = 1.00001 error= −0.0000124462 f''[0]= 0.33206

U = 1. error= -4.1487×10^{-6} f''[0]= 0.332059

U = 1. error= -1.38289×10^{-6} f''[0]= 0.332058

U = 1. error= -4.60962×10^{-7} f''[0]= 0.332058

Flow past a wedge

We again need to make a system of first order ODE's . These are

y1' = y2
y2' = y3
y3' = - (m+1)/2 y1 y3 + m*(1-y2^2)

This time I use a real shooting method, with a pseudo Newton-Raphson iteration. It works fo
most cases. If the angle is too negative, the stress will go through 0, this means that there i
no layer and thus it is not surprising that the answer blows up. Of course if we look at th
physical situation, we might be surprised that we can get any solution for negative β. Th
trend is correct and it is interesting to examine the case of a diverging flow.

- **Numerical solution**
- **Plots of the results at different angles**

Flow past a wedge

Flow past a wedge

Governing equations and problem set up

From Denn the flow past a wedge (figure 15-1) is given by

$U(x) = A\, x^m$, m=beta/(2 pi-beta)

Recall that the pressure gradient, $\frac{\partial p}{\partial x}$, will change as -V(x) V'(x). This gives

$$\rho \left(v_x \frac{\partial v_x}{\partial x} + v_y \frac{\partial v_x}{\partial y}\right) = V(x)\, V'(x) + \eta \frac{\partial^2 v_x}{\partial y^2}$$

****** **Thus, the flow past a wedge is a model problem for telling how the boundary layer will change as the pressure gradient changes**. ******

Denn tells us that the equation can be reduced to a nonlinear ode again taking advantage o similarity of solution profiles. This gives

$$f''' + (m+1)/2\, f\, f''/2 + m\,(1-f'^2) = 0.$$

If m=0, the equation is the same as for the flat plate.

Solution code

■ **Numerical solution**

Note that you will want to run this several times changing the value of β over the possible range. If β=0, the result is the same as a flat plate.

```
fppinit = 1;
eps = 1;
```

$$\mathtt{beta} = 0 * \frac{\pi}{2};$$

$$\mathtt{m} = \frac{\mathtt{beta}}{2\pi - \mathtt{beta}};$$

```
youter = 10;
```

$$\mathtt{angleindegrees} = \mathtt{N}\left[\frac{\mathtt{beta}\ 180}{\pi}\right]$$

0.

need to run one time to get a value for the outer f'[infinity]

```
zz = NDSolve[{y1'[x] == y2[x], y2'[x] == y3[x],
    y3'[x] == -1/2 (m + 1) y1[x] y3[x] - m (1 - y2[x]^2),
    y1[0] == 0, y2[0] == 0, y3[0] == fppinit},
   {y1[x], y2[x], y3[x]}, {x, 0, youter}];
xz = (zz /. x -> youter)[[1, 2, 2]];
eps = 1 - xz;
Print["U = ", xz, "     error=   ",
   eps, "    f''[0]=   ", fppinit];
```

U = 2.08541 error= −1.08541 f''[0]= 1

```
fppinitold = fppinit;
fppinit = fppinit + .01;
xzold = xz;

While[Abs[eps] > .000001,
 zz = NDSolve[{y1'[x] == y2[x], y2'[x] == y3[x],
     y3'[x] == -1/2 (m + 1) y1[x] y3[x] - m (1 - y2[x]^2),
     y1[0] == 0, y2[0] == 0, y3[0] == fppinit},
    {y1[x], y2[x], y3[x]}, {x, 0, youter}];
 xz = (zz /. x -> youter)[[1, 2, 2]]; eps = 1 - xz;
 xcorrect = ((1 - xz) (fppinit - fppinitold))/(xz - xzold);
 fppinitold = fppinit; xzold = xz;
 fppinit = fppinitold + xcorrect;
 Print["U = ", xz, " xzold= ", xzold,
  "     error=   ", eps, "    f''[0]=   ",
  fppinit, " xc= ", xcorrect]; xzold = xz;]
```

U = 2.09929 xzold = 2.09929 error=
−1.09929 f''[0]= 0.217988 xc = −0.792012

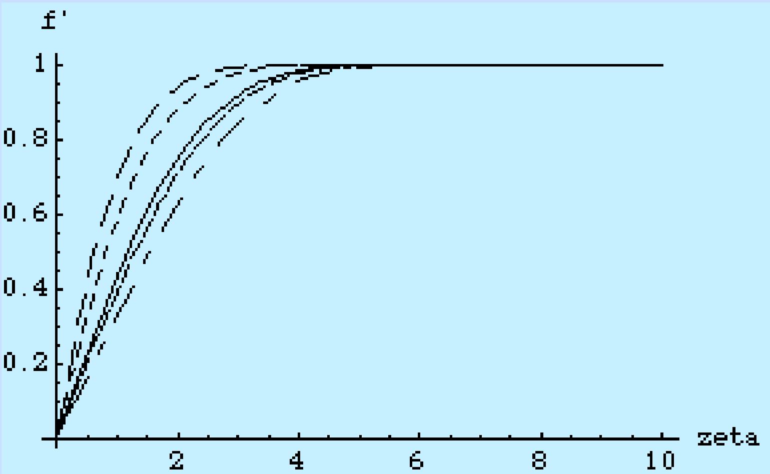
U = 0.755344 xzold = 0.755344 error=
0.244656 f''[0]= 0.362168 xc = 0.144181

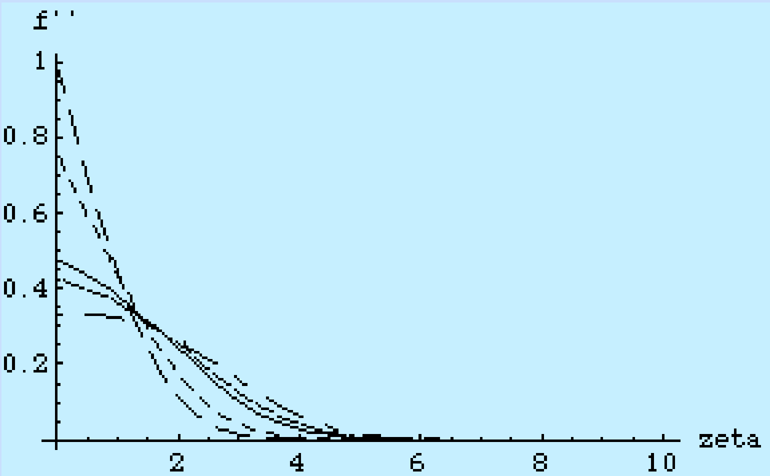
U = 1.05957 xzold = 1.05957 error=

mjm@nd.edu

.nd.edu/~chegdept

Plots at different angles





Solution of natural convection boundary - layer flow near a heated flat plate

This notebook has been written in *Mathematica* by

Mark J. McCready
Professor and Chair of Chemical Engineering
University of Notre Dame
Notre Dame IN 46556
USA

mjm@nd.edu
http://www.nd.edu/~mjm/

Version: 5/25/00
More recent versions of this notebook may be available at the web site:
http://www.nd.edu/~mjm/thermal_boundarylayer.nb

Problem overview

Derivation of the ODE's from the original PDE's.

Correlation of the experimental data

Numerical solution of the coupled ODE's that result after a similarity variable is introduced to the natural convection boundary - layer equations for flow near a flat plate.

Conclusions

Problem overview

- Physical situation of interest
- High Grashof number heat transfer
- Boundary-layer physics
- The basic mass, momentum and energy equations for a free convection boundary - layer

- **Numerical solution uses a shooting method and a Runge - Kutta Integration.**
  Reference: Numerical Recipes in Fortran, See also the notes from ChEg 258, (D. T. Leighton).
  http://www.nd.edu/~dtl/cheg258/cheg258-1999/notes/137/overheads.html

Return to conclusions

The key equations for the shooting method are:

$\boldsymbol{\alpha} \cdot \delta\, \mathbf{V} = -\mathbf{F}$

$\mathbf{V}\text{new} = \mathbf{V}\text{old} + \delta\mathbf{V}$

where $\boldsymbol{\alpha}$ is the Jacobian Matrix obtained by varying the initial guesses for the unknown initial conditions, **F** i the error in the solution produced from the current initial guesses, **V**old, and δ**V** is the correction to the initia guesses for the next step.

This turns out to be a touchy calculation. Here are a set of initial conditions that give a solution with some trouble. You will find that higher *Pr* is harder and that if youter is too large, the calculation does not work.

```
finit = {{.8},{-.25}};
pr=10; (*Prandtl number*)
youter=5.5; (* outer value of η *)
deltv = .0006;(*  increment on the initial guesses used to generate the Jacobian*)
```

- Here is the numerical solution
  - Initialize some variables.
  - We need to run through the integration once to get a first values for the error.
  - Here is the main iteration loop.
  - Here are some plots of the results

Conclusions

1. For natural convection flows where the Grashof number is larger, a boundary layer can be expected close to a solid surface.

2. For situations where a natural convection boundary-layer is occurring, heat transfer will be governed by coupled energy and momentum equations.

3. For transport processes that occur on a semi-infinite domain, where there is no geometric length scale, it is often possible to define a (dimensionless) similarity variable that contains the natural length scale.

4. It is often possible to reduce PDE's to ODE's through simplifications made possible by use of the similarity variable.

5. From the scaling identified by the similarity variable, it is often possible to predict the macroscopic behavior without solving the differential equations. In this case $Nu \sim Gr^{1/4}$.

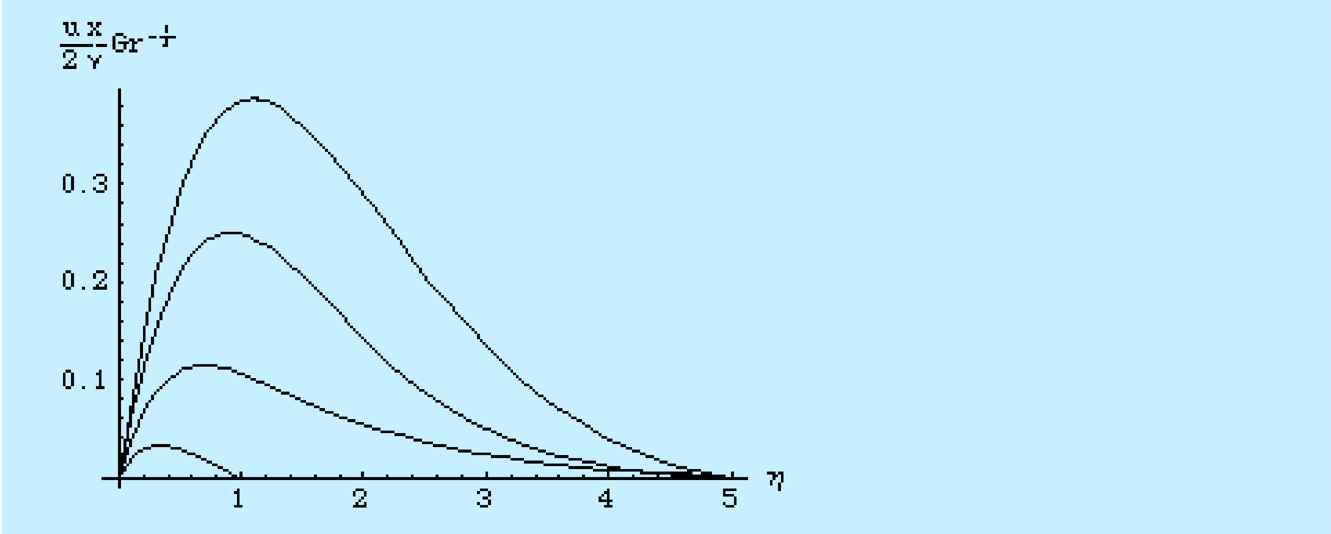
6. This prediction agrees with the recommended correlation for high Gr heat transfer.

7. The coupled, nonlinear ODE's can be readily solved with a shooting method.

8. Note the shape of the temperature profile and the location of the maximum velocity.

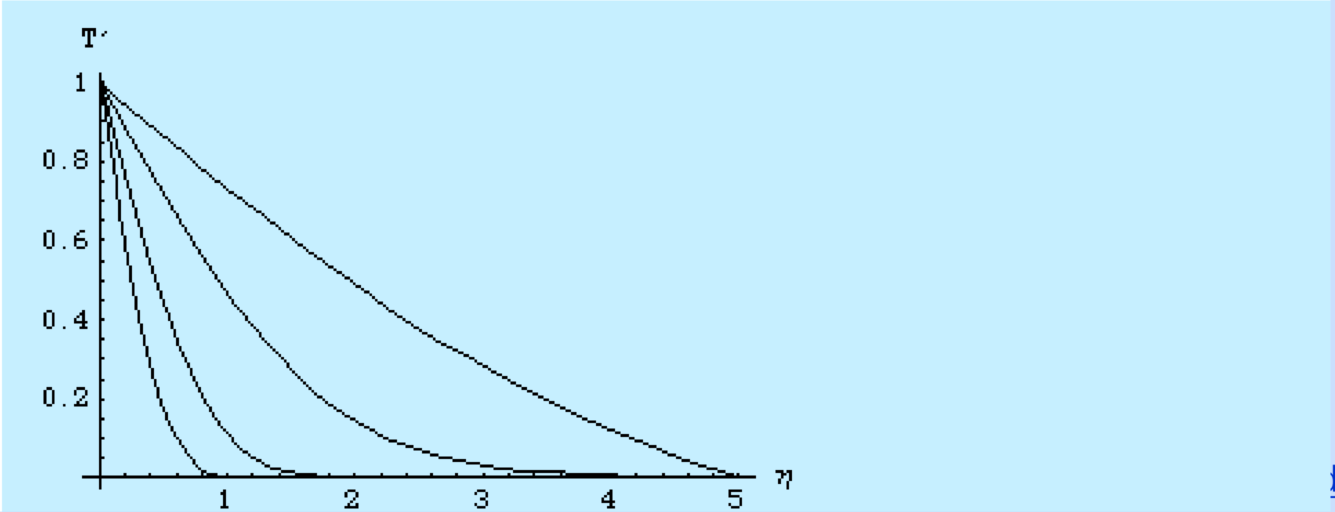
mjm@nd.edu http://www.nd.edu/~mjm http://www.nd.edu/~chegdept

```
Show[Upr10, Upr01, Upr100, Upr1];
```



return to conclusions

```
Show[Tpr10, Tpr01, Tpr100, Tpr1];
```



mjm@ ... dept

- **The basic mass, momentum and energy equations for a free convection boundary - layer**

Continuity

$$\frac{\partial u(x, y)}{\partial x} + \frac{\partial v(x, y)}{\partial y} = 0$$

Momentum

$$u(x, y)\frac{\partial u(x, y)}{\partial x} + v(x, y)\frac{\partial u(x, y)}{\partial y} = g\,(T(x, y) - T0)\,\beta + \nu\frac{\partial^2 u(x, y)}{\partial y^2}$$

Energy

$$u(x, y)\frac{\partial T(x, y)}{\partial x} + v(x, y)\frac{\partial T(x, y)}{\partial y} = \alpha\frac{\partial^2 T(x, y)}{\partial y^2}$$

return to conclusions

These equations are coupled, meaning you cannot solve any of them without simultaneously solving the othe two. Further, the momentum equation is nonlinear.

Demonstration of the effect of flow regime on pressure drop in multifluid flows

Summary

This notebook is intended to give a first introduction to multifluid flows through the use of "model" *flow regimes* calculated fror exact solutions for laminar flow in different configurations. By comparing pressure drop over a range of flow rates for thes different configurations, that **show differences of factors of up to 30**, the importance of knowing the flow regime is demonstrated Insight into the physical reasons for the variation in pressure drop with flow rates and physical properties is given.

http://www.nd.edu/~mjm/Effect.of.Flow.Regime.pdf

Preview of Major points

Flow rate pressure drop relations for the three regimes.

Pressure drop comparisons

Recap of Major points

Suggestions for future study

mjm@nd.edu http://www.nd.edu/~mjm http://www.nd.edu/~chegdept

Recap of Major points

(same as the preview above)

[BACK to Preview]

We have shown, using simple models for flow regimes, stratified, slug and dispersed, that

1. The *qualitative* as well as the *quantitative* behavior of multiphase flows will change as the ratios of flow rates and physica properties change.

2. The pressure drop predictions differ substantially with flow configuration. The pressure drop for dispersed flow wa predicted to be a factor of 35 higher than for slug flow in one case and a factor of 20 greater than stratified flow for another case This key result is true for process flows and makes correct prediction of the flow regime crucial to successful design of multiflui systems. Most engineering designs cannot stand an uncertainty of a factor of 2 in the main design variable, let alone 30.

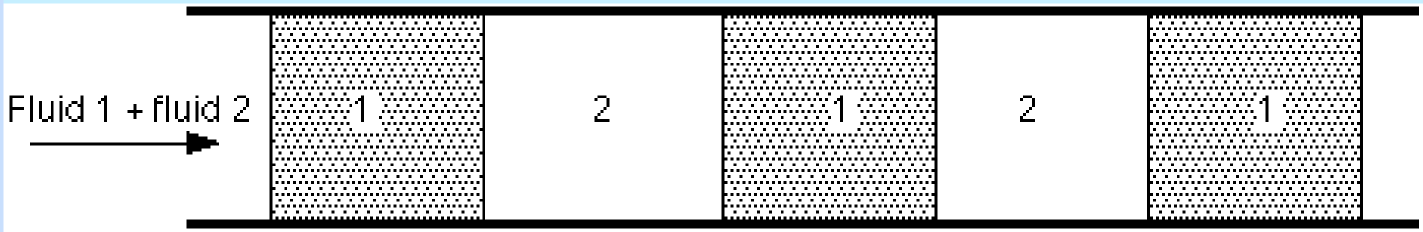
3. Stratified flow is the most efficient configuration, of the three tested here (compare stratified/slug, dispersed/stratified), fo fluid transport when the more viscous fluid has a higher flow rate. This is due to the lubricating effect of the less viscous fluid tha reduces shear in the more viscous fluid. This is the basis of lubricated pipeline transport of heavy oil (See D. D. Joseph and Y. Y Renardy, *Fundamentals of Two-Fluid Dynamics*, Springer-Verlag, 1993, Vol. 2.) If the more viscous fluid is present in lesse amounts the advantage is lost because it is subjected to high shear and acts to reduce the available flow area for the less viscou fluid.

4. The loss of lubricating effect of a less viscous fluid in stratified flow can cause a region where *decreasing* the flowrate o the less viscous fluid, *increases* the pressure drop (click for specifics about *retrograde* pressure drop) -- contrary to physica intuation gained from most other flow situations.

5. The specific conclusions for dispersed/slug, dispersed/stratifed and stratified/slug can be accessed directly.

6. The reason for the differences in the pressure drop with configuration for the examples in this notebook is that th dissipation is altered. Differences in dissipation arise primarily when fluids of different viscosities are located in regions o different stress. We also find that changing the effective flow area (i.e., by having a stratified region of more viscous fluid) for th fast moving fluid changes the dissipation significantly. These general observations should hold for either laminar flow (as show here) or turbulent flow. However, if the primary contributions to pressure drop are from fluid acceleration, or gravity, then th m pressure drop differences caused by the flow regime could be less than shown here. Examples are unsteady or transient flows developing flows or vertical flows.

Slug (Alternating)



View the slug flow movie, Slug Flow

Stratified flow

Two-layer, horizontal two-fluid flow

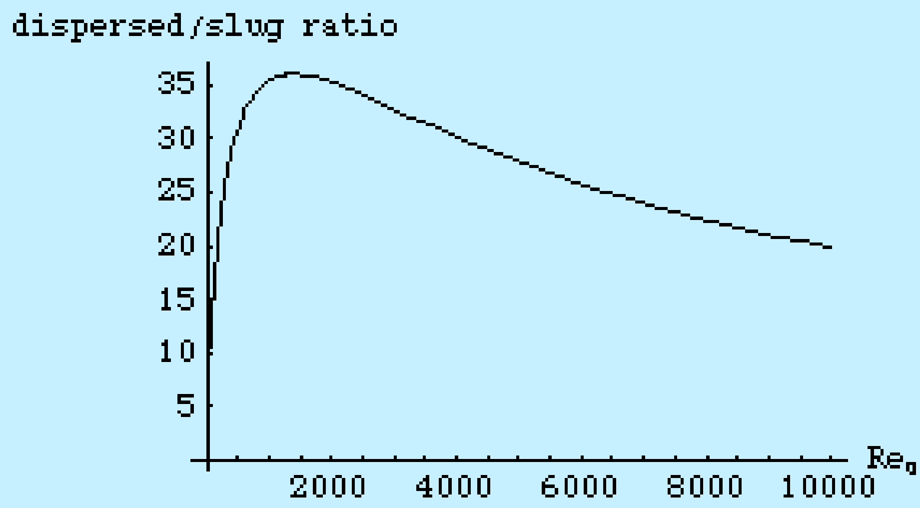
[BACK to Preview]
[BACK to Recap]

mjm@nd.edu http://www.nd.edu/~mjm http://www.nd.edu/~chegdept

Surprising result

- **Pressure drop comparison for dispersed vs. slug flow**

```
Plot[Evaluate[dispslugratio /. {ReL -> 100, μL -> .01 , μG -> .00018,
          ρL -> 1,  ρG -> 1 / 899, H -> 1}],
  {ReG, 1, 10000}, AxesLabel -> {"ReG", "dispersed/slug ratio"}];
```



Creeping flow past a stationary sphere

This notebook has been written in *Mathematica* by

Mark J. McCready
Professor and Chair of Chemical Engineering
University of Notre Dame
Notre Dame IN 46556
USA

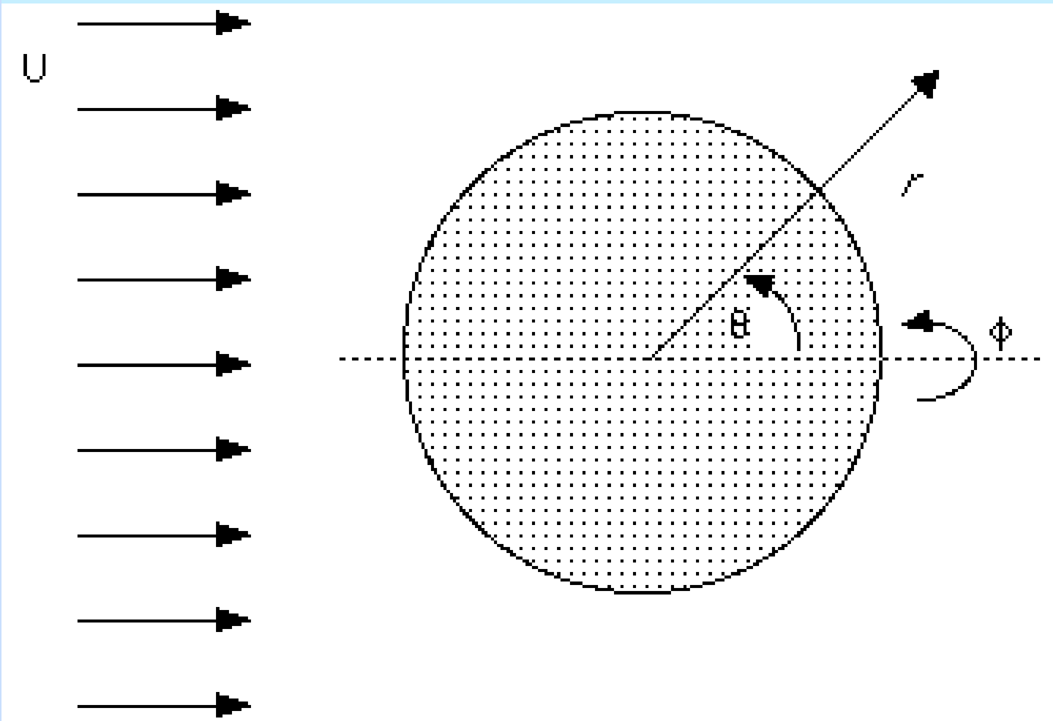
mjm@nd.edu

http://www.nd.edu/~mjm/

Version: 8/8/00
More recent versions of this notebook should be available at the web site:
http://www.nd.edu/~mjm/creepingsphere.nb

This notebook shows how to solve creeping (intertialess) flow past a stationary sphere (Stokes's Problem)

Reference: S. Middleman (1999) *An Introduction to Fluid Dynamics*:Principles of Analysis and Design, Wiley pp 166-171.



mjm@nd.edu http://www.nd.edu/~mjm http://www.nd.edu/~chegdept

Flow past a sphere problem

Problem of interest
Learning Objectives
Mathematical Formulation
Solution of the equations with the boundary conditions
Examination of the solution
Conclusions

mjm@nd.edu http://www.nd.edu/~mjm http://www.nd.edu/~chegdept

Learning Objectives

- **Physical Issues**

1. We are restricting the problem to the case where inertia forces are much weaker than viscous forces Thus we expect that the interia terms of the Navier-Stokes equation should be much smaller than the viscous and pressure terms and that thus they can be neglected. For this case the Reynolds number is very small. If we make the equations dimensionless all terms are no larger in magnitude than about unity. Thus the parameters that appear in these equations, and which can have values much different from on, determine which terms are needed for the solution. In the nondimensional equations, the Reynolds number multiplies the intertia terms and these will consequently be neglected in the solution.

2. Since the Reynolds number is small, the fluid goes only where specifically pushed and it will stop if the forcing is stopped. Thus the geometry of the flow field is determined by the boundaries.

3. Because viscous forces dominate the flow field, the fluid can never accelerate above the free stream value even if an obstacle causes the fluid to be squeezed. Thus the velocity in the region of the sphere just slows down and then returns to the free stream value.

4. Both normal stresses and tangential stresses contribute to the drag on the sphere. These can be termed form drag and skin drag.

5. Consistent with the fluid not accelerating, the pressure never increases above the free stream value. The fluid has no inertia that would cause a pressure increase as the fluid slows down.

6. The velocity decays slowly (as $\frac{1}{r}$) and thus the disturbance is felt very far away from the sphere. This makes it difficult to do a real experiment, in a reasonable size container, that allows that sphere to fall at a speed specified by the drag that is predicted from the analysis here. The very high Reynolds number case decays much faster.

Conclusions

1. Creeping flow is a term used for a flow that have effectively no inertia. In this case the inertia terms are neglecte and the solution is obtained from the resulting linear equations. The Reynolds number is very much smaller than unity.

2. The solution technique involves using a solution form that is deduced from boundaries of the flow field, farawa from the sphere.

3. Because viscous forces dominate the flow field, the fluid can never accelerate above the free stream value even if a obstacle causes the fluid to be squeezed. Thus the velocity in the region of the sphere just slows down and then return to the free stream value.

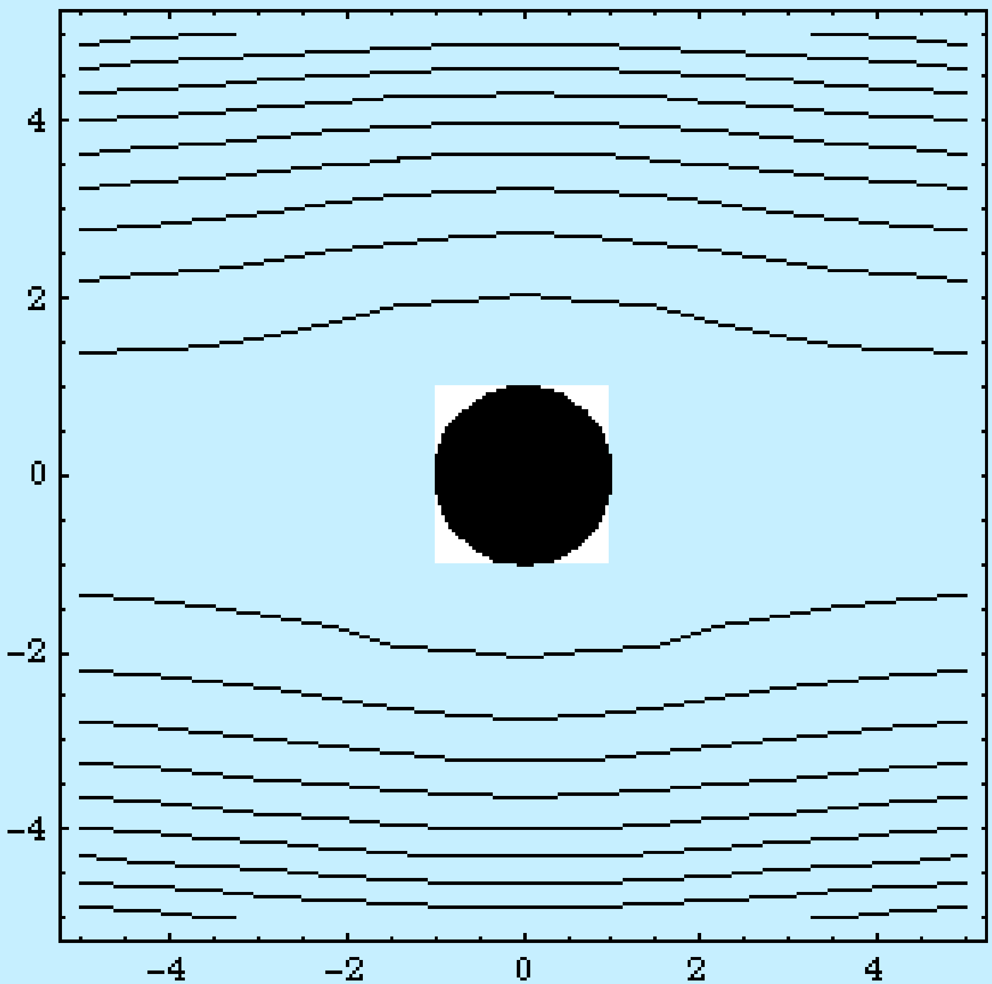
4. Both normal stresses and tangential stresses contribute to the drag on the sphere. These can be termed form drag an skin drag. Note that both of these are **linear** in the velocity (consistent with the linear governing equations) and flui viscosity. The density, and thus the Reynolds number, does not appear.

5. Consistent with the fluid not accelerating, the pressure never increases above the free stream value. The fluid has n inertia that would cause a pressure increase as the fluid slows down.

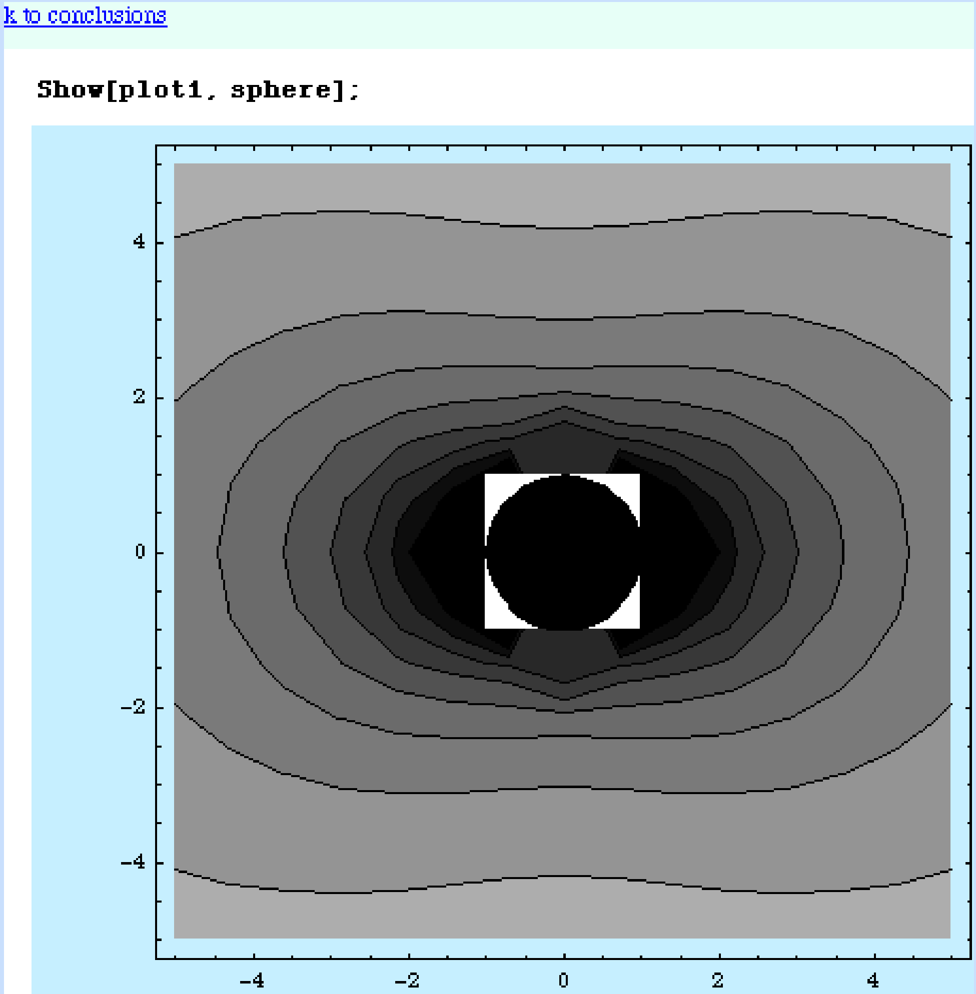
6. The velocity decays slowly (as $\frac{1}{r}$) and thus the disturbance is felt very far away from the sphere. This makes i difficult to do a real experiment, in a reasonable size container, that allows that sphere to fall at a speed specified by th drag that is predicted from the analysis here. The very high Reynolds number case decays much faster.

mjm@nd.edu http://www.nd.edu/~mjm http://www.nd.edu/~chegdept

```
Show[plot5, sphere, DisplayFunction → $DisplayFunction];
```



Velocity field magnitude



Conclusions

- Some elements of mixing are incorporated into the fluid dynamics course
  - Laboratory experiment
  - Dimensional analysis
- The main idea we attempt to convey about computational fluid dynamics is that it is wonderful if it works, but make sure your solution is correct.
  - Strategy is like using different excess Gibbs Free Energy models to design distillation columns with a process simulator

Conclusions (cont.)

- *Mathematica* notebooks can be used to show students
  - Computations
  - To do algebra that is too tedious for them to do
  - To allow them to explore the solution
  - To incorporate other media
- Questions remain as to if our approach gives a significant or incremental benefit to the students.