Learning Differential-Equation Aspects of Fluid Mechanics with Spreadsheet-Facilitated Computational Fluid Dynamics

Jean-Pierre Delplanque and Robert J. Kee
Division of Engineering
Colorado School of Mines, Golden, CO 80401

1 Introduction

It is well known that continuum fluid flow is described at its most fundamental level by the Navier-Stokes equations, a system of nonlinear, second-order, partial differential equations. In spite of this solid foundation, the equations themselves are sufficiently difficult to understand and solve that they are often introduced only superficially in fluid-mechanics courses. Because it is difficult to connect mathematical theory to engineering practice, a differential-equation perspective is often perceived as "too mathematical," or "too disconnected" to be of any value. It is our view that the differential-equation view does have value in facilitating the fundamental understanding of fluid flow, but it needs to be made accessible to the students with a reasonable level of effort. The intent of this paper is to describe how we have incorporated computational fluid dynamics (CFD) into fluid-mechanics courses, especially at the advanced levels.

In graduate-level fluid mechanics, the students must work with theoretically and mathematically complex vector and vector-tensor concepts. While such concepts and operators are understandable in a purely mathematical sense, it is usually a challenge to relate the mathematical abstractions to a practical flow field. We use CFD simulation as a de-facto laboratory that permits the students to explore the flow field.

In a class that is primarily oriented to the theoretical aspects of fluid mechanics, it is impractical to dedicate enough time for the students to create their own CFD simulations and to write the post-processors required to explore theoretical aspects of the flow. Instead, we computationally solve certain physically interesting problems beforehand and write the flow field (i.e., velocities, temperatures, pressures, etc.) into a spreadsheet (e.g., Excel) that is distributed electronically to the students. The CFD computation is usually done on a logically rectangular mesh network, which simplifies the student effort in the subsequent formation of the derivatives required to represent certain operations (e.g., forming the vorticity field). Based on the computed flow fields as represented in the spreadsheets, we assign homework exercises to reinforce theoretical concepts that are being developed in the lectures. It is relatively easy for the students to form vector or tensor operators by manipulating the solution fields in the spreadsheets. Also, Excel graphing features facilitate visualization of results. Overall, the students can understand theoretical complexities of a flow field, with a modest time investment.

The paradigm proposed here combines the advantages of laboratory teaching (e.g., hands-on experience) with an effective teaching of scientific methods and problem solving [1]. Of course, a primary benefit of this method is that students play an active role in tackling abstract concepts, which have not been traditionally conducive to such participation. Furthermore, these exercises result in an improved competency of the students in using spreadsheets for engineering purposes, thus preparing them better for their future professional endeavors. [1] The novelty of the proposed technique resides in its objective to illustrate abstract concepts. This is a departure from prior efforts to use spreadsheets, CFD, or software tools such as MATHCAD in the engineering curriculum, since such efforts have typically focused on reducing the amount of repetitive calculations that are associated with certain engineering tasks such as pump selection [2], replacing traditional coding of numerical solutions by the use of "engineering arithmetic tools" [3, 4], or familiarizing the students with the new tools of their future trade (e.g., CFD) [5].

1.1 General conservation equations

Inasmuch as the overarching objective of this work is to improve the student's understanding of the fluid conservation equations in a very general setting, it is appropriate to state the basic conservation equations. In general vector form the system that we consider is

Overall mass continuity:

$$\frac{\partial \rho}{\partial t} + \boldsymbol{\nabla}\boldsymbol{\cdot}(\rho \mathbf{V}) = 0, \tag{1}$$

Momentum (Navier-Stokes):

$$\rho \frac{D\mathbf{V}}{Dt} = -\nabla p + \mathbf{f} - \boldsymbol{\nabla}\times\left[\mu(\boldsymbol{\nabla}\times\mathbf{V})\right] + \nabla\left[(\lambda + 2\mu)\boldsymbol{\nabla}\boldsymbol{\cdot}\mathbf{V}\right]. \tag{2}$$

Thermal energy:

$$\rho c_p \frac{DT}{Dt} = \frac{Dp}{Dt} + \boldsymbol{\nabla}\boldsymbol{\cdot}(k\nabla T). \tag{3}$$

In these equations, the variables take their usual meanings: t time, ρ the mass density, $\mathbf{V}$ the velocity vector, p the pressure, $\mathbf{f}$ a volumetric body force, μ the dynamic viscosity, λ the bulk viscosity, c_p the specific heat at constant pressure, T the temperature, and k the thermal conductivity. The vector operators are: the divergence of a vector field $\boldsymbol{\nabla}\boldsymbol{\cdot}$, and ∇ the gradient of a scalar field. The substantial derivative operator of a vector field is given generally as

$$\frac{D\mathbf{V}}{Dt} = \left[\frac{\partial \mathbf{V}}{\partial t} + (\mathbf{V}\boldsymbol{\cdot}\boldsymbol{\nabla})\mathbf{V}\right] = \left[\frac{\partial \mathbf{V}}{\partial t} + \nabla\left(\frac{\mathbf{V}\boldsymbol{\cdot}\mathbf{V}}{2}\right) + \mathbf{V}\times(\boldsymbol{\nabla}\times\mathbf{V})\right], \tag{4}$$

and the substantial derivative of a scalar (e.g., temperature) is stated generally as,

$$\frac{DT}{Dt} \equiv \frac{\partial T}{\partial t} + \mathbf{V}\cdot\nabla T = \frac{\partial T}{\partial t} + (\mathbf{V}\cdot\boldsymbol{\nabla})\,T. \tag{5}$$

From the system of equations themselves, it is apparent that many of the terms have less than intuitive meaning to most students. Moreover, throughout the course of the derivations, there are a great many manipulations that are understandable in an abstract mathematical setting, yet have less-than-clear physical meaning. Finally, for certain flow fields, the equations can often be reduced or simplified via physical reasoning, similarity transformation, or scaling arguments. One of our objectives is to facilitate the understanding of these processes in the context of exploring real flow fields.

2 Setup: The instructor's preparation

2.1 Choosing an adequate fluid flow problem

The success of the approach described here depends primarily on the specific fluid flow problem chosen. The premise is that a single flow problem will be used to support various assignments so that the additional time needed for the students to familiarize themselves with the flow geometry and spreadsheet structure decreases significantly after the first assignment. We believe that a "good" choice must be based on the following criteria:

- The fluid flow problem must be relevant to a practical engineering device or system.
- The configuration considered must be "rich" enough to allow the development of several, varied exercises.

Relevance to a practical engineering device or system is critical to the main objective of these exercises: highlighting the usefulness of the differential-equation approach to "real world" fluid flow problems. There are obvious limits to the complexity of the case to be considered however. First, the geometry of the flow domain must be simple enough to allow the use of a structured mesh (this issue is further addressed in the next section) and to yield a spreadsheet data structure that does not unduly obscure the manipulations that will be performed by the students. A case in point is that of the problem dimensionality. Spreadsheets are two-dimensional and while it might be possible to extend the method described in this paper to three-dimensional cases, the necessary data structure would certainly offset the added learning benefits targeted by such an extension. Therefore, according to this criterion, the flow inside an intake manifold would not constitute an appropriate configuration.

The second criterion is a consequence of the premise that a single flow problem will provide the data for various exercises. This implies that the flow field must be complex enough to allow a variety of meaningful analyses. For example a fully developed pipe flow would only present limited interest for the type of application considered here. On the other hand, the various flow features and processes that may exist in the configuration studied should not be so tightly coupled as to hinder the pedagogical value of the exercise.

The flow field chosen as an example in this paper (see Fig. 1) satisfies both criteria. It represents a stagnation-flow chemical-vapor-deposition reactor - a flow configuration that is used widely in thin-film growth. It is rich in fluid-mechanical content that can be explored and understood in the context of theoretical concepts in the Navier-Stokes equations. For example, the downward flow issuing from the porous inlet manifold very nearly approximates a potential flow until it merges into a viscous stagnation-flow boundary layer just above the stagnation plane, which is where deposition occurs in the CVD reactor. The purge-gas flow that enters through the annular region from above undergoes a wall boundary-layer development. There is vorticity generation on the walls, which is especially great as the process gas merges with the purge gas and turns downward at the edge of the stagnation surface. The geometry is axisymmetric and therefore two-dimensional. In this flow the gas density and viscosity are temperature dependent.

It should be noted that the paradigm presented here could also be used very efficiently to illustrate "qualitative" concepts. A good example is that of the scaling arguments made in boundary layer analysis. The derivation of the boundary-layer equations from the Navier-Stokes equations is based on a scaling analysis that shows that axial-momentum diffusion can be neglected as compared to transverse-momentum diffusion. This analysis could easily be reinforced by having the students evaluate and compare the momentum-diffusion terms in a dataset obtained by solving the full Navier-Stokes equations for a flow over a flat plate. Such a dataset could also be used to illustrate concepts such as the displacement thickness. In this way, geometrically simple configurations can also be very beneficial.

2.2 From Fluid Flow Problem to Spreadsheet

Once the fluid-flow problem has been chosen, the resulting flow field must be obtained. This is done using a Computational Fluid Dynamics (CFD) software. Since the resulting datasets will be treated like a "numerical experiment," it is important that the software chosen be reasonably advanced and efficient. The code used in the examples presented here is CFD-ACE+™ version 5 (CFD Research Corporation, Huntsville, AL). The details and intricacies of the numerical solution of fluid-flow problems are not within the scope of this paper. However, a few points having a direct bearing on the development of the present paradigm must be emphasized.

The physical problem considered here is described in the previous section and illustrated in Fig. 1. The computational domain is imposed by the problem's geometry. The first choice to be made relates to the type of mesh to be used. Today's CFD codes offer a wide variety of mesh types allowing the simulation of fluid flow in complex domains. However, one must keep in mind that the fields produced by the simulation will be imported into a spreadsheet and then manipulated by the students. Therefore, in order to keep the organization of the spreadsheet amenable to this type of exercise, unstructured meshes should be avoided. Fields obtained with unstructured meshes would require additional post-processing and interpolation before being imported in the spreadsheet or they would lead to spreadsheet with

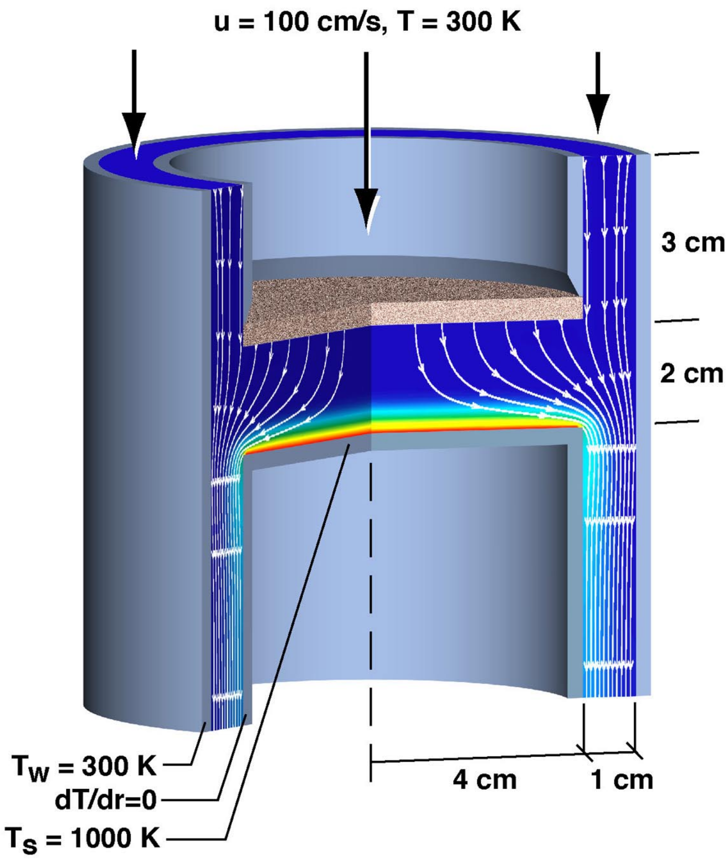


FIGURE 1: Computational Fluid Dynamics simulation of a flow field that is representative of a stagnation-flow chemical vapor deposition reactor. The colors (gray shades) represent the temperature field and the streamlines are shown in white.

a complex data structure including connectivity tables and in which the shape of the arrays would not resemble the flow problem geometry. The mesh used in the present example is structured and uniform (square, see Fig. 2) and comprises 1614 cells. Mesh uniformity is not necessary for the resulting dataset to be usable as described below. Indeed, the procedure outlined in the next section is unchanged for a non-uniform mesh. Nonetheless, it is advisable to keep the total number of cells as small as the mesh sensitivity will allow so that the resulting arrays are not prohibitively large and therefore too cumbersome to manipulate efficiently.

Most CFD codes let the user define the computational mesh using “blocks” or sub-domains to allow the description of non-rectangular domains in a structured manner while keeping grid stretching and skewness to a minimum. The mesh shown in Fig. 2 is comprised of 4 blocks as shown. Blocks may require an additional step in the importing process if the resulting fields are output by block.

Finally, the numerical simulation yields fields that must be imported into a spreadsheet. If the software used allows some flexibility in terms of the format in which the fields are written, i.e. if the source is accessible or (for commercial codes) if user-provided subroutines

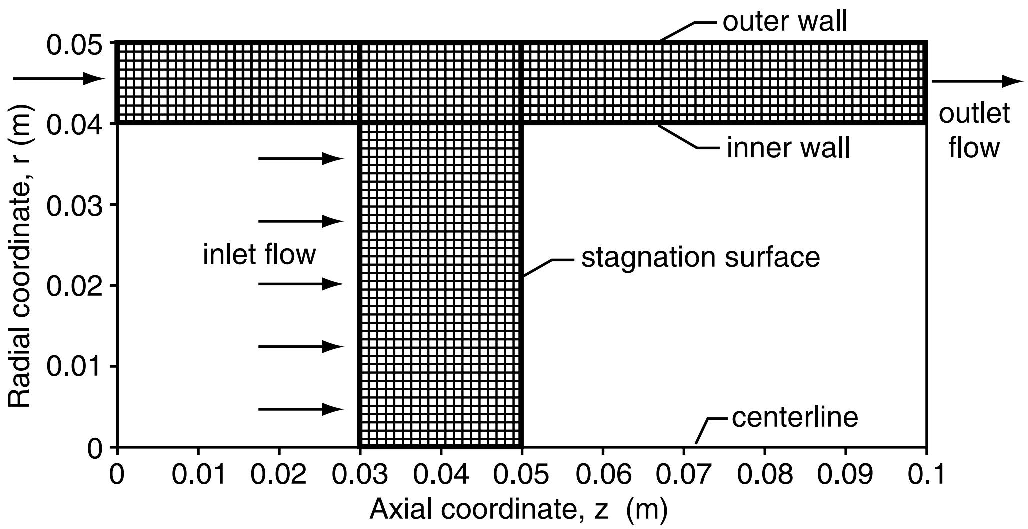


FIGURE 2: Block-structured mesh used for the CFD simulation of the stagnation-flow case illustrated in Fig. 1. Note that the mesh orientation is rotated 90° counterclockwise relative to the normal orientation of the reactor. The mesh is oriented to facilitate the user interface for axisymmetric problems in CFD-ACE+™.

are a possibility, the importing process can be significantly streamlined. Alternatively, an appropriate filter can be written using a generic scripting language (e.g., PERL). In the present case, the fields were re-formatted manually by cutting and pasting the result files using a word-processor and the files thus obtained were imported into the spreadsheet using EXCEL's import feature.

3 Exercises: The student's perspective

As initially presented to the class, the spreadsheet contains the flow field represented as individual "sheets" for the velocity components, the temperature, and the pressure. Each sheet has exactly the same spatial layout, which mimics the computational domain. Also, the top row of each sheet has the radial coordinate and the left column contains the axial coordinate. These coordinates are necessary for the subsequent formation of spatial derivatives.

The following section describes in considerable detail the procedure that a student must follow to create the sheets that are needed to complete certain exercises. For those already familiar with EXCEL and its syntax, there is much too much detail provided here. However, for those who are not so familiar with EXCEL, we anticipate that the detail will be helpful.

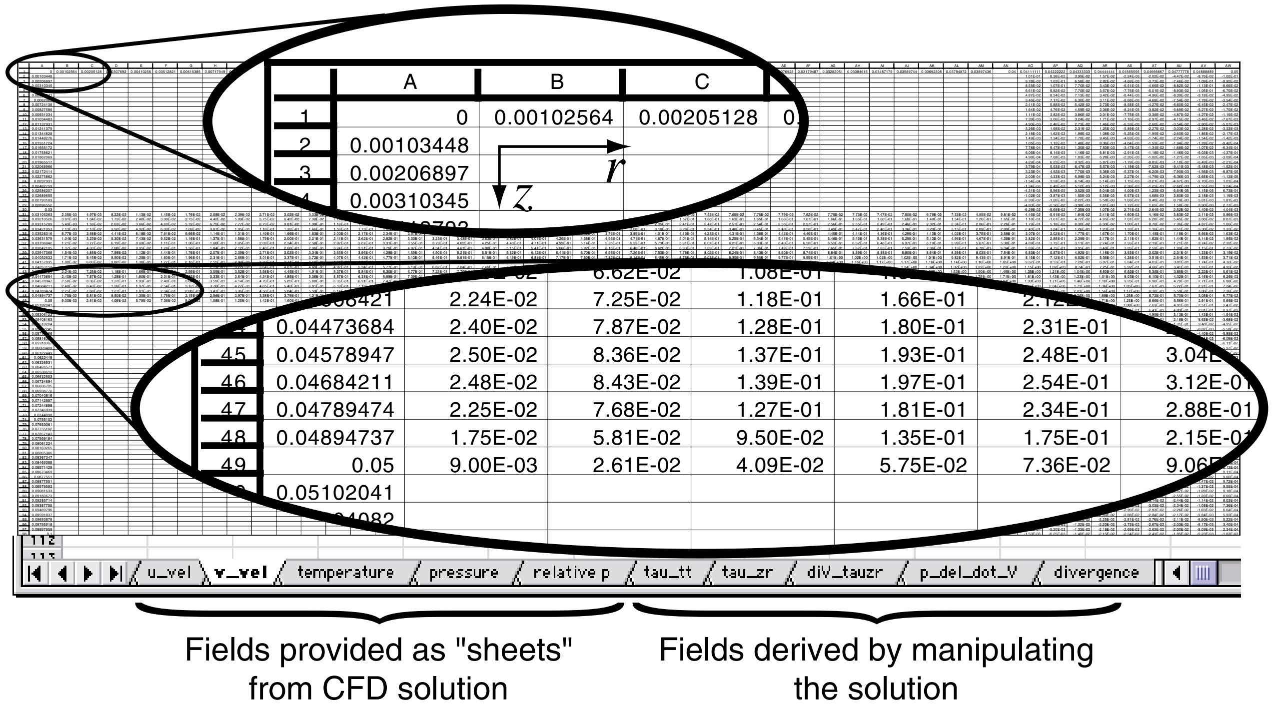


FIGURE 3: Spreadsheet layout to represent the flow field for the stagnation-flow reactor illustrated in Fig. 1. The radial-velocity sheet is shown here, as indicated by the selected "v_vel" sheet tab. Even though the numbers are much too small to read in this figure, the general shape of the flow field is apparent. The centerline is on the left, with the flow issuing downward from the inlet manifold toward the stagnation surface. The region on the right-hand side represents the annular purge-flow channel.

3.1 Developing a new sheet

Nearly all the exercises require the formation of a vector operator involving spatial partial derivatives, which typically requires information from more than one sheet. Consider, for example, the divergence of the velocity field. In axisymmetric coordinates,

$$\nabla \cdot \mathbf{V} = \frac{\partial u}{\partial z} + \frac{\partial v}{\partial r} + \frac{v}{r}. \tag{6}$$

To begin exploring the behavior of the divergence, the student first must create a new sheet that will contain this field. A sheet can be created in Excel with the `INSERT_WORKSHEET` command. As initially created, the sheet has the generic name like "Sheet 4." By double clicking on the tab that contains the sheet name, the name can be changed the a more representative name like "divergence." As created, of course, the sheet is empty. The next step is to fill the sheet with numbers that establish the pattern for the flow field. For example, click the tab for the "v_vel" sheet, which takes the user to the sheet containing the radial velocity as initially provided. Once in this sheet, one can drag the cursor to highlight all the numbers in the sheet and copy them onto the clipboard (`EDIT_COPY` command). Alternatively, while

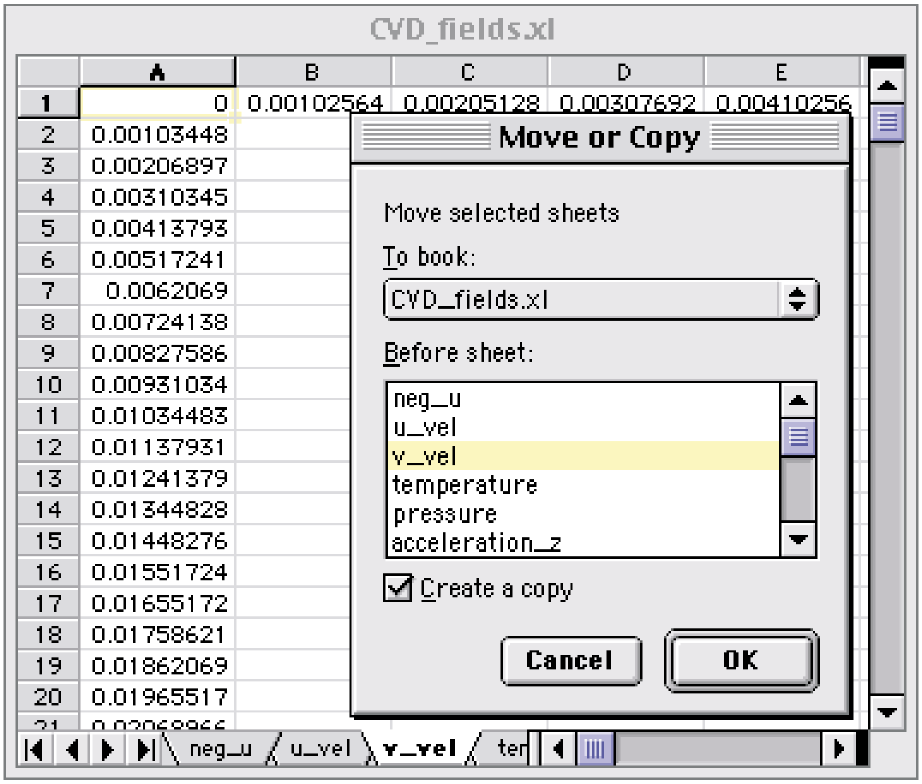


FIGURE 4: Illustration of the `Move or Copy Sheet` process.

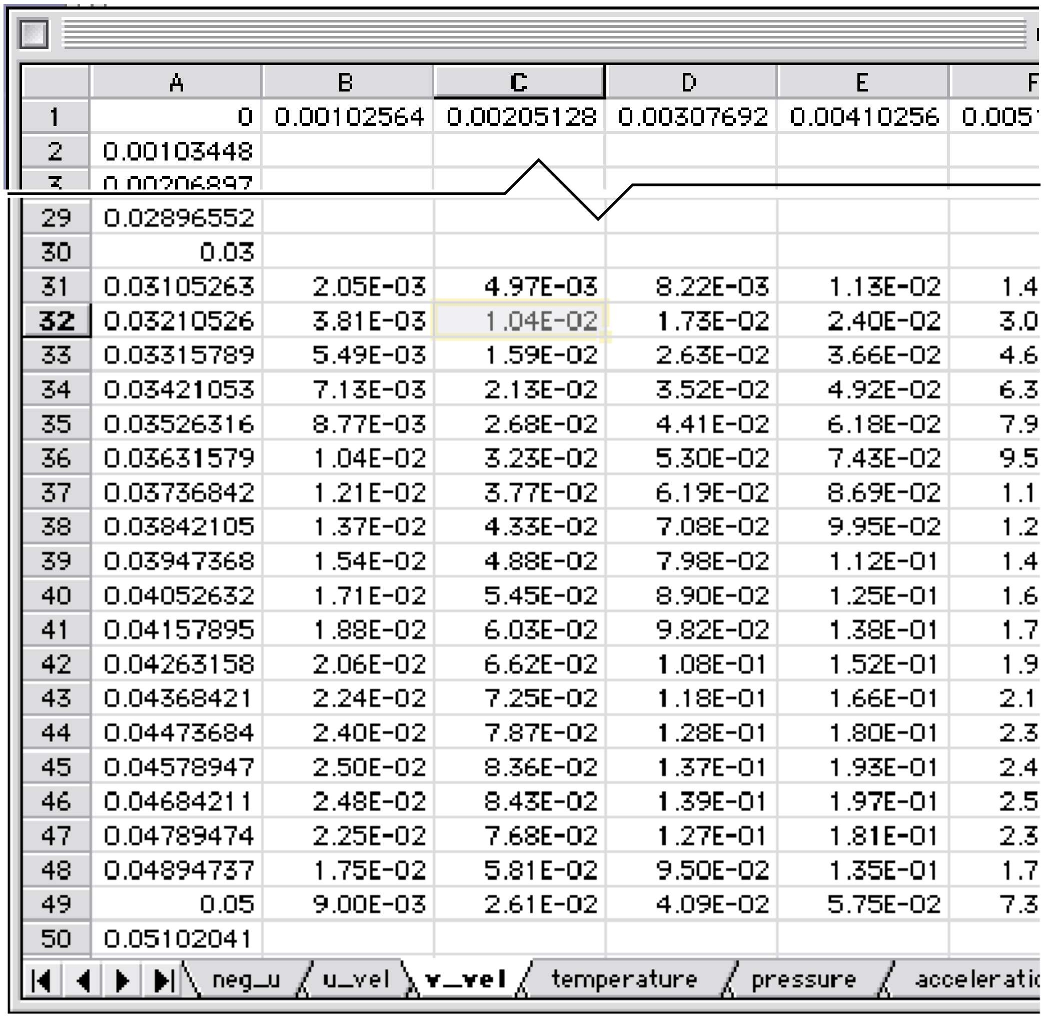
in the "v_vel" sheet, one can use the `EDIT_Move or Copy Sheet` command to copy the entire sheet. This command opens the window illustrated in Fig. 4. By highlighting a certain sheet, e.g., "v_vel," and checking the "Create a copy" box, a new sheet will be created with the contents of the "v_vel" sheet. Next, change the copied sheet's name from "v_vel(2)" to "divergence."

The students task in the "divergence" sheet is to enter a finite-difference representation of the divergence operator, i.e., Eq. 6,

$$\frac{\partial u}{\partial z} + \frac{\partial v}{\partial r} + \frac{v}{r} \approx \left(\frac{u_{i+1,j} - u_{i-1,j}}{z_{i+1} - z_{i-1}} \right) + \left(\frac{v_{i,j+1} - v_{i,j-1}}{r_{j+1} - r_{j-1}} \right) + \frac{v_{i,j}}{r_j}, \tag{7}$$

where i and j represent the z and r cell indices. Such a formula is easily and intuitively translated to neighboring-cell references in the spreadsheet, which is geometrically arranged to mimic the flow domain.

To begin forming the finite-difference divergence field (Eq. 7), proceed to the highlight first interior cell of the flow field, i.e., cell C32 as illustrated in Fig. 5. In cell C32, enter the



	A	B	C	D	E	F
1	0	0.00102564	0.00205128	0.00307692	0.00410256	0.005
2	0.00103448					
3	0.00206897					
29	0.02896552					
30	0.03					
31	0.03105263	2.05E-03	4.97E-03	8.22E-03	1.13E-02	1.4
32	0.03210526	3.81E-03	1.04E-02	1.73E-02	2.40E-02	3.0
33	0.03315789	5.49E-03	1.59E-02	2.63E-02	3.66E-02	4.6
34	0.03421053	7.13E-03	2.13E-02	3.52E-02	4.92E-02	6.3
35	0.03526316	8.77E-03	2.68E-02	4.41E-02	6.18E-02	7.9
36	0.03631579	1.04E-02	3.23E-02	5.30E-02	7.43E-02	9.5
37	0.03736842	1.21E-02	3.77E-02	6.19E-02	8.69E-02	1.1
38	0.03842105	1.37E-02	4.33E-02	7.08E-02	9.95E-02	1.2
39	0.03947368	1.54E-02	4.88E-02	7.98E-02	1.12E-01	1.4
40	0.04052632	1.71E-02	5.45E-02	8.90E-02	1.25E-01	1.6
41	0.04157895	1.88E-02	6.03E-02	9.82E-02	1.38E-01	1.7
42	0.04263158	2.06E-02	6.62E-02	1.08E-01	1.52E-01	1.9
43	0.04368421	2.24E-02	7.25E-02	1.18E-01	1.66E-01	2.1
44	0.04473684	2.40E-02	7.87E-02	1.28E-01	1.80E-01	2.3
45	0.04578947	2.50E-02	8.36E-02	1.37E-01	1.93E-01	2.4
46	0.04684211	2.48E-02	8.43E-02	1.39E-01	1.97E-01	2.5
47	0.04789474	2.25E-02	7.68E-02	1.27E-01	1.81E-01	2.3
48	0.04894737	1.75E-02	5.81E-02	9.50E-02	1.35E-01	1.7
49	0.05	9.00E-03	2.61E-02	4.09E-02	5.75E-02	7.3
50	0.05102041					

FIGURE 5: Illustration of the "divergence" sheet as the process of creating the divergence field begins. In the state illustrated, the numbers still represent the radial velocity, "v_vel."

following central-difference, finite-difference, formula to represent the divergence operator, i.e., Eq. 7,

```
=(u_vel!C33 - u_vel!C31) / (u_vel!$A33 - u_vel!$A31)
   + (v_vel!D32 - v_vel!B32) / (v_vel!D$1 - v_vel!B$1)
    + (v_vel!C32 / v_vel!C$1)                                  (8)
```

The entries in the formula refer to either the "u_vel" or the "v_vel" sheets. The syntax `u_vel!C33` refers to cell C33 in sheet "u_vel." The delimiter "!" separates the name of the sheet and the cell reference. The $ before the row or column reference is required in the denominators where the spatial coordinates are used. The $ indicates an absolute reference to the first row and column, which will not be altered in subsequent relative copying of the formulas.

The next step is to drag the divergence formula into all the other interior cells. With cell C32 highlighted, place the cursor on the lower-right-hand corner of the cell, making the "+" cursor appear. With this cursor, drag the formula in cell C32 down the column to cell C48. Now highlight the full column from C32 to C48 and drag it with the lower-right-hand "+" cursor across the sheet. Continue this process until all the interior cells of the sheet are filed with the divergence formula.

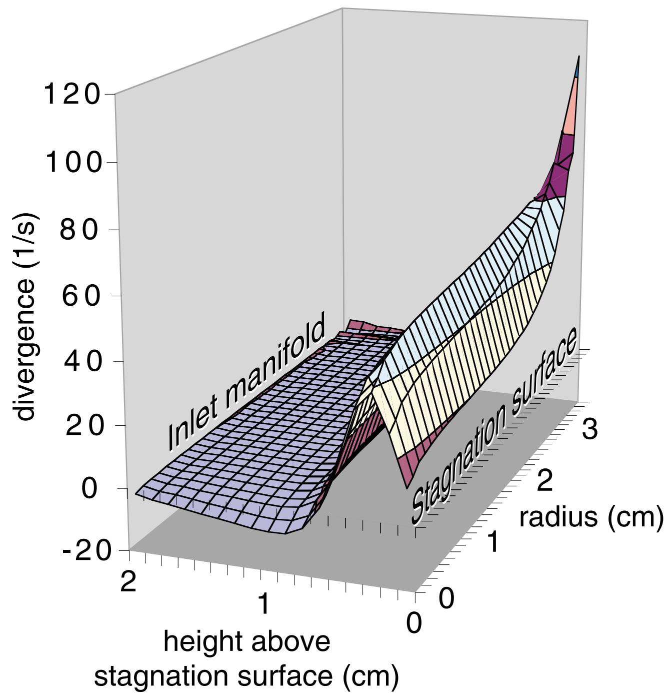


FIGURE 6: Plot of the divergence field in the region between the inlet showerhead manifold and the wafer susceptor.

The first interior cells around the boundaries still contain the velocity from the initially copied sheet. On the boundary cells, the central-difference formula (Eq. 8) is not appropriate since some of the derivatives would reach outside the boundaries. Instead, one must use one-sided differences. Consider, for example, row 31 which defines the inlet flow through the stagnation-flow manifold. The divergence operator for cell C31 can be approximated as

```
=(u_vel!C32 - u_vel!C31) / (u_vel!$A32 - u_vel!$A31)
   + (v_vel!D31 - v_vel!B31) / (v_vel!D$1 - v_vel!B$1)
    + (v_vel!C31 / v_vel!C$1) .
```
(9)

This formula is then dragged across row 31 until it meets the annular channel flow. An analogous procedure must be implemented for all the boundaries.

The entire interior of the flow is represented with a single formula that is dragged throughout the domain, while the boundaries require special attention. In many cases, the principal learning objectives can be accomplished without considering the boundary cells, thus considerably reducing the programming effort. For the present divergence example, nothing particularly important happens in the boundary cells, and exploring the interior alone is sufficient.

Figure 6 illustrates the divergence field in the region between the inlet manifold and the stagnation surface. Because this particular flow is a low-speed (low Mach number and Reynolds number) flow, it has many aspects of incompressible flow in the isothermal regions. Thus, as seen in Fig. 6, the flow is nearly divergence free ($\boldsymbol{\nabla}\cdot\mathbf{V} \approx 0$) in the region between the inlet manifold and the thermal boundary layer above the stagnation surface. Because the gas density ρ depends inversely on the temperature, it is apparent from the mass-continuity equation (Eq. 1) that there must be considerable velocity divergence in the thermal boundary layer near the heated surface.

3.2 Fluid acceleration

The Navier-Stokes equations are essentially a statement of Newton's second law, i.e., $\mathbf{F} = m\mathbf{a}$. Unlike solid-body dynamics, however, the notion of acceleration is a bit more complex for a fluid system. In deriving the Navier-Stokes equations one must introduce the concepts of a system and a control volume, relating the fluid acceleration to the substantial derivative through the Reynolds Transport Theorem. In a general Eulerian framework, the acceleration is given in terms of the substantial derivative as

$$\mathbf{a}(t, \mathbf{x}) = \frac{D\mathbf{V}}{Dt} \equiv \frac{\partial \mathbf{V}}{\partial t} + (\mathbf{V} \cdot \boldsymbol{\nabla})\,\mathbf{V} \tag{10}$$

While the operation $(\mathbf{V} \cdot \boldsymbol{\nabla})\,\mathbf{V}$ is relatively straightforward in cartesian coordinates, it is a bit trickier in other coordinate systems. In general the operation is defined by a vector identity as

$$(\mathbf{V}\cdot\boldsymbol{\nabla})\,\mathbf{V} \equiv \frac{1}{2}\nabla\,(\mathbf{V}\cdot\mathbf{V}) - [\mathbf{V}\times(\boldsymbol{\nabla}\times\mathbf{V})] \tag{11}$$

In cylindrical coordinates, which are required for the illustration here,

$$\begin{aligned}(\mathbf{V}\cdot\boldsymbol{\nabla})\,\mathbf{V} \quad = \quad & \mathbf{e}_z \left(u\frac{\partial u}{\partial z} + v\frac{\partial u}{\partial r} + \frac{w}{r}\frac{\partial u}{\partial \theta} \right) + \\ & \mathbf{e}_r \left(u\frac{\partial v}{\partial z} + v\frac{\partial v}{\partial r} + \frac{w}{r}\frac{\partial v}{\partial \theta} - \frac{w^2}{r} \right) + \\ & \mathbf{e}_\theta \left(u\frac{\partial w}{\partial z} + v\frac{\partial w}{\partial r} + \frac{w}{r}\frac{\partial w}{\partial \theta} + \frac{vw}{r} \right),\end{aligned} \tag{12}$$

where u, v, and w are the velocity components in the z, r, and θ directions.

Even though these equations are fundamental to all of fluid mechanics, most students find the general vector statements nonintuitive and confusing. It is therefore helpful to explore their behavior in a real flow field. In the spreadsheet, two new sheets are created

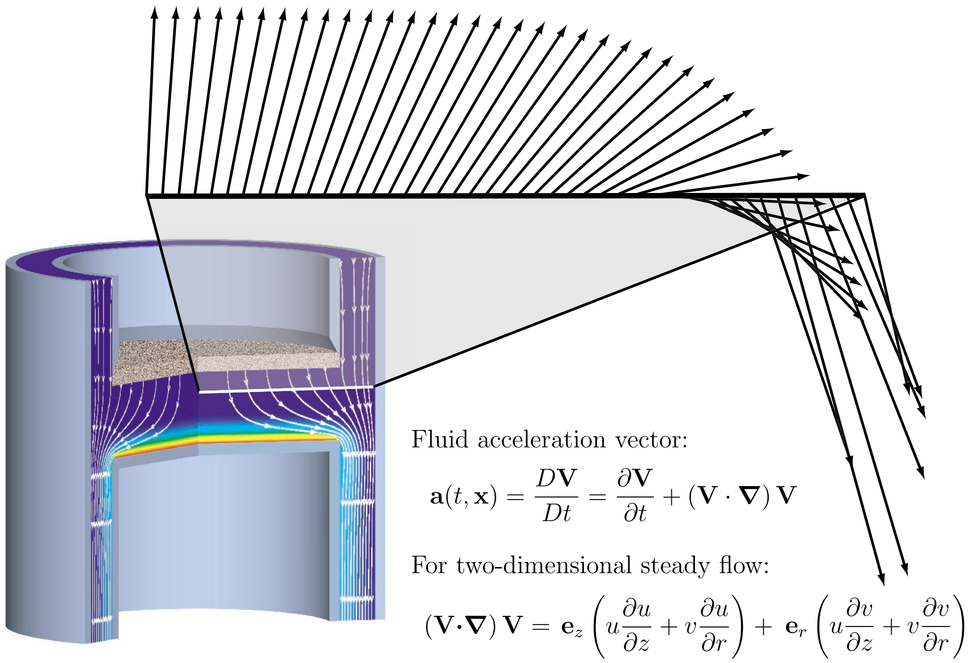


FIGURE 7: Fluid acceleration vectors along a line between the inlet manifold and the stagnation surface.

for the axial and radial components of the acceleration vector. The general expressions are easily programmed, following the procedure described in the previous section for the velocity divergence.

Figure 7 illustrates the acceleration vectors along a particular line in the flow field. The abstract vector notation notwithstanding, this plot produces an intuitive and easily reconciled result. As the flow leaves the inlet manifold, it must decelerate as it approaches the solid stagnation surface, i.e., the acceleration vectors are generally opposite the flow direction. The flow must also turn towards the annular outlet channel, i.e., acceleration vectors pointing radially outward to increase the radial velocity. At the outer edge of the stagnation surface, the flow turns sharply downward, flowing through the annular channel towards the exhaust.

Even though the vector components of the acceleration are easily determined, there is not a spreadsheet-based graphical interface for displaying them as vectors. Figure 7 was prepared by bringing acceleration components into a separate spreadsheet and computing the coordinates of the arrowheads. A fair bit of hand work in a graphics program was used in creating this figure.

3.3 Vorticity field

For an axisymmetric flow, the vorticity vector $\boldsymbol{\omega} = \boldsymbol{\nabla} \times \mathbf{V}$ has only one component, i.e.,

$$\boldsymbol{\omega} = \mathbf{e}_\theta \left(\frac{\partial v}{\partial z} - \frac{\partial u}{\partial r} \right) \tag{13}$$

which is easily formed in the spreadsheet setting.

In a graduate fluid-mechanics course we concentrate on many aspects of viscous boundary-layer behavior. One important topic is the similarity behavior of semi-infinite stagnation flows [6] and the stagnation flow in a finite gap [7, 8]. An important similarity attribute of these flows is that outside the viscous boundary layer near the stagnation surface the fluid behaves as an inviscid, rotational, flow. In this outer region, the vorticity scales as

$$\frac{\omega_\theta}{r} = \text{constant}, \tag{14}$$

which is an essential aspect of the similarity behavior. It is often important in the design and operation of stagnation-flow chemical-vapor-deposition reactors to seek the similarity behavior, which leads to highly uniform thin-film deposition.

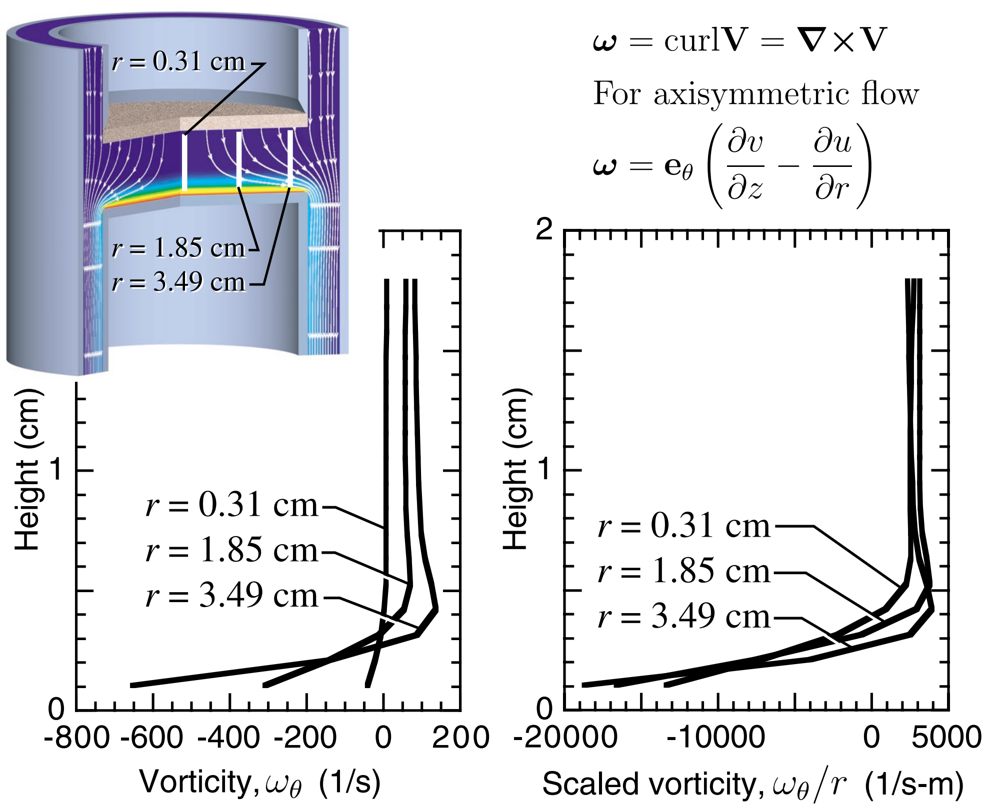


FIGURE 8: Illustration of the vorticity field along three vertical lines between the inlet manifold and the stagnation surface.

Based on the spreadsheet formation of the general vorticity field, Fig. 8 illustrates the vorticity in the region between the inlet manifold and the stagnation surface. Clearly, there is a large source of vorticity at the stagnation surface, and the magnitude of the vorticity increases with increasing radial position. The vorticity has a nearly constant value on each vertical line above the viscous boundary layer, indicating inviscid, rotational flow. The right-hand panel plots the scaled vorticity ω_θ/r, which would be nearly a constant in the outer regions if the flow field had achieved the classic stagnation-flow similarity. In this case, it is clear that the flow tends toward a similar behavior, but the scaling is not perfect.

3.4 Advanced examples

There are a great many possibilities to explore a flow field, seeking to explain or clarify a mathematical or physical concept. The details and emphasis, or course, depend on the nature and direction of the class. By way of illustration, we simply mention a few of the other exercises that we find beneficial.

An essential step in deriving the Navier-Stokes equations involves the use of Stokes postulates to relate the strain-rate tensor and the stress tensor. This is most often accomplished in the principal coordinates, and then rotated into a general coordinate system. The entire process is usually done in purely abstract mathematical terms.

At any particular point in a flow field, it is a straightforward matter to form the strain-rate and stress tensors from the velocity field. Written in matrix form, the low-dimension eigenvalue problem can be solved relatively easily using elementary matrix operations in EXCEL [9]. Thus, the principal stresses and strain rates can be found as well as the principal directions. Any number of potentially interesting issues can be explored. For example, the principal directions must be the same for both tensors. The diagonal invariant of the strain-rate tensor must be equal to the divergence of the velocity. In each of the principal directions, the stress and strain rate must be related through Stokes postulates as

$$\tau = -p + 2\mu\epsilon + \lambda \nabla \cdot \mathbf{V}. \tag{15}$$

In deriving the thermal-energy equation, it is usually a good idea to cast the substantial derivative in terms of enthalpy rather than internal energy. In doing so, however, the substantial derivative of pressure is introduced into the energy equation. Thus, it is not readily apparent that a transformation to the enthalpy form is a good idea. It is certainly a subtle point in the derivation (some might even claim esoteric), but one trades a $p\nabla\cdot\mathbf{V}$ term in the internal-energy form for a Dp/Dt term in the enthalpy form of the thermal-energy equation. By forming and comparing these two operators in a real flow, the student can be convinced that the Dp/Dt term is often negligible compared to the $p\nabla\cdot\mathbf{V}$ term. Thus, it is quite often the case that an explicit pressure dependence can be eliminated from the energy equation.

Clearly, the two foregoing examples are well beyond any theory that is introduced at the undergraduate level. However, at the graduate level, the ability to work with and explore subtle issues in the conservation equations can be a great assistance in strengthening the student's understanding at a very fundamental level. Again, by presenting real flow fields in a spreadsheet form, the student effort to work with these issues firsthand is reduced to a manageable level.

4 Limitations

The ease of exploring flow fields within the spreadsheet is offset by some practical limitations on accuracy. The original CFD computation is done on a mesh with a certain discretization basis, usually either in a finite-volume or finite-element setting. The simple finite-difference representations that are easy to implement in the spreadsheet are not fully consistent with the underlying computed flow field.

Computational Fluid Dynamics quite often uses a "staggered" mesh, in which the velocity is represented on cell faces and the temperature and other scalar fields are represented at cell centers. This is indeed the case for the simulation illustrated in this paper. Such an offset is a great nuisance in the spreadsheet, where there is an obvious benefit to having all variables represented at the same cell position on each sheet. After all, the whole idea of the spreadsheet representation is to reduce the student's overhead in exploring a flow. Either an interpolation or a simple half-cell shift of the velocity field introduces a source of inaccuracy in the evaluation of the various operators from which the student is expected to draw conclusions.

As a consequence of the foregoing considerations, one must recognize that the spreadsheet results are somewhat qualitative or semi-quantitative. Nevertheless, they are usually quite sufficient to meet the pedagogical objectives.

References

[1] W. J. McKeachie. *Teaching Tips: Stategies, Research, and Theory for College and University Teachers*. Houghton Mifflin, Boston, MA, 1999.

[2] C. W. Somerton. A spreadsheet program for the calculation of piping systems and the selection of pumps. In *ASEE Annual Conference Proceedings*, volume 1633, Washington, DC, 2000. American Society for Engineering Education.

[3] B. K. Hodge. The use of mathcad in viscous-flow courses. In *ASEE Annual Conference Proceedings*, volume 2666, Washington, DC, 1997. American Society for Engineering Education.

[4] A. I. Shalaby and S. E. Zanganeh. Teaching fluid mechanics using mathcad. In *ASEE Annual Conference Proceedings*, volume 1492, Washington, DC, 2000. American Society for Engineering Education.

[5] H. K. Navaz, B. S. Henderson, and R. G. Mukkilmarudhur. Bringing research and new technology into the undergraduate curriculum: A course in computational fluid dynamics. In *ASEE Annual Conference Proceedings*, volume 1602, Washington, DC, 1998. American Society for Engineering Education.

[6] F. M. White. *Viscous Fluid Flow*. McGraw-Hill, New York, 1991.

[7] R.J. Kee, J.A. Miller, G.H. Evans, and G. Dixon-Lewis. A computational model of the structure and extinction of strained, opposed-flow, premixed, methane-air flames. *Proc. Combust. Inst.*, 22:1479–1493, 1988.

[8] M.E. Coltrin, R.J. Kee, and G. H. Evans. A mathematical model of the fluid mechanics and gas-phase chemistry in a rotating disk chemical vapor deposition reactor. *J. Electrochem. Soc.*, 136(3):819–829, 1989.

[9] S. C. Bloch. *EXCEL for Engineers and Scientists*. John Wiley, New York, 2000.

JEAN-PIERRE DELPLANQUE
Dr. Jean-Pierre Delplanque is an Assistant Professor in the Engineering Division at CSM. His research interests focus on theoretical and computational fluid dynamics and transport phenomena in inert and reactive multiphase flow with applications in combustion and materials processing. He has a Engineering Diploma from ENSEEIHT (Toulouse, France), an M.S. ("DEA") in Mechanics from the the National Polytechnic Institute of Toulouse (France), and an M.S. and Ph.D. in Mechanical and Aerospace Enginering from UC Irvine.

ROBERT J. KEE
Since 1996, Dr. Kee has been a member of the Engineering Division at the Colorado School of Mines, where he holds the George R. Brown chair. Prior to joining CSM, he was at Sandia National Laboratories. Dr. Kee's sponsored research efforts are in the modeling and simulation of thermal and chemically reacting flow processes, with applications to combustion and materials manufacturing.