CONVERGE
CFD SOFTWARE

NUMERICAL VISCOSITY

Convergent Science White Paper

Rev. August 06,2018

OVERVIEW

When using computational fluid dynamics (CFD) to solve the Navier-Stokes equations, the act of discretizing the equations necessarily replaces terms in the equations with approximations. These approximations add error to the solution. Some of this error can be represented as an additional diffusive term, which is referred to in the literature as "numerical diffusion," "diffusive error," or "numerical viscosity." Whatever your preferred terminology, this is a nonphysical artifact of CFD that can pollute your results. CONVERGE has several features to minimize the effects of numerical viscosity and improve solution quality.

In order to understand the adverse influence of numerical viscosity, consider the following thought experiment. Suppose we have a laminar flow of two parallel flow streams, one hot and one cold. If we set the fluid diffusion coefficients to zero, the analytical solution should not show any diffusion of mass, momentum, or energy. In other words, the temperature profile would remain discontinuous, as shown in Figure 1(b). Any smearing of the temperature profile in a numerical solution, as in Figure 1(a), can be attributed to nonphysical numerical diffusion.

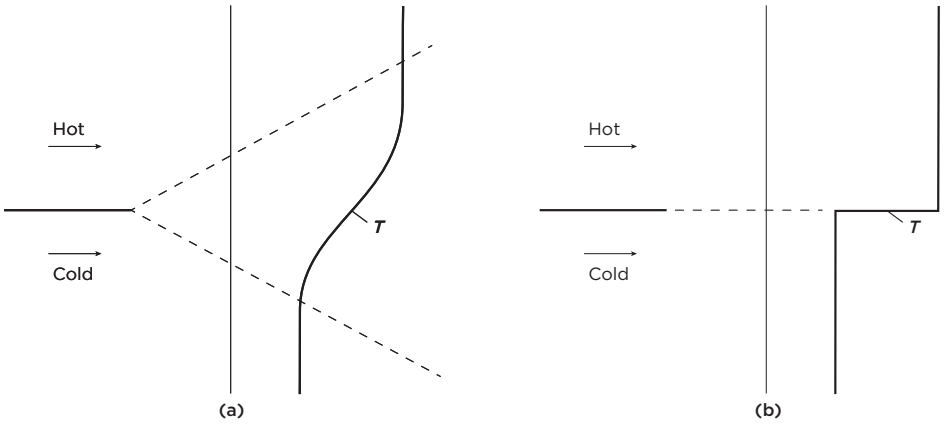

Figure 1: *A bifurcated unidirectional stream of two fluids with different temperatures, with physical diffusion coefficients set to zero. The idealized solution is shown on the right.*

We will discuss three pathways that introduce diffusive errors to a CFD solution and how CONVERGE minimizes such errors.

NUMERICAL SCHEME

When solving the Navier-Stokes equations using a finite volume technique, the convective term is discretized and the spatial derivative is approximated. As the Navier-Stokes equations are elliptic in nature for subsonic flows, a central difference scheme is appropriate.

A second-order central scheme has a leading dissipative error term on the order of ΔX^4, rather than the ΔX^2 of a first-order upwinded scheme.

Because a first-order upwind scheme is more stable than a central scheme, CFD codes switch to a first-order scheme in some regions to maintain global stability. The criteria for the local switch from second-order to first-order influence the tradeoff between global accuracy and global stability. A highly conservative switch may guarantee convergence by switching to first-order fluxes in challenging regions, but it may converge to an inaccurate solution due to the introduction of excessive numerical viscosity.

Many CFD codes prioritize stability, and so they readily switch to a first-order scheme. This switching is often hidden from the user. The majority of a flow domain may be solved using first-order numerics, even if second-order was selected by the user. There is rarely an easy way for the user to ascertain how many cells have been switched to a first-order scheme. To evaluate switching behavior, a user can run both first- and second-order simulations and compare the results. If the results are similar, then it is likely that the CFD code has switched to first-order fluxes in a large portion of the domain.

CONVERGE maximizes the usage of second-order numerics by using strict switching criteria, which minimize the numerical diffusion. Specifically, CONVERGE incorporates Total-Variation Diminishing (TVD) switches to minimize the use of first-order spatial fluxes. A TVD switch dynamically evaluates local fluxes as first-order upwind in regions where the existing flux scheme would diverge from the existing flow state. CONVERGE automatically detects and recovers from diverging residuals, so the TVD switches can maximize the use of second-order fluxes throughout the domain.

INFLUENCE OF MESH ALIGNMENT

Typically, a CFD solution is represented by piecewise constant values at every cell center (or every node). A spatial gradient in any physical quantity is best represented if the gradient vector is exactly aligned with the grid. Any misalignment of the flow with the grid results in a stair-stepping pattern that generates numerical viscosity and pollutes the solution. For a uniform Cartesian grid, it can be shown using Eq. 1 that numerical diffusion is maximized when the flow is at 45 *degrees* to the grid cells:

$$\Gamma_{\text{false}} = \frac{\varrho U \Delta x \Delta y \sin(2\theta)}{4\left(\Delta y \sin^3\theta + \Delta x \cos^3\theta\right)}$$

Equation 1: *False diffusion equation. Courtesy Patankar.*

We can consider our idealized mixing flow with a 45-*degree* grid as a worst-case test for numerical diffusion. Figure 2 presents a solution, which was run in CONVERGE. The flow is from lower left to upper right, with a 2000 *K* stream above a 1000 *K* stream. In this case, fluid diffusion coefficients are again set to zero, so the temperature profile should remain discontinuous. As can be seen for this coarse mesh solution below, the temperature profile broadens as it moves across the domain. This is due to numerical viscosity generated by the misalignment of the flow and the mesh.

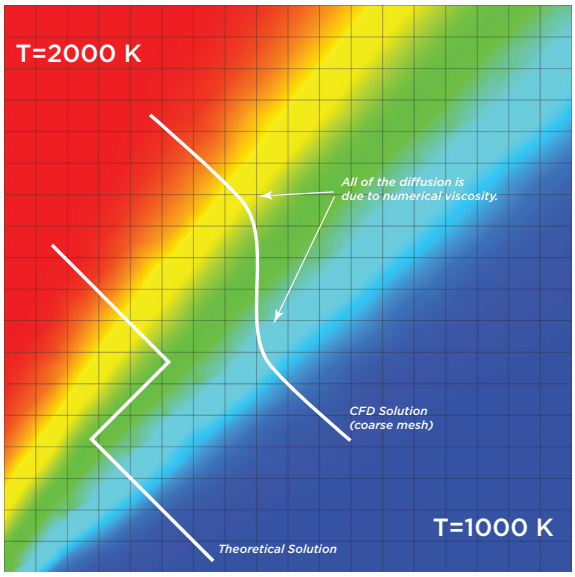

Figure 2: *Numerical diffusion test case with a coarse grid.*

Usually, we cannot predict the exact direction of physical gradients in a real engineering flow before we perform a calculation. Worse, complicated flows often feature physical gradients that are not even aligned with each other. A gradient-aligned grid is not possible in such a case. Fortunately, Equation 1 tells us that the numerical diffusion generated by grid misalignment is also a function of cell size. Accordingly, if we can use a fine grid in regions of strong gradients, we can minimize the effects of numerical viscosity due to misalignment.

CONVERGE incorporates Adaptive Mesh Refinement (AMR), which automatically refines cells based on curvature in field variables such as temperature and velocity. If we activate temperature-based AMR for our 45-*degree* test case, we see that AMR automatically refines the grid in the vicinity of the large temperature gradient.

In Figure 3, we see that as CONVERGE refines the cells in the mixing region, the numerical viscosity decreases. As a result, the smeared nonphysical temperature distribution seen in Figure 2 collapses to the correct physical temperature profile. By adding mesh refinement when and where it is most needed, AMR allows CONVERGE to minimize the

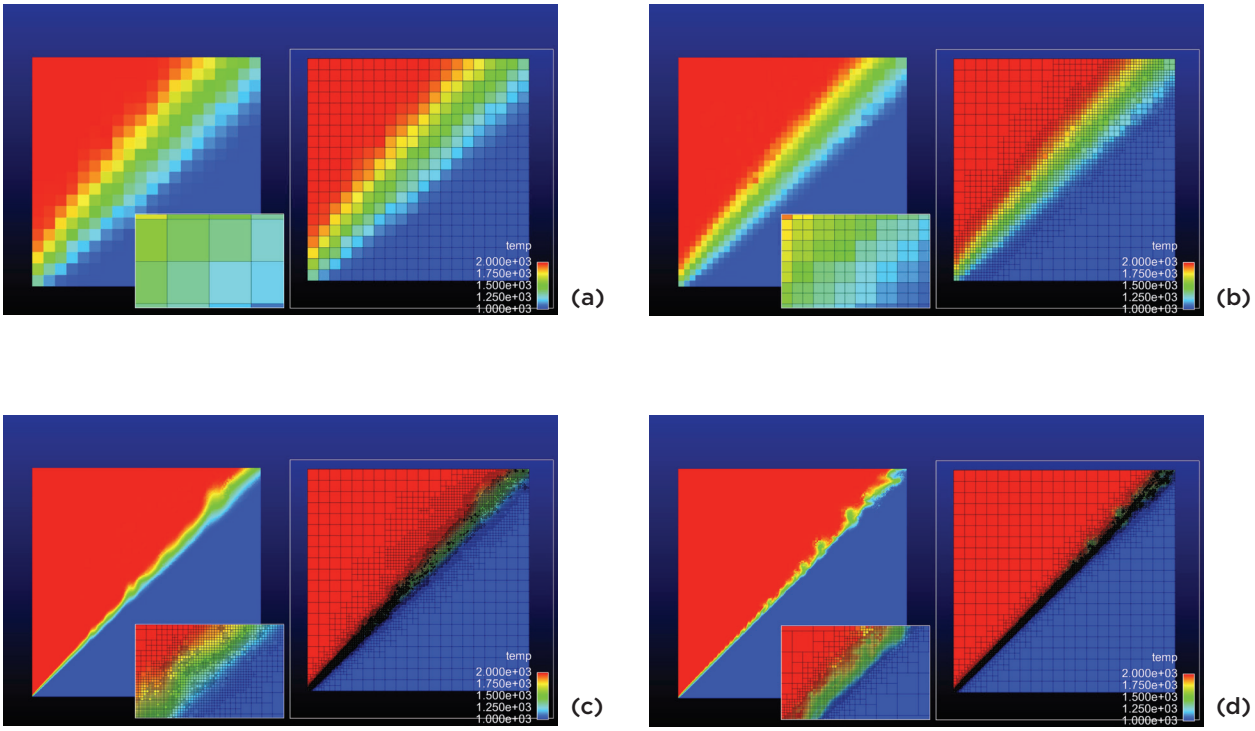

Figure 3: *Numerical diffusion test with adaptive mesh refinement enabled. As AMR automatically adds grid cells in the high-gradient region, the non-physical broadening of the solution disappears.*

numerical viscosity without foreknowledge of the final result and without an infeasibly fine grid that is dense everywhere.

INFLUENCE OF MESH MOTION

Many engineering applications of interest contain moving components. Internal combustion engines, compressors, and gas turbines all exhibit a complex fluid flow strongly affected by boundary motion. To perform a CFD simulation for such cases, either a single initial grid must move with the boundary, or the grid must be re-generated after motion has taken place.

Motion of a pre-generated mesh may not be able to accommodate the necessary deformation without excess skewness and associated poor solution quality. Furthermore, any mesh motion in regions of flow gradients will generate numerical diffusion, even if the subsequent meshes remain of high quality (Koltakov and Fringer). Consider the discontinuous temperature profile on an aligned 1D mesh, as shown in Figure 4(a). It is an exact piecewise constant representation of the profile. If we move the mesh upward, that profile cannot be represented exactly. Energy either will be transported with the motion of the mesh as shown in Figure 4(b), or the spatial temperature profile will broaden as shown in Figure 4(c). This effect is independent of the flow solver or numerical method.

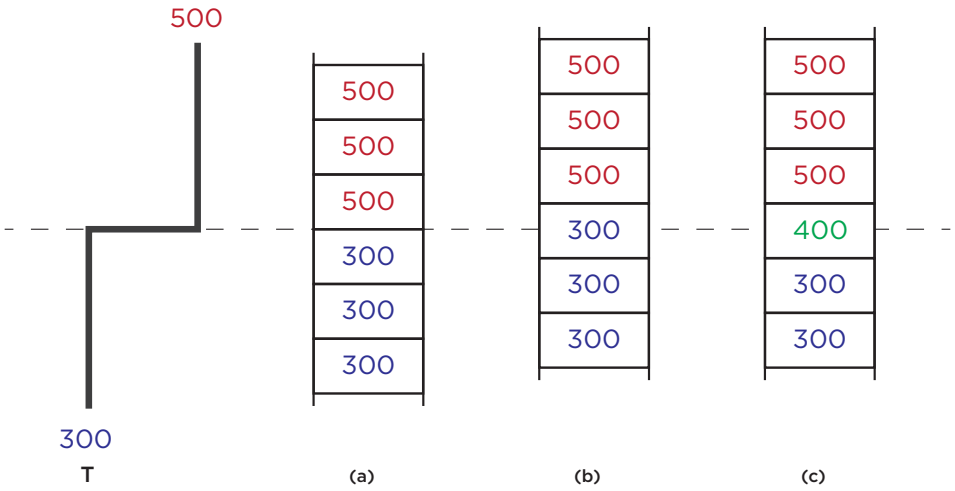

Figure 4: *Numerical diffusion due to mesh motion. At left, exact piece-wise representation. At center, mesh motion transports energy. At right, mesh motion broadens temperature distribution.*

CONVERGE does not move or deform the mesh. Rather, for cases with boundary motion, CONVERGE generates a high-quality, orthogonal, stationary mesh automatically at each time-step. Because the Navier-Stokes equations are always solved on a stationary mesh, numerical viscosity due to mesh motion is eliminated.

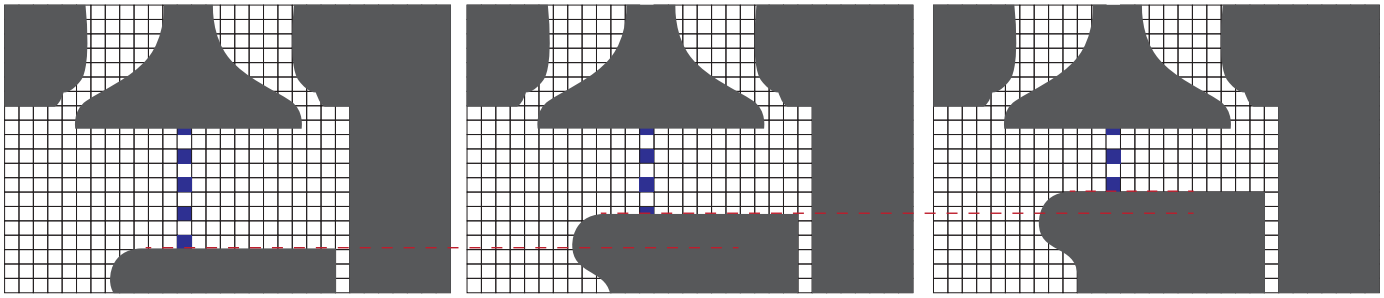

Figure 5: *Schematic illustrating **CONVERGE** using orthogonal and stationary mesh elements for a case with moving components. As the geometry moves through the volume, cells are resized, created, and removed.*

CONCLUSION

All CFD methods and approaches generate some nonzero level of diffusive error due to the contribution of numerical viscosity. In order to produce the most accurate simulation possible, we must minimize this diffusive error. CONVERGE limits the production of numerical diffusion by:

1. Using central differencing schemes wherever possible,
2. Using a fine mesh in regions of substantial gradients, and
3. Using a stationary mesh.

As a result, CONVERGE produces characteristically low levels of diffusive error and helps you generate accurate flow solutions.

Questions or comments? Please email **contact@convergecfd.com**.

REFERENCES

Patankar, S.V., Numerical Heat Transfer and Fluid Flow, CRC Press, 1980.

Koltakov, S. and Fringer, O.B., "Moving grid method for numerical simulation of stratified flows," Int. J. Numer. Meth. Fluids, 71(12), 1524–1545, 2012. DOI: 10.1002/fld.3724

Founded in Madison, Wisconsin, Convergent Science is a world leader in computational fluid dynamics (CFD) software. Its flagship product, CONVERGE, includes groundbreaking technology that eliminates the user-defined mesh, fully couples the automated mesh and the solver at runtime, and automatically refines the mesh when and where it is needed. CONVERGE is revolutionizing the CFD industry and shifting the paradigm toward predictive CFD.

CONVERGENT SCIENCE: WORLD HEADQUARTERS
6400 Enterprise Ln
Madison, WI 53719
Tel: +1 (608) 230-1500

CONVERGENT SCIENCE: TEXAS
1619 E. Common St., Suite 1204
New Braunfels, TX 78130
Tel: +1 (830) 625-5005

CONVERGENT SCIENCE: DETROIT
21500 Haggerty Rd.
Detroit, MI 48167
Tel: +1 (248) 465-6005

CONVERGENT SCIENCE: EUROPE
Hauptstraße 10
Linz, Austria 4040
Tel: +43 720 010 660

CONVERGENT SCIENCE: INDIA
Office #701,
Supreme Headquarters
Mumbai-Bangalore Highway
Baner, Pune, Maharashtra 411021
+91 741-0000-870

CONVERGENT SCIENCE

For additional information or to contact us, visit **convergecfd.com**