J. D. Freels[1] · **A. J. Baker**[2]

High Fidelity TEA Theory Altered Compressible Navier-Stokes Using COMSOL Equation-Based Modeling

Abstract It is common knowledge to readers of these COMSOL conference proceedings that COMSOL, as well as other established CFD codes, produces unstable numerical solutions of the Navier-Stokes (NS) equations without "consistent stabilization" and/or "inconsistent stabilization" enabled. Indeed, the ability to disable the stabilization method in COMSOL is unique among most commercial CFD codes; i.e., it is "hidden" as if stabilization is not even present!

With aim to replace this approach of added stabilization (perhaps a code option in the future) a novel new theory [1,2] is summarily presented in this paper denoted "Truncation Error Annihilation" (TEA). The TEA theory provides a new algorithm that mathematically annihilates the algebraic instability mechanism via continuum equation systems alteration with analytically derived Reynolds number dependent cubic nonlinear tensor product calculus functionals. The need for numerical stabilization obviated, tridiagonal stencil equivalent CFD algorithm discretization of theory modified NS/RaNS equations are directly coupled into the unchanged COMSOL "High-Mach Number Flow" (HMN) equation system following standard procedure for equation-based modeling. Consequently, TEA theory enables an algebraically stable generation of resolutely oscillation-free $O(h^4)$ Taylor-series accurate state variable distribution that is monotone to iteration convergence digit on any discretization.

TEA is applicable to the entire class of NS/RaNS physics, but the choice here is to focus on transonic and supersonic compressible flows. This choice of flow regime is primarily due to the challenge in solving at high-fidelity meshing level, and a requirement of smooth monotonicity, while retaining pure Galerkin interpolation test functionality, with inclusion of accurate shock present flows. Secondarily, both authors have sustained efforts on this issue after the time of primary-author Ph.D. dissertation [3].

The approach is simple for the COMSOL construct. Both consistent and inconsistent stabilization are disabled entirely by "mouse click" in the option box of the code graphical user interface (GUI). Simultaneously, the TEA terms are added through direct option within the COMSOL model tree in weak form as contribution additions for all the state variable equations, written in non-conservative form, on pressure, velocity, temperature, and turbulent eddy viscosity. These equations solve for the conservation of mass, momentum, energy, and turbulent eddy viscosity closure using Spalart-Allmaras turbulence model. TEA theory is presented in non-dimensional form, so the input for solving using COMSOL in both dimensional form on one problem, and non-dimensional form on another problem, are consistently applied to established benchmark cases and demonstrated to yield valid results.

Two classic validation model problems are presented from a subset of the suite of challenge problems in recent AIAA High-Fidelity Workshop Proceedings [4]. The Sajben-diffuser problem is also included in the COMSOL application library (CAL). The dimensional form inputs were improved based on NASA references to archival data for geometry and test data. A direct comparison with COMSOL consistent stabilization results is also included for the Sajben-diffuser with all other solution features being equal (mesh, inputs, solver, etc.). Results for a Smooth-Bump Transonic flow AIAA challenge problem, defined by non-dimensional input, are also presented. Included is a complete history of COMSOL adaptive mesh procedure, along with classical energy norm convergence data.

[1] (retired) Oak Ridge National Laboratory (ORNL)
Adjunct Faculty, The University of Tennessee, USA
cell: (865)919-0320
E-mail: jdfreels@pm.me

[2] Professor Emeritus
The University of Tennessee, USA
cell: (865)207-1537
E-mail: ajbaker@utk.edu

1 Introduction/Foreword

After retirement from ORNL on 1/1/2018, a primary goal for Jim was to investigate features of COMSOL that were out of scope during his full-time working career. Primarily, COMSOL equation-based modeling, but other areas such as the relatively new application builder, and perhaps even the physics builder were features seldom used. Focusing on the equation-based modeling first, an obvious route to take was to repeat the equations used in his PhD dissertation [3] (FORTRAN code); indeed, a goal since FEMLAB v3.0 at the onset.

Simultaneously, a primary goal for AJ after his retirement from UTK teaching and research, has been to compose a monograph summarizing his many years of research on finite-element based CFD, and also to lay the groundwork for future research by others. When AJ learned of Jim's post-retirement goals, he invited Jim to consider including valuable improvements in CFD algorithms after his graduation over a quarter century earlier. A renewed collaboration began.

The decision to collaborate turned out to be very rewarding. For Jim, not only would the goal of learning COMSOL equation-based modeling be achieved, but the knowledge and implementation of TEA theory has resulted in much-improved CFD solutions. For AJ, our collaboration has expanded the scope of the monograph to include significant improvement to the compressible NS contribution and utilization of COMSOL for TEA theory implementation.

Our goal for this paper is to share with the reader the significance of TEA theory toward high-fidelity, monotone, stable, CFD solutions of superb quality. TEA theory is applicable to all CFD problem types. In this paper, we demonstrate and validate for compressible flows. We hope that TEA theory might be directly included into COMSOL as an option or replacement for "consistent stabilization" freely available as "open-source" technology.

2 A Synopsis of TEA Theory Applied to Compressible Navier-Stokes Equations

A complete discussion of the TEA theory is not possible here due to conference constraints, but can be found in the cited references[1, 2]. We present a short summary herein. A generalized form of multi-dimensional NS equations is written as

$$f_j(q,x_i)\frac{\partial q}{\partial x_i} - \varepsilon(x_i)\frac{\partial^2 q}{\partial x_j^2} = 0 + O(h^2), \tag{1}$$

where f is the flux vector corresponding to the equation index $_j$ (mass, momentum, energy), $_i$ is the spatial index (x,y,z), q is the $_j$ equation state variable $\{\rho, m_j, E\}^T$ in conservative form or $\{p, u_j, T\}^T$ for the non-conservative form used in the COMSOL CFD module for high Mach-number compressible flows. Note also that the term $O(h^2)$ used in equation 1 denotes the 2^{nd}-order accurate discretization error, where h denotes the element measure (area for 2D), that is omnipresent in essentially all CFD codes presently used; including COMSOL.

A rigorous expansion of each of the nodal expressions (after finite-element assembly) are utilized to arrive at the $O(h^2)$ error terms, and are then evaluated. From these nodal expressions, an alteration is quantified to evaluate the truncation error via cubically nonlinear functionals defined in most legacy NS finite-difference and/or finite-element 2^{nd} order algorithms. Utilizing tridiagonal stencils [1,2], the truncation error is then *annihilated* as a result of this rigor, yielding an altered expression of the form

$$f_j(q,x_i)\frac{\partial q}{\partial x_i} - \frac{1}{Re}\frac{\partial^2 q}{\partial x_j^2} - \frac{Reh^2}{12} f_k(q,x_i)\frac{\partial}{\partial x_k}\left[f_j(q,x_i)\frac{\partial q}{\partial x_j}\right] = 0 + O(h^4), \tag{2}$$

wherein, a new index $_k$ is created on the equation-system array, the Reynolds-number notation is symbolic as the result of conversion to non-dimensional form for all the equations, and the truncation error is now $O(h^4)$,and therefore, the $O(h^2)$ error is *annihilated*! By symbolic, for example, the non-dimensional constant would be $RePr$ for the energy equation, or another quantity depending on the equation evaluated. Note that any equation containing the higher-order viscous terms, such as turbulence model equations, is eligible for the TEA alteration operation.

As a simplification, we only consider the diagonally dominate terms; i.e., $j = k = 1,2$ in 2D for this paper. From Equation 2, the first and second term are the original, unaltered equation terms, whereas the third term is the alteration and is all that we focus on in this analysis. Consideration of the effects of including the off-diagonal terms $(j \neq k)$ would be an excellent choice for further research. If we limit our analysis to $2D$, and $j = k$ terms only, the TEA terms become

$$\cdots + \frac{Reh^2}{12}\left\{ f_1(q,x_i)\frac{\partial}{\partial x_1}\left[f_1(q,x_i)\frac{\partial q}{\partial x_1}\right] + f_2(q,x_i)\frac{\partial}{\partial x_2}\left[f_2(q,x_i)\frac{\partial q}{\partial x_2}\right]\right\} = 0 + O(h^4). \tag{3}$$

Further simplification yields the essential terms needed for 2D TEA theory

$$\cdots - \frac{Re^*\left(\frac{h_e}{m}\right)^2}{12}\left[u^2\frac{\partial^2 q}{\partial x_1^2} + v^2\frac{\partial^2 q}{\partial x_2^2}\right] = 0 + O(h^4). \tag{4}$$

Specific to the COMSOL HMN compressible flow subset of the CFD module using Spalart-Allmaras turbulence model, the array of state variables is $q = \{p, u, v, T, \mu_T\}^T$, and the corresponding non-dimensional parameter is
$Re^* = \{\tau_p, \frac{Re}{\mu_{tot}}, \frac{Re}{\mu_{tot}}, \frac{RePr}{k_{tot}}, \frac{\sigma_\mu Re}{v_{tot}}\}^T$ respectively. Note the following: (1) the subscript tot denotes the "total" quantity for turbulent flow; i.e., $\mu_{tot} = \mu + \mu_T$, (2) all variables shown above are dimensionless form, (3) the parameter τ_p is discussed further in the next section, (4) the variable h_e denotes the element "measure" or in this case for 2D, the element area and is not to be confused with the built-in COMSOL quantity h which is a representative element length; i.e., the TEA theory requires the element length in 1D, area in 2D, and volume in 3D, and (5) the parameter m denotes the element order (1,2,3) for (linear, quadratic, cubic), respectively.

3 Application of TEA Theory into COMSOL Equation-Based Modeling

Near the end of the previous section, starting with Equation 4, we introduce the implementation of the TEA theory terms into the COMSOL HMN flow equations. This section provides all the details to complete the transition to using TEA theory.

3.1 To Be or Not to Be: Dimensionless or Not

Working in dimensionless form is ideal for theory and algorithm development. But, most COMSOL users prefer to use dimensional form. Further, COMSOL requires the user to be familiar with using dimensionless form in order to enter the correct information for the input stream. Regardless of the choice, TEA theory is very simple to manage both dimensionless and dimensional methods of working with COMSOL.

The TEA theory is written in dimensionless form with the parameters Re, Pr, etc., appearing directly in the equations shown. If the user decides to work in dimensionless form, he designates that option in the model tree, and then consistently processes all the variables, input and output, to be dimensionless. For example for the HMN (non-conservative form) equations, $p^* = p/p_o$, $\bar{u}^* = \bar{u}/u_o$, $T^* = T/T_o$. In addition, specific variables such as dynamic viscosity, and thermal conductivity (which naturally appear on the diffusion terms), must be input, in addition to dimensionless, as divided by the Re and or Pr where appropriate. For example, $\mu^* = \frac{\mu}{\mu_o Re}$, and $k^* = \frac{k}{k_o RePr}$ for the dynamic viscosity and thermal conductivity, respectively. If this is done consistently, then the TEA terms may be input as written.

Alternatively, most users of COMSOL will create their model in dimensional form. The input stream will be created as normal, and a very simple change is made to the TEA theory terms shown in Equation 4: (1) make sure all the variables are in dimensional form (which they should be), and (2) drop the Re and Pr from the equations (or alternatively set to unity).

To add the TEA terms in dimensional form to the COMSOL model tree, the following steps are taken: (1) uncheck all the stabilization (both consistent and inconsistent) options built in to the physics provided (override the default), and (2) add the following entries as separate "weak contributions" to the HMN physics settings:

conservation of momentum in the x-direction ($q_2 = u$):

```
-(h_2d/order)^2/12/mu_tot*
(test(ux)*u^2*ux+test(uy)*v^2*uy)
```

conservation of momentum in the y-direction ($q_3 = v$):

```
-(h_2d/order)^2/12/mu_tot*
(test(vx)*u^2*vx+test(vy)*v^2*vy)
```

conservation of energy ($q_4 = T$):

```
-(h_2d/order)^2/12/k_tot*
(test(Tx)*u^2*Tx+test(Ty)*v^2*Ty)
```

constitutive equation for the Spalart-Allmaras turbulence model ($q_5 = \mu_T$):

```
-(h_2d/order)^2/12*hmnf.sigmanu/nu_tot*
(test(nutildex)*u^2*nutildex
+test(nutildey)*v^2*nutildey)
```

Note the following: (1) `h_2d` denotes the h_e variable discussed above for the 2D element area (see COMSOL built-in functions, (2) `order` is a parameter specifying the element order m discussed above, (3) `_tot` denotes the total of fluid and turbulent summation, (4) the test function will link the weak contributions to the appropriate equations/terms in the HMN physics settings automatically.

Also note that the conservation of mass contribution, through the ($q_1 = p$) variable, has not been defined from the TEA theory input stream (yet). This important detail is discussed and accounted for in the next subsection.

3.2 A Special Case: Continuity Equation

Based on the theory, a fundamental requirement of the TEA method is that a viscous term must be present in order to be valid. This is because the theory is based on the alteration of the rigorous transformation from the base Equation 1 to the altered Equation 2. Without the viscous terms, the altered TEA terms would not be obtainable. The continuity equation is an excellent example of an equation that does not contain a viscous term. Another example is the momentum equation that assumes inviscid fluid; which in itself is theoretical and not physically possible. In order to obtain a solution of an inviscid problem using TEA theory, one must solve the problem using the viscous form of the equations and apply a slip-wall boundary condition, *and* utilize a Reynolds number as high as possible for a given mesh density. We demonstrate an inviscid problem solution later in this paper.

One method to get around the issue of not being able to apply the TEA theory to the continuity equation is to utilize a legacy $O(h^2)$ method on the continuity equation. We have done this for the HMN problems demonstrated herein, but found that it was not as powerful as what we saw in the TEA-enabled equations for momentum, energy, and turbulence model coupling. Further, we found that a legacy $O(h^2)$ solution compared to the improved TEA $O(h^4)$ solution was not as accurate, and did not perform as well in solving for a converged solution. The convergence rates for the $O(h^4)$ are significantly improved over the $O(h^2)$ solutions. Therefore, we needed to incorporate the TEA theory into the non-viscous termed continuity equation in an different manner.

Based on the pattern of the other equations already altered to solve with the TEA method, it was decided to create a "fake" viscous term in the continuity equation as written and applied to the COMSOL HMN equations that solve for pressure of the form:

$$-\frac{1}{\tau_p}\left[\frac{\partial^2 p}{\partial x^2}+\frac{\partial^2 p}{\partial y^2}\right]=p_\varepsilon, \tag{5}$$

where τ_p is a parameter added to obtain consistent units for the solved continuity equation, with units of time, and an arbitrary magnitude that can be used as a control mechanism. The value of p_ε is driven as small as possible toward zero to minimize the effect of the added term by making the value of τ_p as large as possible. The entire appearance of Equation 5 is that of a Laplacian equation (pure diffusion) with the inverse of an arbitrary parameter used as a multiplier of negative value.

In order to *annihilate* (remove) the $O(h^2)$ truncation error from the computed value of p_ε, an additional TEA term is also added to provide $O(h^4)$, in a similar manner to the other equations solved, to the continuity equation discrete formulation of the form [HMN 2D continuity on p]:

$$\cdots+\frac{\tau_p}{12}\left[u^2\frac{\partial^2 p}{\partial x_1^2}+v^2\frac{\partial^2 p}{\partial x_2^2}\right]=0+O(h^4). \tag{6}$$

From Equations 5 and 6, the inputs to the COMSOL model tree also in the weak formulation are:

conservation of mass fake viscous term ($q_1 = p$)

```
+h_2d/order/tau_p*(test(px)*px+test(py)*py)
```

conservation of mass fake TEA term ($q_1 = p$):

```
-(h_2d/order)^2*tau_p/L_p^2*
(test(px)*u^2*px+test(py)*v^2*py)
```

Note the following: (1) in the fake viscous term `(h_2d/order)` is raised to a power of 1 not 2, (2) the parameter `L_p` has units of meters and a value of unity and is only there to provide the correct set of units to be included into the continuity equation consistently, (3) the method of using this fake continuity viscous term is not yet incorporated into the TEA theory [1,2], and (4) we have found that the performance of this approach with the compressible flow continuity equation to be superior.

4 Validation

In a previous COMSOL conference[5], we validated the TEA theory (FaNS) on 1d compressible problems; including "viscous Burger's equation", and "Riemann Shock Tube". At that time, a legacy TWS $O(h^2)$ method was implemented to substitute for the

$O(h^4)$ TEA method for the continuity equation solving for density. An interesting exercise would be to go back to the 1D compressible NS equations to implement the "fake viscous" TEA $O(h^4)$ terms and see the improvement.

In this 2D validation of TEA theory, we present two classic problems that have been investigated by many researchers worldwide. Indeed, we have been active with an AIAA group concentrating on state-of-the-art "High-Fidelity CFD Verification" and both of the problems presented here were also included in the manuscript that was produced as a result of that collaboration[4]. The `COMSOL` model files are provided for the user with the conference proceedings.

4.1 Transonic Smooth Bump

The first problem presented is called the "transonic smooth bump" or "smooth bump". This simple problem describes a subsonic flow of air across a parallel plate surface, with a smooth bump over the bottom surface with a specified shape. The flow is assumed inviscid and enters on the left at Mach 0.7. The fluid conditions are described in dimensionless terms and are likewise setup dimensionless in `COMSOL`. The ambient temperature and pressure are set to 1.0 and $1/\gamma$ respectively. The input temperature and pressure set set to the total temperature and pressure respectively obtained from the following equations using ideal gas relationships:

$$\frac{T_o}{T_\infty} = 1 + \frac{(\gamma - 1)}{2} M_\infty^2 \tag{7}$$

$$\frac{p_o}{p_\infty} = \left[\frac{T_o}{T_\infty}\right]^{\left[\frac{\gamma}{(\gamma-1)}\right]} \tag{8}$$

The outlet pressure is also assumed fixed at ambient conditions. The top and bottom edges are set to no-slip walls. The geometry is specified from $(-1.5 \leq x \leq 1.5)$ and $(0.0 \leq y \leq 0.8)$. Three edges are constant $(x = -1.5, y = 0.8, x = 1.5)$ representing left, top, and right, respectively. The bottom edge is defined by the relationship:

$$y = 0.0625 e^{\{-25x^2\}}, (-1.5 \leq x \leq 1.5). \tag{9}$$

The geometry and boundary conditions are depicted in Figure 1. Note this problem is very similar to the "3D Euler Bump" problem given in the CAL. The flow produces a shock with

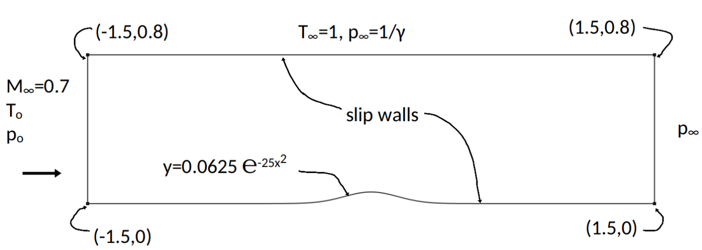


Fig. 1 Transonic smooth bump geometry and problem definition.

peak Mach number near the bump surface (NNE side, about 2 o'clock). Further, the theoretical total enthalpy should be constant throughout the domain; even across the shock. The object of the challenge is two fold: (1) predict and tabulate the precise shock location on the bump surface, and (2) produce a result as near to the theory as possible; in particular, across the shock.

This problem ran very well on `COMSOL` with TEA theory implemented. We produced results using linear, quadratic, and cubic finite-element basis functions as shown in the attached Figure 2. Note the overlay zoom along the bump wall edge is required to visualize any difference in the results! Perhaps the most impres-

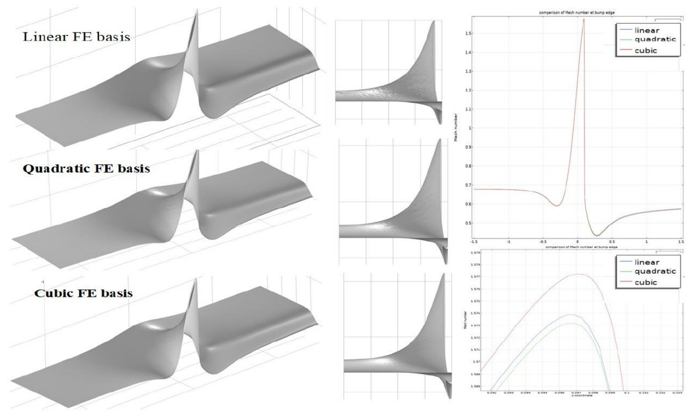


Fig. 2 TEA $O(h^4)$ transonic smooth bump Mach number surfaces comparing linear, quadratic, and cubic finite-element basis discretization.

sive finding was how well the adaptive mesh solver worked with this problem. As many as 17 adaptations from extremely coarse up to extremely fine mesh at the last adaptation where the mesh density was highest about the shock. Both the error in energy norm and error in total stagnation enthalpy L2 norm were computed to evaluate the solution quality. This is depicted in Figure 3 where the error in energy norm is colorized by averaging over each element using `COMSOL` "elemavg" built-in operator using the error in energy norm evaluation $0.5 \bigtriangledown^2 (\delta T)$. The integrated

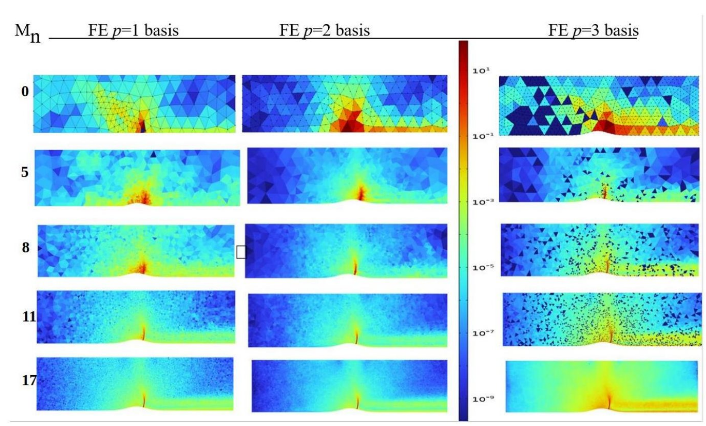


Fig. 3 TEA $O(h^4)$ transonic smooth-bump energy norm distribution as a function of adaptive mesh and basis function (note: colorized as average energy norm per element)

error in energy norm was also evaluated and plotted to compare against convergence theory as shown by Figure 4. Note that the slope of the linear and quadratic log-log curves are both equal to 4 due to the overriding $O(h^4)$ performance of the TEA terms in the applicable range. The final plot of smooth-bump results,

$$\left\| \{e^h(t)\} \right\|_{E,\mathrm{M}_{n+1}} = \frac{\left\| \{q^h(x_j,t)\} \right\|_{E,\mathrm{M}_{n+1}} - \left\| \{q^h(x_j,t)\} \right\|_{E,\mathrm{M}_n}}{(\mathrm{M}_{n+1}/\mathrm{M}_n)^{f(p)} - 1}, \quad f(p=1,2,3) = 4,4,6$$

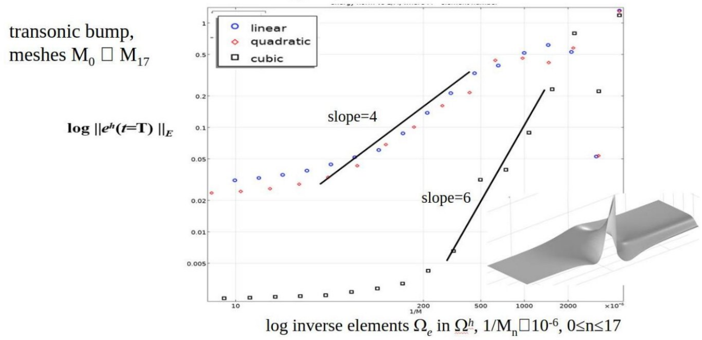


Fig. 4 TEA $O(h^4)$ transonic smooth-bump adaptive-mesh solution, energy-norm convergence rate study, variation in mesh density ($M_n = 0, 17$), and finite-element discretization ($p = 1, 2, 3$).

Figure 5, is a collage of interesting results that requires close examination. Part (a) shows the final adaptive mesh for the cubic basis. In addition to the high mesh density about the shock, the COMSOL adaptive algorithm also refined the inlet edge region, and a significant portion of the bump edge. The total number of cubic elements for this case was 134776. Part (b) shows the total enthalpy distribution. The theoretical value for the stagnation enthalpy is a constant, $H_o = 3.843$, and most of the distribution is very near this quantity. However, in the shock vicinity there is a significant dip down to $H_o \approx 3.68$. Part (c) shows a surface plot of the pressure distribution. Of significance is the additional plateau developed within the shock. Finally, Part (d) shows a gray-shade surface plot of the Mach number distribution. Of significance is the zero gradient in the normal direction for all the edges, including the bump edge. This results in the maximum Mach number being slightly inside the domain near the bump edge.

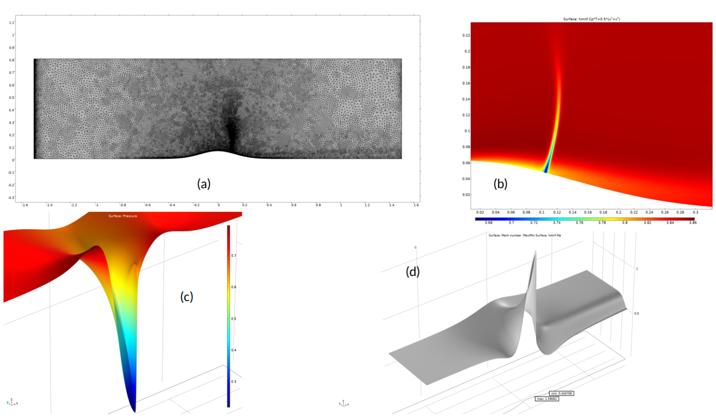


Fig. 5 TEA $O(h^4)$ transonic smooth-bump collage of interesting results (see discussion in the text).

4.2 Sajben Diffuser

The Sajben Diffuser problem is a widely analyzed 2D problem for compressible flow that is included in the CAL. Several references are provided in the application documentation that describe the problem in detail. In addition, long before COMSOL existed, the authors studied this problem, and results are included in Jim's PhD dissertation[3] completed on 5/1992. Most importantly, NASA has made available the geometry and test configurations along with several groups of test data and CFD results of the problem[6] after 1992.

The CAL was certainly a great place to start in building an updated model for this research. There were many changes made to improve on what was there initially. Some highlights of model changes are:

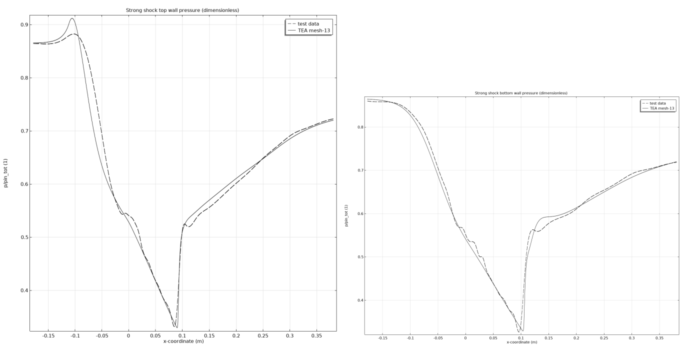
- The geometry definition was changed to equation-based as provided in the original papers. The original CAL model utilizes an interpolation of individual points to create the geometry. This change provides a much smoother transitions in the geometry regions and removes some unnecessary oscillations in the solutions.
- All the comparison data available in the NASA archive is utilized for comparison as opposed to a select few.
- The mesh design was completely changed to be more parameter-based, essentially providing an extensive manually-generated adaptive mesh solution.
- A short slip-wall entrance was added to provide a gradual change from free flow, slip wall, to no-slip wall design instead of a sudden no-slip wall change at the entrance.
- The COMSOL built-in consistent and inconsistent stabilization options were completely disabled, and the TEA theory physics-based terms are added as weak-form contributions. A special case was also performed in the end to directly compare the COMSOL consistent stabilization with TEA theory and is included in these results.
- The logic to automatically run the weak-shock case, and then restart/switch to the strong-shock case was removed. Instead, the present analysis investigated only the more-difficult strong-shock case by performing a sequence of 13 successive mesh refinement cases.
- And finally, we modified and added most of the results settings.

While the model was completely changed in many ways from the original provided by the CAL, there are still many subtle remnants of the original model still present.

The strong-shock settings were used as initial conditions. Because the model requires a boundary-layer mesh on the no-slip walls, the adaptive mesh solver could not be used effectively. It was tried several times, but just could not quite provide the required fine mesh near the wall that was needed for the Low-Reynolds Spalart-Allmaras turbulent model. Instead, a succession of parameter-based, manually-created mesh designs were executed, where the steady-state results from the coarser mesh was used as an initial condition for the next finer mesh to arrive at a final steady state. A total of 12 mesh refinements were generated in this manner using the parameter-based design mentioned earlier. The final mesh 13 yielded a total of 24,064,032 degrees of freedom (dof) to be solved.

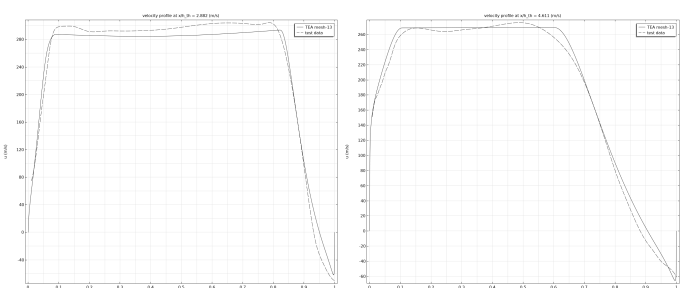
After the final mesh density steady-state was achieved using the TEA-altered equations, it was decided that it was important to perform a direct comparison with the COMSOL unaltered original equation with the TEA terms disabled, and the consistent stabilization re-enabled. In performing this direct comparison, it is very important to keep all the other variations in the code consistent with the TEA-computed results. This was performed by running one single case using the mesh-13 TEA-altered strong-shock steady state as an initial condition, and then restart with TEA turned off, and `COMSOL` consistent stabilization (CCS) turned on, and then run to a new steady-state with the same residual ($\varepsilon \leq 0.0002$) stop criteria.

The first set of plots presented are 1D plots of the TEA-enabled results for the final mesh-13 steady-state (SS). Figures 6, 7, and 8 compare the TEA-enabled result along 1D cut lines for wall pressures, and shock-downstream velocity profiles (1 and 2). While not perfect, the deficiency is clearly due to the turbulence model inherent shortcoming, and the `COMSOL` code is producing an accurate result of the equations being solved. The



(a) top (b) bottom

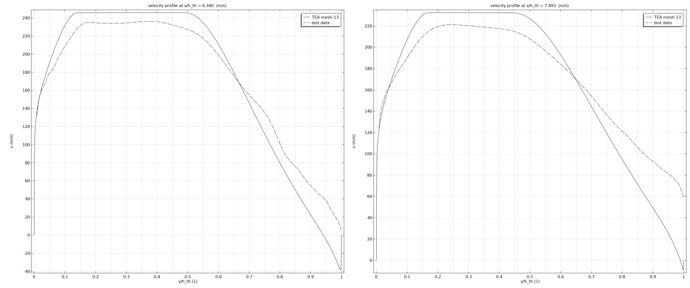
Fig. 6 TEA $O(h^4)$ Sajben diffuser comparison with test data - strong-shock wall pressure, top (left) and bottom (right).



(a) $\frac{x}{h_{th}} = 2.882$ (b) $\frac{x}{h_{th}} = 4.611$

Fig. 7 TEA $O(h^4)$ Sajben diffuser comparison with test data, strong-shock, u velocity comparison, shock downstream locations $\frac{x}{h_{th}} = 2.882$ and 4.611, (a) and (b) respectively, (1 of 2).

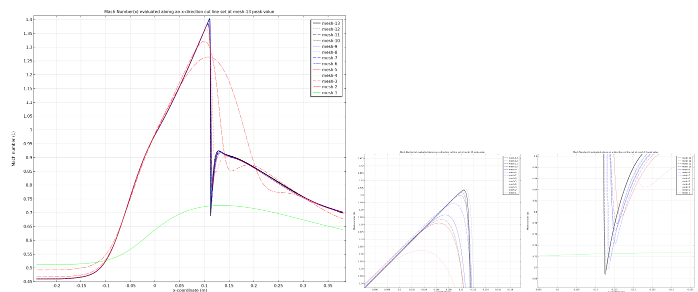
final 1D plot in Figure 9 presents a sequence of Mach number cut lines of each mesh density solution 1-13. The cut lines are taken at the point of maximum Mach number for mesh-13 along the x direction. Note the legend showing color and line design differences for each mesh density with highest-density mesh-13 shown as a thicker black line. Two additional plots show a zoomed view of the top and bottom regions capturing the convergence of the peak toward the accuracy in Mach number. These results clearly demonstrate the importance of significant mesh



(a) $\frac{x}{h_{th}} = 6.340$ (b) $\frac{x}{h_{th}} = 7.493$

Fig. 8 TEA $O(h^4)$ Sajben diffuser comparison with test data, strong-shock, u velocity comparison, shock downstream locations $\frac{x}{h_{th}} = 6.340$ and 7.493, (a) and (b) respectively, (2 of 2).

density in order to obtain the most accurate solution possible (high-fidelity).



(a) maximum Mach number cut line (b) top zoom (c) bottom zoom

Fig. 9 TEA $O(h^4)$ Sajben diffuser 1D plots of maximum Mach number cut line.

Figures 10, 11, and 12 show 2D plots of detail about the shock region. Figure 10 is a comparison plot (TEA and CCS) zoom surface plot, with contour line additions, of the transverse (v) velocity in the shock region and an additional zoom overlay of a critical region near the upper wall boundary layer interaction. The left side, a) TEA, demonstrating smooth. monotone, accurate solution, whereas the right side, b) CCS, demonstrates an oscillatory, unstable, and inaccurate behavior in the narrow shock region. Figure 11 is a comparison plot (TEA and CCS) zoom sequence of surface plots of the fluid density (ρ) in the shock region. Parts a) and b) are the TEA solution perspective and direct view respectively, while part c) is a similar CCS solution perspective. Note the TEA density peak is, again showing, smooth. monotone, and accurate solution; whereas, the CCS solution, again, shows an oscillatory, unstable, and inaccurate behavior at the shock peak (requires additional zoom with the pdf viewer to completely visualize). And finally, Figure 12 shows a zoom view of the flow separation regions at both the top curved wall, and bottom straight wall, where a shock-boundary layer interaction occurs and causes the flow to form a subsonic region, and separate between the wall and the dominate-flow region downstream of the shock. The plot is segmented into four parts (a-d) showing Mach number and transverse velocity (left and right) and wall regions curved and flat (top and bottom) respectively. The near-wall region produces the dominate eddy viscosity from the turbulent model, and hence, is important to

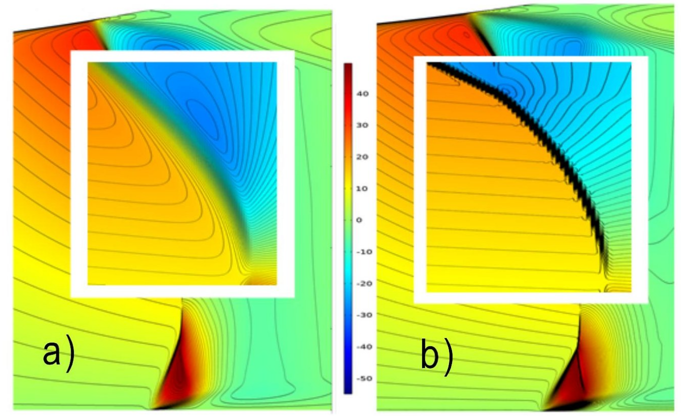


Fig. 10 TEA $O(h^4)$ Sajben diffuser comparison, transverse velocity (v), shock-centric zoom w/ super-zoom snippet imposed: a) TEA $O(h^4)$, b) CCS $O(h^2)$.

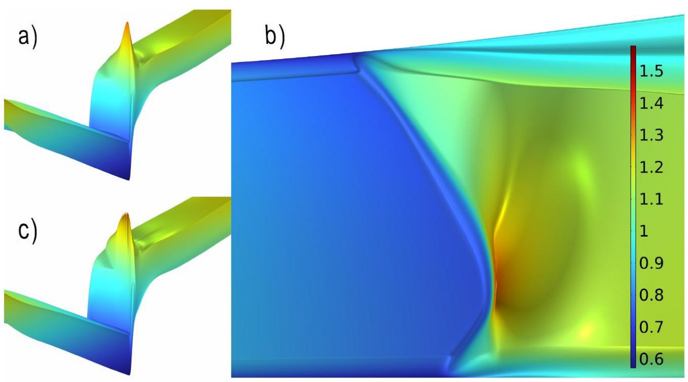


Fig. 11 TEA $O(h^4)$ Sajben diffuser compressible strong-shock flow density alternate shock-centric views a) TEA $O(h^4)$, b) TEA $O(h^4)$, c) CCS $O(h^2)$

obtain accuracy and high-fidelity in order to obtain accurate solutions.

5 Conclusions and Next Steps

We have presented a short summary of TEA theory which will *annihilate* $O(h^2)$ truncation error, while retaining much lower $O(h^4)$ truncation error for CFD solutions. The implementation of TEA theory has been presented using equation-based weak-form coding into the normal `COMSOL` input stream. Two high-fidelity solutions have been presented and compared to the traditional `COMSOL` consistent stabilization method.

In addition to publication of the complete TEA theory via the referenced monograph, our next step is to implement/demonstrate a consistent conservative form, along with Spalart-Allmaras turbulence model, of the compressible NS equations using `COMSOL` equation-based modeling. We believe this could also provide even higher accuracy for compressible flows along with increased Newton algorithm jacobian consistency for improved convergence.

We hope that `COMSOL` might directly incorporate TEA theory into their code for general availability to the user community,

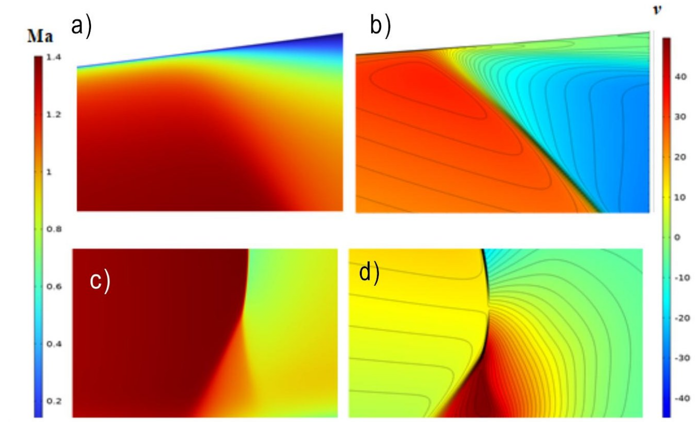


Fig. 12 TEA $O(h^4)$ Sajben diffuser, strong-shock, turbulent boundary layer interaction, shock-centric zoom: a) Mach number in the top wall region, b) transverse velocity (v) in the top wall region, c) Mach number in the bottom wall region, and d) transverse velocity (v) in the bottom wall region.

Acknowledgements We would like to acknowledge the developers, and technical support personnel at `COMSOL` for providing a superb tool for building, simulating, and investigating finite-element based CFD models, and multiphysics models in general.

References

1. Baker, A. J. and Freels, J. D., "High Order, High Fidelity *Error Freed* CFD Mathematics," *Physics of Fluids*, Vol. n, 2024, pp. pp–qq, special issue article to be published.
2. Baker, A. J. and Freels, J. D., *Error FreeD CFD Mathematics*, 2024, monograph draft, to be published.
3. Freels, J. D., "A Taylor Weak-Statement Finite Element Algorithm for Real-Gas Computational Fluid Dynamics," A Dissertation Presented for the Doctor of Philosophy Degree, The University of Tennessee, Knoxville, May, 1992., http://trace.tennessee.edu/utk_graddiss/4724.
4. Huang, T., Zahr, M. J., and others, ., "High-Fidelity CFD Verification Workshop 2024: Shock-Dominated Flows," *AIAA Aviation Forum and ASCEND co-located Conference Proceedings, Las Vegas, Nevada, 7/29-8/2*, , No. DOI 10.2514/6.2024-3695, 2024.
5. Freels, J. D. and Baker, A. J., "Investigation of 1D Compressible Navier-Stokes Equations using COMSOL Multiphysics Equation-Based Modeling," *COMSOL Conference 2020 North America proceedings*, COMSOL, Boston, MA, October 2020, best paper in conference award.
6. Towne, C., Yoder, D., and many others, "Sajben Transonic Diffuser: Study no. 1 and no. 2," *NPARC Alliance Validation Archive*, 2021, https://www.grc.nasa.gov/WWW/wind/valid/transdif/transdif01/transdif01.html and https://www.grc.nasa.gov/WWW/wind/valid/transdif/transdif02/transdif02.html.