

Shape, Convection and Convergence

Roger W. Pryor, Ph.D.*, VP Research
Pryor Knowledge Systems, Inc.

*Corresponding author: 4918 Malibu Drive, Bloomfield Hills, MI 48302-2253, rwpryor@pksez1.com

Abstract:

COMSOL Multiphysics software, when properly configured, can readily solve modeling problems in the laminar flow regime using the standard Navier-Stokes equations or in the fully turbulent flow regime using the kappa-epsilon model. Failure to solve a particular model is typically manifested by instability in the calculation and a failure of the model to converge.

This paper presents a new approach to achieving a convergent model solution through the reconfiguration of the shape of the modeled geometry. This shape-oriented approach stabilizes the convective flow and allows use of the standard Navier-Stokes equations.

Keywords: convection, convergence, laminar, Navier-Stokes, shape

1. Introduction

The models presented in this paper employed the COMSOL Multiphysics software and the Heat Transfer Module. The models developed and presented herein are 2D axially symmetric models of differently configured, dual-layer, insulated, heated, water tanks. The purpose of the information presented in this paper, through use of these models, is to demonstrate a first principles approach to the building and running of these models. A first principles approach was used to modify the modeling parameters and the tank geometry, so that the model was able to achieve convergence, in the typically difficult Navier-Stokes (NS) laminar flow environment.

The models presented in this work represent an expansion and modification of the approach to solution contained in two of the COMSOL Heat Transfer Module Model Library Models [1,2]. In the work contained herein, the primary factors that have been changed are the geometry of each of the six models, some of the boundary conditions and also certain meshing and solver parameters, to achieve model convergence in the potentially difficult NS modeling regime.

2. First Principles Applied to Fluid Flow

Calculation of fluid flow behavior is a complex and difficult area to model. The modeled behavior (free convection) of the fluid (air, in this case) depends highly on the external force (gravity, herein), the properties of the fluid (thermal conductivity, density, density gradient, viscosity, etc.) and the relative direction of the temperature gradient [3].

In a First Principles Analysis, analytical consideration needs to be used to determine which factors (materials properties, geometry, etc.) in the model in question predominantly contribute to the generation of unstable regions in these models that are presumed to have nominally laminar flow patterns.

In a free convection situation, the flow of air, laminar and/or turbulent, is driven by the temperature differential, the differential density, the geometry of the flow path and the gravitational force. For short path lengths, the gravitational force is substantially constant and is so treated in this paper. The primary driving force of the airflow (free convection) mechanism in this model is the temperature of the material in the tank (water). In this 2D Axisymmetric model, the surface geometry (shape) of the top of the tank is the primary factor that contributes to the creation of turbulence in the nominally laminar free convection airflow patterns.

In this paper, the temperature gradient is generated by heat leakage through the wall of the insulated tank. The turbulence in the free convection airflow is created by the degree of abruptness in the geometrical configuration of the tank. The free convection currents in the models reported herein comply with the laminar assumption (percent laminar versus percent turbulent) to a reasonable degree. This allows most of the models to converge to solution. The geometrical shape of the top of the tank is changed from model to model. This change is the predominant factor that dictates the degree to

which convergence is achieved (residual error being adjusted).

For those tanks with the most abrupt change of curvature (slope) between sidewall and the top, convergence fails. The First Principle dictating that those models will fail to converge stems from the basic physics of fluid flow. Local instability (mostly laminar flow plus a few percent turbulent flow) is directly induced in the flow pattern as a function of the curvature of the top surface, in the region closest to both the axis of the model and the top of the tank. As a result of this local instability, the calculated values for the properties of the convected material in the models (air, in this case) fail to stabilize in those regions of the models. Such instabilities are the result of the rotational/chaotic behavior imparted to the free convection current as it rises and expands beyond the vertical termination of the tank wall.

A comparative analysis of the results derived by modeling the same tank (dimensionally), with differently shaped tops is presented below. This analysis shows that there exists a mathematical boundary below which convergence fails.

3. Building the 2D Axisymmetric Model

This modeling calculation in this paper employs a two-stage process. The first stage develops a model derived from the “*thermos laminar hcoeff*” model in the COMSOL Heat Transfer Module Model Library, Tutorial Models folder. The first stage model is the starting point for the second stage free convection model based on the COMSOL, Heat Transfer Module, Model Library, Tutorial Models, “*thermos laminar flow*”. In this paper, detailed instructions for the building of one of the convergent two-stage models will be presented. The building of other models of different shapes is presented in the form of tabular guidelines, due to space limitations in this presentation.

3.1 The First Model: Creating the Water Tank

Start, by opening the COMSOL Multiphysics Model Navigator. Select: **Axial symmetry (2D)**.

Next, Select:

Heat Transfer Module>General Heat Transfer>Steady-state analysis>Click OK.

To create the Control Volume (airflow region), Select: **Draw>Specify Objects>Rectangle**. Enter in the edit windows: **Width=0.25, Height=0.55>Click OK>Click Zoom Extents**.

Creating the insulated water tank is a multi-step process. The first step is to create the outer-shell.

1. Select: **Draw>Specify Objects>Rectangle**. Enter in the edit windows: **Width=0.15, Height=0.25>Click OK**

2. Select: **Draw>Specify Objects>Ellipse**. Enter in the edit windows: **A-semiaxes=0.15, B-semiaxes=0.15>z=0.25>Click OK**

3. Select: **Draw>Specify Objects>Rectangle**. Enter in the edit windows: **Width=0.15, Height=0.4>r= -0.15>Click OK**

4. Select: **Draw>Create Composite Object**. In the **Set formula** edit window, Enter: **E1-R3>Unselect Keep Interior Boundaries>Click OK**.

5. Select: **Draw>Create Composite Object**. In the **Set formula** edit window, Enter: **R2+CO1>Unselect Keep Interior Boundaries>Click OK**.

Steps 1>5 yield CO2.

The outermost layer of the water tank has been formed. The next step is to create the inner boundary of the metallic outer shell (0.0025m).

6. Select: **Draw>Specify Objects>Rectangle**. Enter in the edit windows: **Width=0.15-0.0025, Height=0.25-0.0025>z=0.0025>Click OK**

7. Select: **Draw>Specify Objects>Ellipse**. Enter in the edit windows: **A-semiaxes=0.15-0.0025, B-semiaxes=0.15-0.0025>z=0.25>Click OK**

8. Select: **Draw>Specify Objects>Rectangle**. Enter in the edit windows: **Width=0.15, Height=0.4>r= -0.15>Click OK**

9. Select: **Draw>Create Composite Object**.
In the **Set formula** edit window,
Enter: **E1-R3>Unselect Keep Interior Boundaries>Click OK**.

10. Select: **Draw>Create Composite Object**.
In the **Set formula** edit window,
Enter: **R2+CO1>Unselect Keep Interior Boundaries>Click OK**.

Steps 6>10 yield CO3.

The inner boundary of the metallic outer shell has been formed. The next step is to form the inner boundary of the insulation layer (0.01m).

11. Select: **Draw>Specify Objects>Rectangle**.
Enter in the edit windows: **Width=0.15-0.0025-0.01, Height=0.25-0.0025-0.01>z=0.0125>Click OK**

12. Select: **Draw>Specify Objects>Ellipse**.
Enter in the edit windows: **A-semiaxes=0.15-0.0025-0.01, B-semiaxes=0.15-0.0025-0.01>z=0.25>Click OK**

13. Select: **Draw>Specify Objects>Rectangle**.
Enter in the edit windows: **Width=0.15, Height=0.4>r= -0.15>Click OK**

14. Select: **Draw>Create Composite Object**.
In the **Set formula** edit window,
Enter: **E1-R3>Unselect Keep Interior Boundaries>Click OK**.

15. Select: **Draw>Create Composite Object**.
In the **Set formula** edit window,
Enter: **R2+CO1>Unselect Keep Interior Boundaries>Click OK**.

Steps 11>15 yield CO4.

The inner boundary of the metallic outer shell has been formed. The next step is to form the inner boundary of the inner metallic shell (0.0025m).

16. Select: **Draw>Specify Objects>Rectangle**.
Enter in the edit windows: **Width=0.15-0.0025-0.01-0.0025, Height=0.25-0.0025-0.01-0.0025>z=0.015>Click OK**

17. Select: **Draw>Specify Objects>Ellipse**.
Enter in the edit windows: **A-semiaxes=0.15-0.0025-0.01-0.0025, B-semiaxes=0.15-0.0025-0.01-0.0025>z=0.25>Click OK**

18. Select: **Draw>Specify Objects>Rectangle**.
Enter in the edit windows: **Width=0.15, Height=0.4>r= -0.15>Click OK**

19. Select: **Draw>Create Composite Object**.
In the **Set formula** edit window,
Enter: **E1-R3>Unselect Keep Interior Boundaries>Click OK**.

20. Select: **Draw>Create Composite Object**.
In the **Set formula** edit window,
Enter: **R2+CO1>Unselect Keep Interior Boundaries>Click OK**.

Steps 16>20 yield CO1.

Next, the volume occupied by the water is created.

21. Select: **Draw>Specify Objects>Rectangle**.
Enter in the edit windows: **Width=0.15-0.0025-0.01-0.0025, Height=0.25-0.0025-0.01-0.0025>z=0.015>Click OK**

22. Select: **Draw>Create Composite Object**.
In the **Set formula** edit window,
Enter: **CO2+CO3+CO4+CO1+R2 >Select Keep Interior Boundaries>Click OK**.

Select: **File>Save**

3.2 The First Model: Constants

Select: **Options>Constants**

Enter the following values:

| | | |
|------------------|----------------------|-----------------------|
| k_insul | 0.01[W/(m*K)] | Insulation k |
| p_atm | 1[atm] | Air pressure |
| Length | 0.4[m] | Height of tank |
| T_water | 95[degC] | Water Temp. |
| T_amb | 20[degC] | Ambient Temp. |
| rho_insul | 60[kg/m^3] | Density Insul. |
| Cp_insul | 200[J/(kg*K)] | Heat Capacity |
| Length2 | 0.15[m] | Width of Tank |

Where **Length** and **Length2** are the projected height of the vertical wall and width of the horizontal top of the water tank, respectively.

Save the **Constants** file for use in the other shape related models. The constant **Length** will need to be adjusted to the appropriate projected value as different size elliptical tops are created for different models.

3.2 The First Model: Heat Transfer (HT) Subdomain Settings

Select **Physics>Subdomain Settings**

Select from **Subdomain selection** window, Subdomains 4 and 6, clear the **Active in this domain** check box,

Enter the following values in the **Subdomain Settings** edit window:

| | |
|-------------|----------------------------------|
| Subdomain # | Basic Materials Library (BML) |
| 1 | BML Load: Structural Steel |
| 2 | Constants: Insulation Properties |
| | k_insul |
| | rho_insul |
| | Cp_insul |
| 3 | BML Load: Structural Steel |
| 5 | BML Load: Air |

In the density expression for **Air**, replace **p** with **p_atm**.

3.3 The First Model: Boundary Settings

Select: **Physics>Boundary Settings**

Select: Boundaries 8, 14, and 20 in the **Boundary selection** window. Change the **Boundary condition** to **Temperature**. Enter **T_water** in the **T₀** edit window.

Select: Boundary 23 in the **Boundary selection** window. Change the **Boundary condition** to **Heat Flux**. Load the **Nat. Vertical Wall ...** into the **h (Heat transfer coefficient)** edit window. Change **L_htgh** to **Length**. Enter **T_amb** in the **T_{inf}** edit window.

Select: Boundary 29 in the **Boundary selection** window. Change the **Boundary condition** to

Heat Flux. Load the **Nat. Horiz. Plane, Upside ...** into the **h (Heat transfer coefficient)** edit window. Change **L_htgh** to **Length2**. Enter **T_amb** in the **T_{inf}** edit window.

3.4 The First Model: Mesh and Solve

Click **Solve**

Now that the first model is solved, Select: **File > Save** and save the solved model under a new file name.

3.4 The Second Model: Adding Navier-Stokes (N-S)

Open the new model. Select **Multiphysics>Model Navigator>Heat Transfer Module>Weakly Compressible Navier-Stokes>Steady-state analysis>Click Add>Click OK**.

Select: **Options>Expressions>Scalar Expressions**. Enter the following expressions:

| Name | Expression |
|-------------|---|
| F_bouyancy | 9.81[m/s^2]*(rho_air_ref-rho_chns) |
| rho_air_ref | mat2_rho(p_atm[1/Pa], T_amb[1/K])[kg/m^3] |

3.5 The Second Model: N-S Subdomain Settings

Select **Physics>Subdomain Settings**

Select from **Subdomain selection** window, Select Subdomains **1 – 5** clear the **Active in this domain** check box, Click **Apply**

Select from **Subdomain selection** window, Select: Subdomain **6>** Select **Air** from the **Library material** list. Enter **F_bouyancy** in the **F_z** edit window. Click the **Density** tab.

Replace **rho(p[...]** with **rho((p+p_atm)[...]**, Click **Apply**.

Select the **Pressure p** and **Temperature** check boxes. Enter **T** in the Temperature edit window. **Click OK**.

3.5 The Second Model: Physics Settings HT Subdomain Settings

Select: **Multiphysics>General Heat Transfer>Physics>Subdomain settings**.

Select from **Subdomain selection** window, Select: Subdomain 6> Select the **Active in this domain** check box. Select **Air** from the **Library material** list. Click the **Convection** tab. Select the **Enable convective heat transfer** check box.

Replace $\rho(p[...]$ with $\rho((p+p_{atm})[...]$
In the **Density** edit window. Click **Apply**

Enter **u** and **v** in the **Velocity field** edit windows.

Click the **Artificial Diffusion** button. Select the **Streamline diffusion** check box. Select **Galerkin least-squares (GLS)** from the list. Click **OK> Click OK**.

HT Boundary Conditions

Select: **Multiphysics>General Heat Transfer>Physics>Boundary Settings**.

Select Boundary 18 from the **Boundary selection** window, set the **Boundary condition** to **Axial symmetry**.

Select Boundary 19 from the **Boundary selection** window, set the **Boundary condition** to **Convective flux**.

Select Boundary 24 from the **Boundary selection** window, verify that the **Boundary condition** is set to **Thermal insulation**.

Select Boundary 25 from the **Boundary selection** window, set the **Boundary condition** to **Temperature**. Enter **T_amb** in the **T₀** edit window. Click **OK**.

N-S Boundary Conditions

Select **Multiphysics>Weakly Compressible Navier-Stokes>**

Select Boundary 18 from the **Boundary selection** window, set the **Boundary type** to

Symmetry boundary and the **Boundary condition** to **Axial symmetry**.

Select Boundary 19 from the **Boundary selection** window, set the **Boundary type** to **Outlet**, the **Boundary condition** to **Normal stress** and **f₀** to **0**.

Select Boundary 25 from the **Boundary selection** window, set the **Boundary type** to **Open boundary**, the **Boundary condition** to **Normal stress** and **f₀** to **0**.

Confirm that Boundaries 23, 24, and 29 have a **Boundary type** of **Wall** and a **Boundary condition** of **No slip**. Click **OK**.

3.6 The Second Model: Meshing

Select: **Mesh>Free Mesh Parameters**
Select the **Subdomain** tab
In the **Subdomain selection** window, select **Subdomain 6** > Enter **0.005** in the **Subdomain mesh parameters** edit window> Select **Quad**> Click the **Remesh** button>Click **OK**

3.6 The Second Model: Solving

Select: **Solve>Solver Parameters**
Click the **Stationary** tab
Select the following Check boxes:
Damped Newton, Highly nonlinear problem, Manual tuning of damping parameters.

Enter the following parameters:

| | |
|-------------------------------|----------------|
| Relative tolerance | 1.0E-2 |
| Maximum ... iterations | 50 |
| Initial damping factor | 1.0E-4 |
| Minimum damping factor | 1.0E-12 |

Click on the **Advanced** tab

| | |
|------------------------|-------------|
| Type of scaling | none |
|------------------------|-------------|

Click **OK**

Select **Solve**

3.7 The Second Model: Postprocessing

Select: **Postprocessing>Plot Parameters**

In the **Plot Parameters** window, Click the **General** tab.

Select the following check boxes: **Surface**, **Streamline**, and **Geometry Edges**.

In the **Surface** window, Select the **Surface Plot** check box. Select the **Surface Data** tab. Select **Temperature** from the **Predefined quantities** list and Select **degC** from the **Unit:** list.

Select the **Streamline** tab. In the **Streamline** window, Select the **Streamline plot** check box. Select the **Streamline Data** tab. Select: **Weakly Compressible Navier-Stokes>Velocity field** from the **Predefined quantities** list. Select: **Magnitude controlled** from the **Streamline plot type** list. **Click OK.**

Result of the above modeling effort of a Water Tank Top with Aspect Ratio 1.0 is as shown in Figure 1 below.

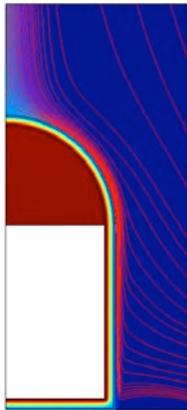


Figure 1 Water Tank Top Aspect Ratio 1.0

4.0 The Second Model: Shape Variation Influence on Convergence and Solution(s)

Six different N-S models were built and run to determine the influence of the curvature of the elliptical top of the water tank on the convergence of the solution of the COMSOL Multiphysics model in the Control Volume. Each of the models built had an elliptical top with a different combination of A-semiaxes and B-semiaxes values. Those values are shown in Table 1 below.

| Model# | A | B | B/A | Convergence |
|--------|------|-------|-------|-------------|
| 1 | 0.15 | 0.15 | 1 | Yes |
| 2 | 0.15 | 0.20 | 1.33 | Yes |
| 3 | 0.15 | 0.05 | 0.33 | Yes |
| 4 | 0.15 | 0.03 | 0.20 | No |
| 5 | 0.15 | 0.04 | 0.267 | Yes |
| 6 | 0.15 | 0.033 | 0.22 | Yes |

The results of these calculations are shown in Figures 2-6 below.

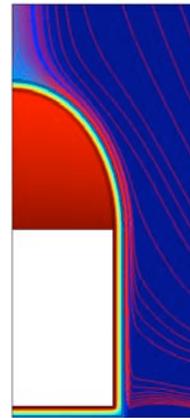


Figure 2 Water Tank Top Aspect Ratio 1.33

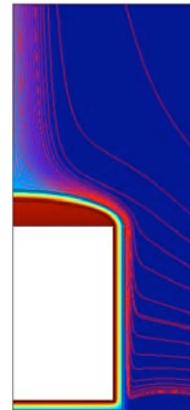


Figure 3 Water Tank Top Aspect Ratio 0.33

A binary search was run on Convergence vs. Aspect Ratio. The results of that search show that models with Aspect Ratios > 0.22 converge. Models with Aspect Ratios < 0.20 do not converge. The streamline modeling results are shown in Figures 6 and 4, respectively.

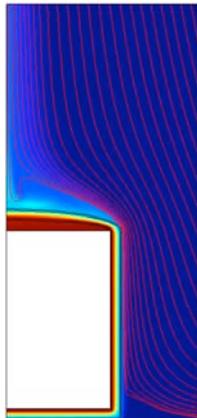


Figure 4 Water Tank Top Aspect Ratio 0.20

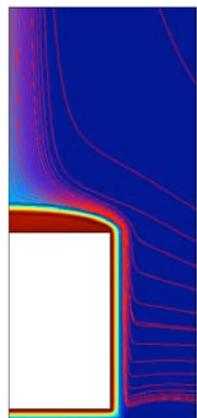


Figure 5 Water Tank Top Aspect Ratio 0.267

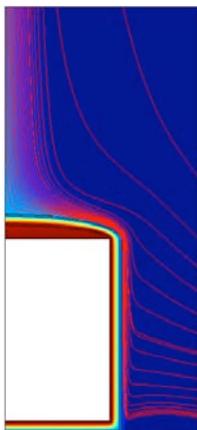


Figure 6 Water Tank Top Aspect Ratio 0.22

Please note the deviations (double-valued loops) from the nominally normal flow patterns exhibited close to the 2D-axis at the top of the

tank and also close to the bottom of the tank near the tank wall in Figure 4.

Solver tuning was achieved through adjustment of the Relative tolerance, the Maximum number of iterations, and the addition of Manual tuning of damping parameters.

7. Conclusions

Through careful adjustment of a modeled object's shape and proper tuning of a model's solver parameters, a modeler can, in many cases, obtain convergence in Navier-Stokes convection models that nominally fail to converge.

8. References

1. COMSOL, Heat Transfer Module, Model Library, Tutorial Models, *thermos laminar hcoeff*
2. COMSOL, Heat Transfer Module, Model Library, Tutorial Models, *thermos laminar flow*
3. F.P. Incropera and D.P. DeWitt, Free Convection, **Heat and Mass Transfer, Fifth Edition**, John Wiley and Sons, NJ, Chapter 9, pp 533-593(2002)