Modeling Sudden Expansion in the Benchmark FDA Nozzle Using Abaqus/CFD

Abaqus Technology Brief

Acknowledgement

Dassault Systèmes SIMULIA would like to thank Drs. Santanu Chandra and Richard Swift of MED Institute Inc. for their simulation study of the FDA Nozzle model.

Summary

Computational Fluid Dynamics (CFD) methods are an integral part of the medical device design process. For devices that interact with blood, CFD can describe complex flow patterns, can further the understanding of forces exerted on the blood, and can help predict blood damage.

In an effort to standardize the use of CFD methods, the US Food and Drug Administration (FDA) engaged the worldwide medical device/CFD community in an open challenge involving CFD computation of blood flow dynamics on a benchmark nozzle model.

In this Technology Brief, we discuss the sudden expansion problem of the FDA Benchmark Nozzle model. Overall, Abaqus/CFD results were found to match reasonably well with experimental data and were competently at par or superior to other software package results. The k-ω SST model was found to fare the best among the three turbulence models for the higher Reynolds number cases.

Background

Understanding blood flow dynamics (or hemodynamics) is a key process in the design and performance assessment of implantable medical devices. Computational approaches for modeling hemodynamics and blood damage are promising techniques that are capable of delivering important thrombogenic information prior to rigorous testing of the device in a bench top setting or in animal models. Such modeling techniques, when used effectively, can save time and resources in the device design process and also ensure patient safety.

Though CFD methods are increasingly being used to model hemodynamics, results can vary widely depending on the analyst's expertise, modeling and meshing techniques, choice of solver type, choice of software, and strategies followed for verification and validation. Therefore there is a vital need to identify best practices and define the critical techniques necessary for reliable results.

With an aim to further standardize CFD practice, the FDA took an initiative to engage the worldwide medical device/CFD community in an open challenge that involved CFD computation of blood flow dynamics and blood damage on a benchmark nozzle model. Experimental validation was provided by Particle Image Velocimetry (PIV).

FDA's "Critical Path" CFD/Blood Damage Project was initiated in 2008 and experimental and computational studies were completed and published in 2011 [1] and 2012 [2]. During the challenge 28 groups around the world participated and presented results using different in-house and commercial CFD software.

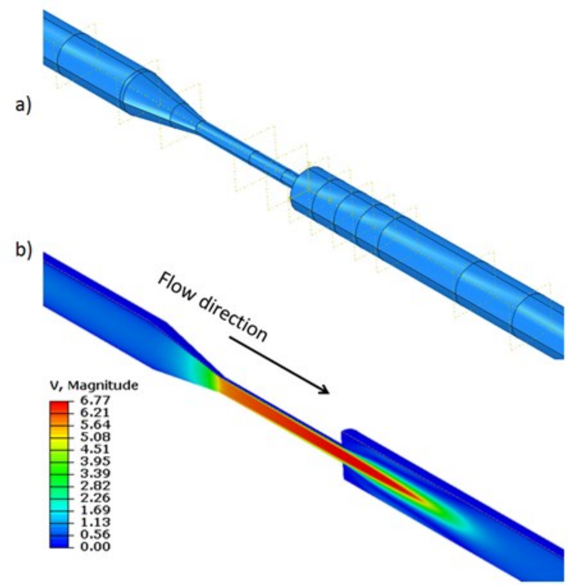


Figure 1: (a) Geometry of the FDA nozzle created in Abaqus/CAE (b) representative velocity characteristics of the flow through the nozzle

Key Abaqus/CFD Features and Benefits

- Conveniently integrated into the Abaqus/CAE environment
- No specific additional software is required for full CFD simulation
- Steady state solver
- Spalart-Allmaras, k-ε RNG and k-ω SST turbulence models

The FDA nozzle model consists of a straight tube (12 mm in diameter), a conical section that reduces the tube diameter to 4 mm in the throat region, and a small straight throat section which is followed by a sudden increase in tube diameter to 12 mm. The device was designed to include accelerating flow, decelerating flow, sudden expansion or gradual diffusion, variations in shear stress and velocities, and recirculating flow—all of which may be related to blood damage in medical devices. This study focused on the sudden expansion problem.

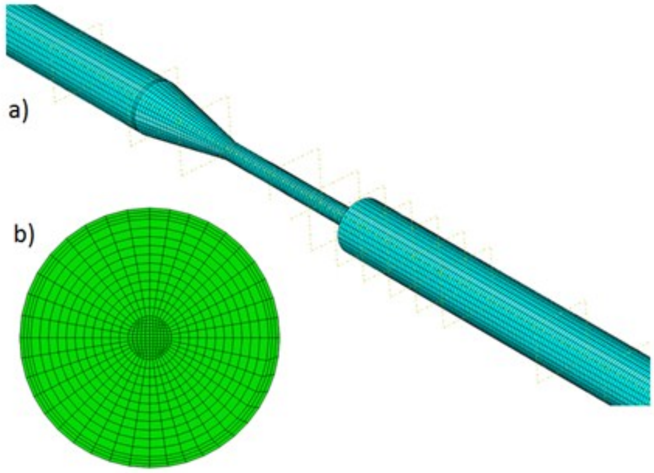


Figure 2: (a) The nozzle model meshed with structured hex elements (b) close view of the mesh at the inlet surface

The model was simulated with 5 different flow rates, with the inlet Reynolds number falling in the range of 167 to 2167. However in the throat region, due to acceleration of the flow, the Reynolds number range is 500 to 6500, which covers the fully laminar to fully turbulent range. The choice of solver type is thus significant for each simulation with different flow rates.

In this Technology Brief we show that Abaqus/CFD is a comprehensive tool, capable of solving laminar, transitional and turbulent flows. Abaqus/CFD can be used within the Abaqus/CAE environment in all three stages of analysis; modeling and meshing, solving, and post-processing. Abaqus/CFD has a robust steady state solver that uses a 2nd order accurate SIMPLE (Semi-implicit Method for Pressure Linked Equation) based algorithm. Three turbulent flow models are available — a one equation Spalart-Allmaras model, and two equation k-ε RNG and k-ω SST models.

Analysis Approach

Geometry and Model

The geometry of the nozzle was developed in Abaqus/CAE according to the published dimensions [1]. The inlet and outlet lengths were not specified in the challenge but left to the user to determine. After initial trials, the inlet length was chosen to be 20 times the inlet diameter and the outlet length to be 60 times the outlet diameter in order to achieve a fully developed velocity profile at the outlet. Figure 1 shows solid and sectioned views of the CAD model geometry.

Mesh

The model was subdivided into volume domains along the length and radius to allow meshing with hexahedral elements. The near-wall mesh was refined to include 3 layers of elements. A mesh refinement study was performed, resulting in a mesh with 1.39 million linear elements used for all analyses. The isometric view is shown in Figure 2a and a close-up view of the boundary layer mesh is shown in Figure 2b.

Material

Blood was modeled as an incompressible fluid with a density of 1056 kg/m^3 , and was assumed to be Newtonian with dynamic viscosity of 0.0035 Ns/m^2 as per the guidelines of the challenge [1].

Boundary Conditions

Simulations were performed with 5 different inlet velocities ranging from 5.21e-6 m/s to 6.77e-5 m/s as documented in [1]. The flow rates were labeled as Q1, Q2, Q3, Q4 and Q5. The corresponding inlet Reynolds numbers were 167, 667, 1167, 1667 and 2167. The respective Reynolds numbers in the throat region were 500, 2000, 3500, 5000 and 6500; therefore the flow type ranged from fully laminar to fully turbulent in the sudden expansion region. In this study, flow rate Q1 represents fully laminar flow, flow rate Q3 represents a transitional flow, and flow rate Q5 represents fully turbulent flow. A parabolic velocity profile was used to model the inlet flow distribution. The outlet pressure was set to zero and a no-slip boundary condition was applied to the wall surface.

Analysis Type

Steady state incompressible analyses were performed. The under-relaxation factors were chosen as 0.7 for the momentum equation and 0.3 for the pressure equation, allowing for a converged solution in 1000-1500 iterations.

For the lowest flow rate, only the laminar flow model was used. For the remaining flow rates, three turbulent flow models were used along with the laminar model for comparison. The Spalart-Allmaras model is a one equation Reynolds Averaged Navier Stokes (RANS) model that solves for Reynolds stress when the kinematic turbulent viscosity is known. The k-ε RNG and k-ω SST models are two equation RANS models that solve for Reynolds stress and include two transport equations for the turbulent properties of the flow, specifically turbulent kinetic energy (k), energy dissipation rate (ε), and specific energy dissipation rate (ω). The k-ε RNG model is a variant of k-ε models where the Re-Normalization Group (RNG) method is used to re-normalize the Navier-Stokes equation. The k-ω SST model is a two equation eddy viscosity model where a Shear Stress Transport (SST) formulation is combined with a standard k-ω model and represents the "best of both" respective models. The variables for the transport equation are available in Abaqus/CFD whereas the parameters k, ε and ω can be calculated depending on the inlet velocity, turbulence length scale and turbulence intensities. Table 1 summarizes the parameter values used for 4 different flow rates (Q2, Q3, Q4 and Q5). For the Spalart-Allmaras model the eddy viscosity term was assumed to be 4 times the kinematic viscosity of blood.

Results & Discussion

Simulation results were post-processed in Abaqus/CFD and directly compared to the FDA experimental data [3]. Centerline velocity in the axial direction and pressure data were recorded using the 'path' feature in the Visualization module and are plotted in Figure 3. The data obtained were for three different flow rates that are representative of fully laminar (Q1), transitional (Q3) and fully turbulent

	u_0 (inlet)	k	eps (ε)	omega(ω)	Turbulence Intensity	length scale
Re = 2000	0.1839	1.6482E-04	3.9654E-04	28.31	0.057	8.40E-04
Re = 3500	0.3218	5.0468E-04	2.1247E-03	49.53	0.057	8.40E-04
Re = 5000	0.4606	1.0339E-03	6.2304E-03	70.89	0.057	8.40E-04
Re = 6500	0.5986	1.7463E-03	1.3676E-02	92.13	0.057	8.40E-04

Table 1: Turbulence parameters used for the k-ε RNG and k-ω SST models, for flow rates corresponding to throat Reynolds number

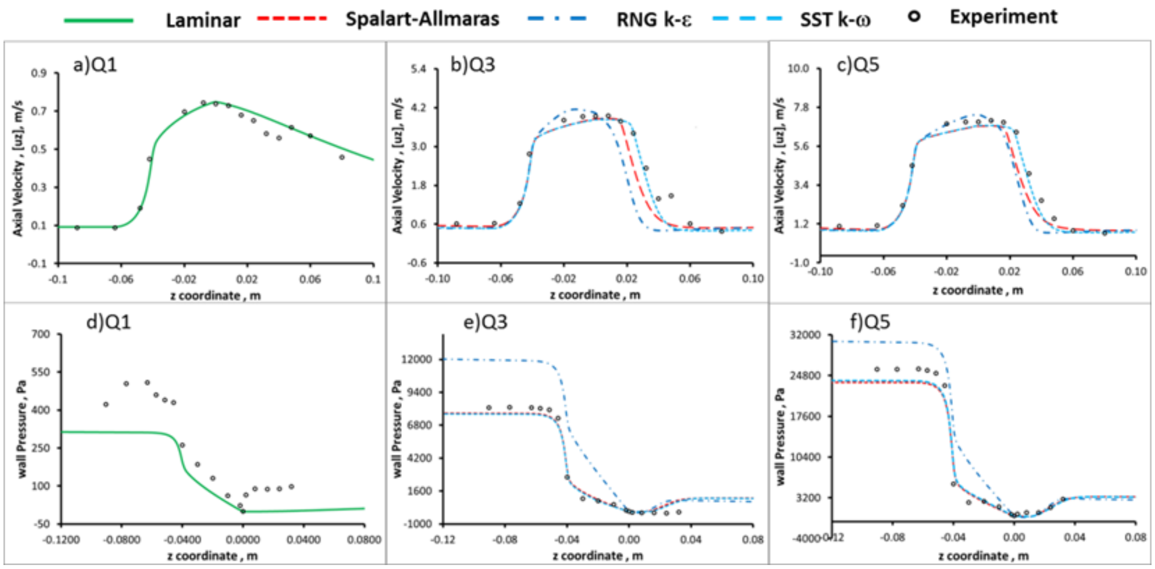


Figure 3: Centerline velocity in the axial direction for flow rates Q1 (a), Q3 (b), and Q5 (c). Wall pressure (represented by the centerline pressure) for flow rates Q1 (d), Q3 (e), Q5 (f). Flow rates Q1, Q3, and Q5 are representative of fully laminar, transitional, and fully turbulent flows, respectively.

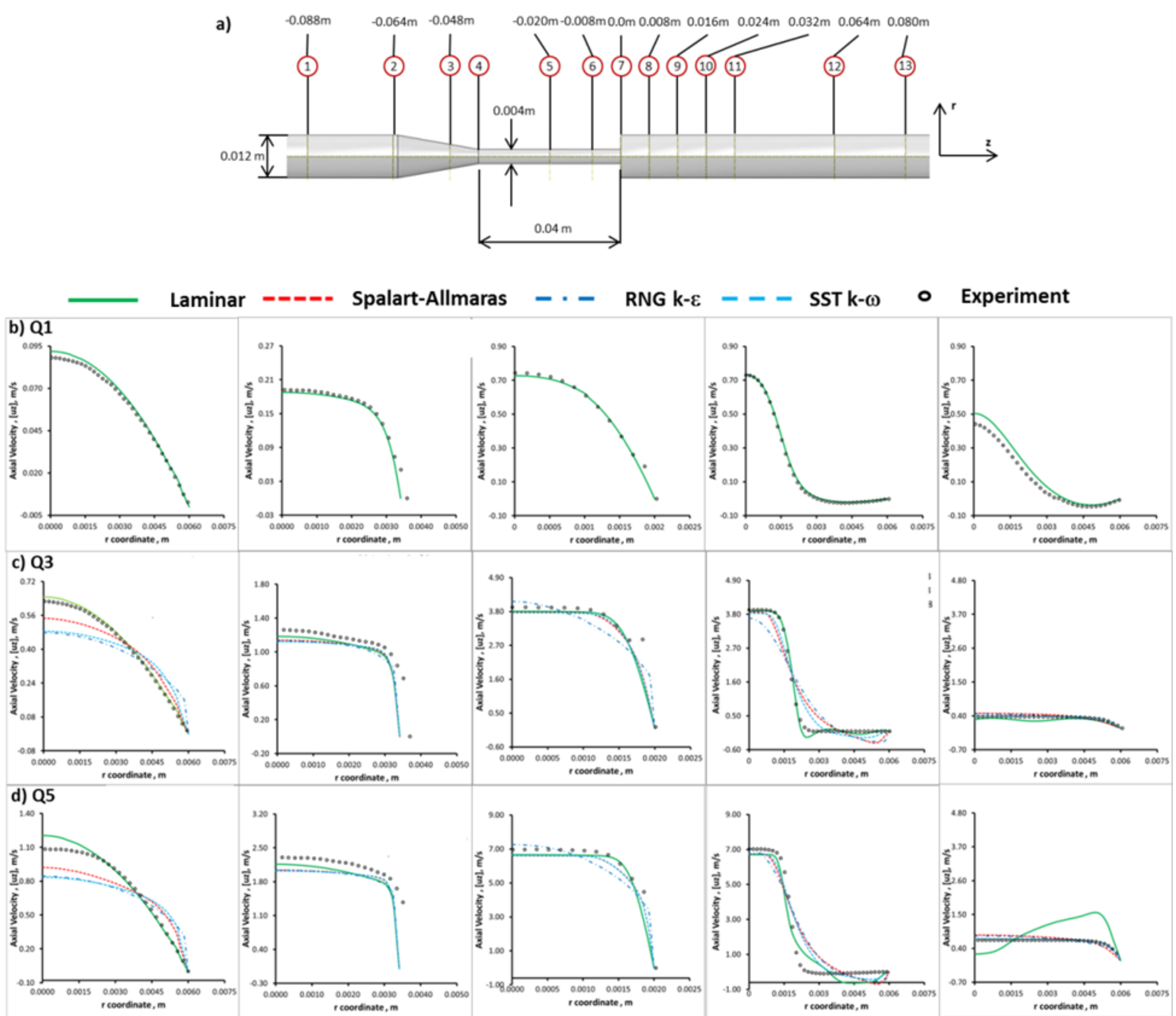


Figure 4: a) Cross sectional surfaces are numbered in increasing order from inlet to outlet. Axial velocity in radial direction at specific cross sectional surfaces (Surf-1, Surf-3, Surf-6, Surf-8, and Surf-13) are obtained from laminar and turbulent simulations and plotted along each row from left to right for three different flow rates: (b) Q1, (c) Q3, and (d) Q5.

(Q5) flow types. Because the radial pressure gradient is small, wall pressure is represented by the centerline pressure. The experimental data presented in Figure 3 are an average of 3 different experimental data sets obtained from the FDA repository [3]. The experimental and computational pressure values are referenced with respect to the pressure at the location of the sudden expansion (z=0.0). Abaqus simulation results matched closely with the experimental data for laminar, transitional and turbulent flow. As the flow through the nozzle before the sudden expansion (z=0) region is truly laminar, the centerline axial velocity characterized by the laminar flow model was in good agreement with the experimental data for all three flow rates. Flow beyond the sudden expansion zone is complicated and none of the models were in strong agreement with the experimental data. Similar behavior was reported by the other contributors to the challenge and was comprehensively discussed in the FDA publication [2]. At the sudden expansion zone the laminar flow model did well for the lower flow rate and demonstrated fluctuations for transitional and fully turbulent flow rates. Here the k-ω SST and Spalart-Allmaras models achieved the best agreement with the experimental data.

Pressure is plotted in Figures 3(d-f). For flow rate Q1 (Figure 3d) pressure did not match with the experimental data. As reported in [2], contributions from other studies also did not match well at rate Q1; the differences are attributed to experimental error in pressure measurement, specifically erroneous reference scaling and failure to use differential pressure transducers. For higher flow rates (Q3 & Q5), most discrepancy in pressure was observed for the k-ε RNG model. It is well known that k-ε RNG does not perform well at transitional Reynolds numbers, which is reflected in cases Q3 and Q5. In addition, the impact of the wall mesh resolution requirement (y+ near the wall) could also be a contributing factor to the deviation from experimental results. Similar behavior of the k-ε model was observed by users of other software [2]. Though none of the turbulent flow models were an exact match with the experimental data at each location, the Spalart-Allmaras and k-ω SST models did a better job in characterizing the axial velocity and pressure before and beyond the sudden expansion zone, whereas the laminar model results deviated slightly in the sudden expansion region.

The nozzle model was subdivided such that 13 cross sectional surfaces can be created as shown in Figure 4a. Five surfaces were selected for results reporting. Surf-1 is closest to inlet, Surf-3 is in the conical section, Surf-6 is in the throat region, Surf-8 is in sudden expansion zone, and Surf-13 is furthest from the sudden expansion region. Axial velocities along the radial direction at these cross-sections were recorded for flow rates Q1, Q3, and Q5, and presented in Figures 4b, c, and d respectively. The experimental data presented is from one set obtained from the FDA repository [3] that best matches the average experimental data. As expected, velocity profile from the laminar flow model matched very well before the sudden expansion zone for all three flow rates. As the flow rate increases, the turbulent flow models did a better job in matching the experimental results at and beyond the sudden expansion region, demonstrated clearly by the Surf-13 results in Figures 4c and 4d. Additional hemodynamic parameters such as shear rate and wall shear stress at selected cross sectional surfaces and on the wall of the nozzle model are presented in [4]. In summary, the results are in good overall agreement with the experimental data and were satisfactory, particularly in light of the wide variability of data presented in the challenge.

Conclusions

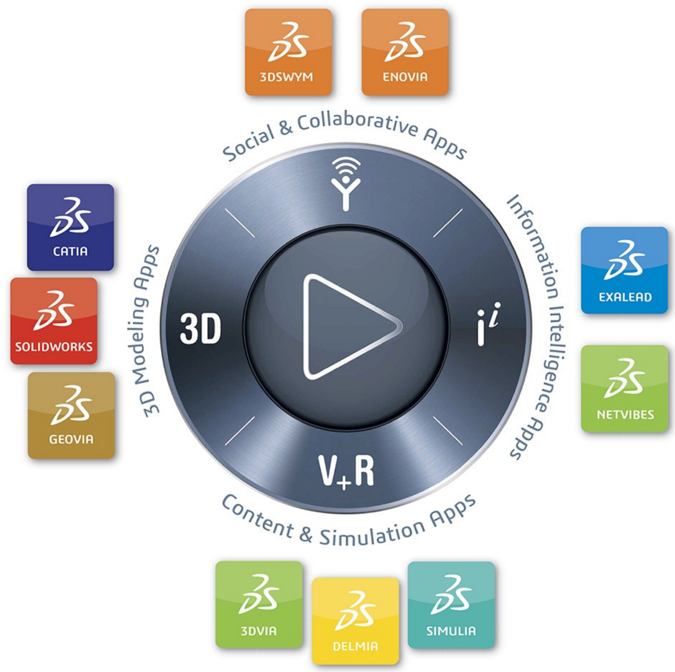
CFD analyses in the design of medical devices can be very challenging; in particular, a mixture of flow regimes may exist in the model, necessitating care in the use of turbulence modeling.

In this Technology Brief, Abaqus/CFD was used to simulate and visualize the flow through a benchmark FDA nozzle model and the simulation results were validated by experimental data. We have shown that, with careful use, the capabilities of Abaqus/CFD can be successfully applied to the analysis of flow problems typical of the medical device design industry.

References

1. Hariharan, P. et al., *Multilaboratory Particle Image Velocimetry Analysis of the FDA Benchmark Nozzle Model to Support Validation of Computational Fluid Dynamics Simulations*, Journal of Biomechanical Engineering, April 2011, Vol. 133 / 041002-1
2. Stewart, S. et al., *Assessment of CFD Performance in Simulations of an Idealized Medical Device: Results of FDA's First Computational Interlaboratory Study*, Cardiovascular Engineering and Technology, Vol. 3, No. 2, June 2012, pp. 139–160
3. FDA Critical Path Initiative-Computational Round Robin #1 (https://fdacfd.nci.nih.gov/interlab_study_1_nozzle)
4. Chandra, S. et al., *Modeling Hemodynamics with Abaqus/ CFD Steady State Solver: FDA Benchmark Nozzle Model*, Simulia Communityv Conference, May19-22, 2014, Providence, RI,USA

About SIMULIA



Our **3D**EXPERIENCE Platform powers our brand applications, serving 12 industries, and provides a rich portfolio of industry solution experiences.

Dassault Systèmes, the **3D**EXPERIENCE Company, provides business and people with virtual universes to imagine sustainable innovations. Its world-leading solutions transform the way products are designed, produced, and supported. Dassault Systèmes' collaborative solutions foster social innovation, expanding possibilities for the virtual world to improve the real world. The group brings value to over 170,000 customers of all sizes in all industries in more than 140 countries. For more information, visit **www.3ds.com**.

Europe/Middle East/Africa
Dassault Systèmes
10, rue Marcel Dassault
CS 40501
78946 Vélizy-Villacoublay Cedex
France

Asia-Pacific
Dassault Systèmes
Pier City Shibaura Bldg 10F
3-18-1 Kaigan, Minato-Ku
Tokyo 108-002
Japan

Americas
Dassault Systèmes
175 Wyman Street
Waltham, Massachusetts
02451-1223
USA

DASSAULT SYSTEMES | The **3D**EXPERIENCE Company