ANSYS Workbench Release 16.0 Project: Fluid-structure interaction model of a patient-specific arteriovenous access fistula

W. P. Guess, B. D. Reddy, A. McBride, T. Franz

This ANSYS Workbench project forms part of an M.Sc. thesis and publication both titled "Fluid-structure interaction modelling of a patient-specific arteriovenous access fistula". In this project an ANSYS Fluent (FVM) blood flow model is coupled with an ANSYS Mechanical (FEM) vessel wall model in a semi-implicit manner by the ANSYS Workbench Coupling Component. A Windkessel boundary condition is applied to the fluid through a UDF in Fluent. Command snippets are used to use the mixed uP formulation and enhanced strain elements. A combination of relaxation techniques, including boundary stabilisation (which can interact poorly with the Windkessel boundary condition), are used to stabilise the "added-mass effect", a numerical instability inherent in partitioned FSI schemes.

The thesis contains the background, discussion, and results of this project in detail (open access handle - http://open.uct.ac.za/handle/11427/22840).

Prerequisites

ANSYS Workbench 16.0, ANSYS Fluent, and ANSYS Mechanical with licenses.

Installation

See https://www.sharcnet.ca/Software/Ansys/17.0/en-us/help/ai_instl/win_install.html

Getting Started

Download the archived workbench file - Patient-specific_fistula_FSI.wbpz and open the file in ANSYS Workbench 16.0. The stabilisation parameters used here did not work when running the model in the future release of ANSYS (Workbench 17.2), reasons for this are unknown and therefore care should be taken when using any other version of ANSYS to run this model.

Running the FSI model

To improve simulation performance it is advised that hyperthreading on the machine is turned off.

a. **Initialising the FSI model (from non-initialised archive)**
   The FSI model should be initialised to start the simulation from a somewhat stable point. This is done by running the FSI model through a number of time steps the size of the cardiac-cycle (0.8s) with 1-element Windkessel outlet boundary conditions.

   i. **Setting up the Fluent blood flow model**

1. In Workbench double-click on the **Fluent system F (ID: FLU 17) Setup** cell to bring up the **Fluent Launcher** window
2. Choose the **Number of Processors** you would like Fluent to use and click **OK** (Mechanical will require some of the remaining processors - on a 16 core machine it was best to use 12 for Fluent and 4 for Mechanical - on a 4 core machine it was best to use 2 for Fluent and 2 for Mechanical). Fluent may partition the mesh through one of the outlet boundaries if more than 12 processors are used, the Windkessel outlet UDF will not function correctly in this case (the UDF needs some work to be parallelised correctly) - see the next step for more details.
3. Click: **User-Defined** > **Execute on demand** > "pre initialise"> **Execute**. Ensure that there are 480 faces on the two outlets **ARTERY** and **VEIN** in the **Console** panel, if this is not the case the **Number of Processors** likely needs to be reduced in the previous step.
4. Initialise with zero for all **Initial Values** using **Standard Initialization**
5. Click: **User-Defined** > **Execute on demand** > "windkessel1_on"> **Execute** > **Close**
6. In **Dynamic Mesh** > **Dynamic Mesh Zones** - double-click on "**fsi_fistula - System Coupling**" > open the "**Solver Options**" panel > ensure that "**Solution Stabilization**" is unselected.
7. In **Dynamic Mesh** > **Dynamic Mesh Zones** - double-click on **fsi_fistula - System Coupling** > open the **Solver Options** panel > unselect **Solution Stabilization** > **Create** > **Close**.
8. In **Solution Controls** - ensure that the **Under-Relaxation Factors** for **Pressure** and **Momentum -** are 0.01 and 0.023 respectively.
9. In **Run Calculation** - set **Max Iterations/Time Step** to 8.
10. Click: **File** > Save **Project**
11. A.1.8. Leave Fluent open and return to Workbench and click **Refresh Project**

ii. Setting up the System Coupling

1. In Workbench double-click on the **System Coupling system J (ID: SC) Setup** cell to bring up the System Coupling panel
2. In **Setup** > **Data Transfers** > **Force Transfer** > **Data Transfer Control** - set **Ramping** to the **none** option.
3. In **Setup** > **Analysis settings** - set **End Time** to 4s , **Step Size** to 0.8s (the period of the cardiac-cycle), **Minimum Iterations** to 2, and **Maximum Iterations** to 6.
4. Click: **File** > **Save** to save the Workbench project
5. Click: **Update** to start running the initialisation process - Check that the **Mapping Summary** in the **Solution Information : System Coupling** Panel reads:

```
+-------------------------------------------------------------+
|                      MAPPING SUMMARY                        |
+-------------------------------------------------------------+
| Data Transfer              |                                |
|   Diagnostic               | Source Side    Target Side     |
+----------------------------+--------------------------------+
| Disp Transfer              |                                |
|   Percent Nodes Mapped     |     N/A           100          |
```

```
| Force Transfer            |                          |
|   Percent Nodes Mapped |     100          100        |
|   Percent Area Mapped   |     100          100        |
+----------------------------------------------------------------+
```

b. Starting the FSI simulation (from initialised state, completed simulation, or saved state)

To run the FSI model from the initialised state ... start the simulation from a somewhat stable point. This is done by running the FSI model through a number of time steps the size of the cardiac-cycle (0.8 s) with 1-element Windkessel outlet boundary conditions.

i. Setting up the Fluent blood flow model for the default time step size 5 ms

1. If **Fluent system F (ID: FLU 17)** is not already open, open it in Workbench by double-clicking on its **Solution** cell. You may change the Choose the **Number of Processors** by selecting the **Setup** cell and using the panel on the right (see directions **a.i.3** and **a.i.4** for more info on deciding on the number Fluent of processors).
2. If Fluent was not already open after initialising - click: **User-Defined** > **Execute on demand** > "pre initialise"> **Execute**.
3. Click: **User-Defined** > **Execute on demand** > "windkessel3_on"> **Execute** > **Close**.
4. In **Dynamic Mesh** > **Dynamic Mesh Zones** - double-click on **fsi_fistula - System Coupling** > open the **Solver Options** panel > select **Solution Stabilization** and set the **Scale Factor** to 200 (**volume-based**) > **Create** > **Close**.
5. In **Solution Controls** - ensure that the **Under-Relaxation Factors** for **Pressure** and **Momentum** are 0.1 and 0.23 respectively.
6. In **Run Calculation** - set **Max Iterations/Time Step** to 4.
7. Click: **File** > **Save Project**.
8. To ensure that Fluent is synchronised with workbench for System Coupling click: **File** > **Write** > **Export** > **Case** and select the most recent cas.gz file (should be something like **ICM-49-55-00003.cas.gz**) and click **OK** and **OK** again to overwrite.
9. Leave Fluent window open and return to Workbench window and click **Refresh Project**.

ii. Setting up the System Coupling

1. In Workbench double-click on the **System Coupling system J (ID: SC) Solution** cell to bring up the System Coupling panel
2. In **Setup** > **Data Transfers** > **Force Transfer** > **Data Transfer Control** - set **Ramping** to the **Linear to Minimum Iteration** option.
3. In **Analysis settings** - set **End Time** the current End Time + 0.8 x Ncc (where Ncc is the number of cardiac-cycles to simulate). NB: the first two cardiac cycles after initialisation are not periodically accurate.
4. In **Analysis settings** - set **Step Size** to 0.005 s, **Minimum Iterations** to 6 (this is the number of ramped boundary transfer coupling steps), and the **Maximum Iterations** (Coupling steps) to 30 (the FSI model should converge before this number of coupling steps each time step after the first few time steps).

5. Click: **File** > **Save** to save the Workbench project
6. Click: **Update** to start running the simulation - Check that the **Mapping Summary** in the **Solution Information : System Coupling** panel reads as in **a.ii.4.**

c. Stopping a misbehaving simulation and returning to the last saved state

i. Open the **Progress** panel with **Show Progress** button and click **red x** button.
ii. Click: **Abort**
iii. Click: **File** > **open** and reopen the same project, click **no** so that the current project is not saved. NB - Do not save the project before exiting project to restart from the beginning of the previous simulation.

d. Clearing all saved data from the FSI simulation and returning to a pre-initialised state

i. Right Click on **System Coupling** - **Solution** cell and click **Clear Generated Data**
ii. Click: **OK**
iii. Start initilisation of the model from **A.a.i**.

Adjustments

To change time-step size the Windkessel UDF (windkessel_3E_fistula_3070_5050R_dt005.c) in the user_files folder also needs to be adjusted.

Acknowledgments

W.G. would like to acknowledge funding from the National Research Foundation (NRF) and the South African Research Chair in Computational Mechanics (SARCCM). The views expressed are those of the authors and not necessarily those of the NRF or the SARCCM. The authors would like to thank Prof. Delawir Kahn, Michael Markl, Ernesta Meintjies, Bruce Spottiswood, and Stephen Jermy for their work in producing and processing the MRI data. W.G. would like to thank Andie de Villiers for the useful discussions concerning the content of this work. The authors are indebted to Danie de Kock, Daniel Correia, and the rest of the Qfinsoft team for their ANSYS software support and their guidance with modelling in general.