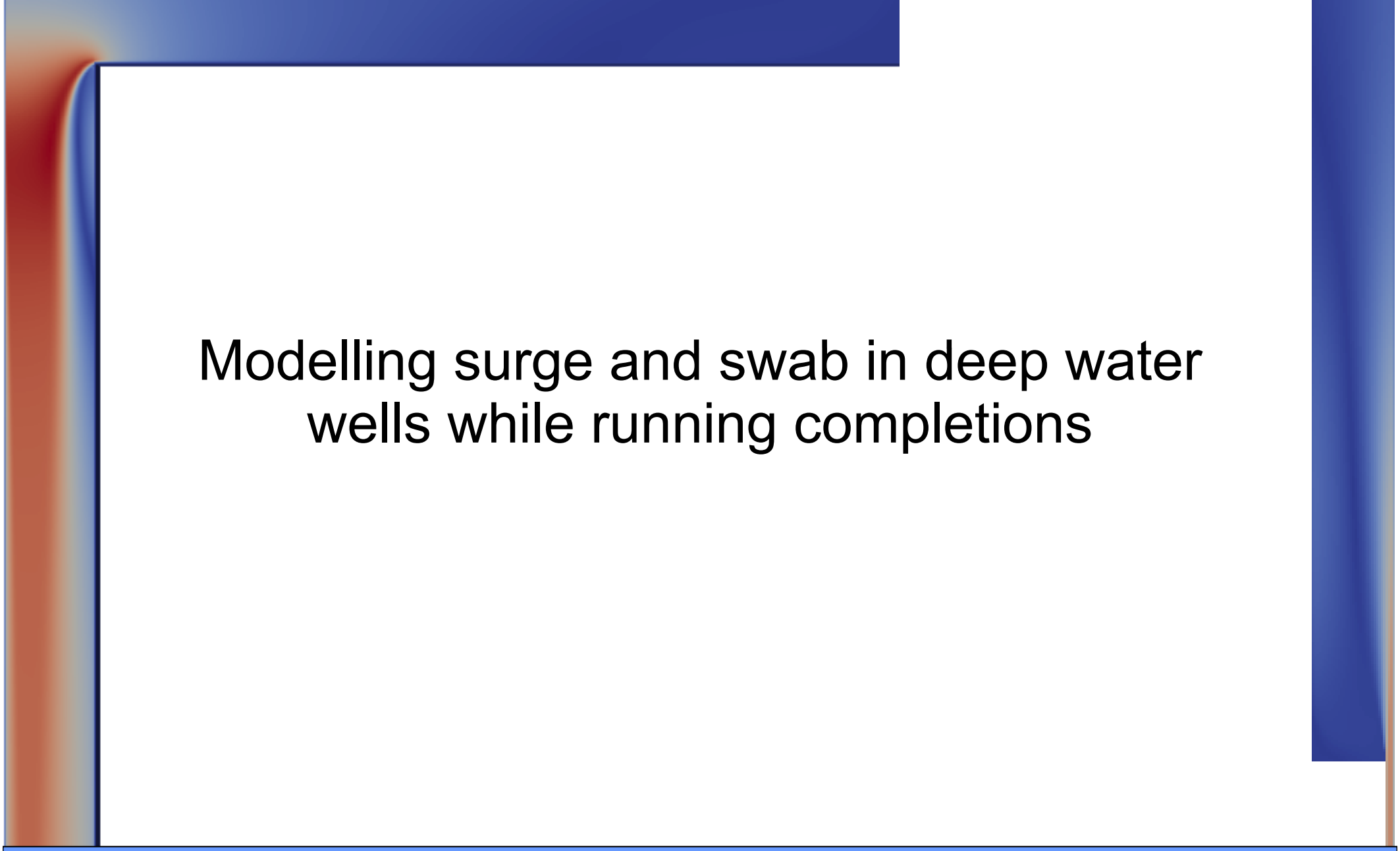
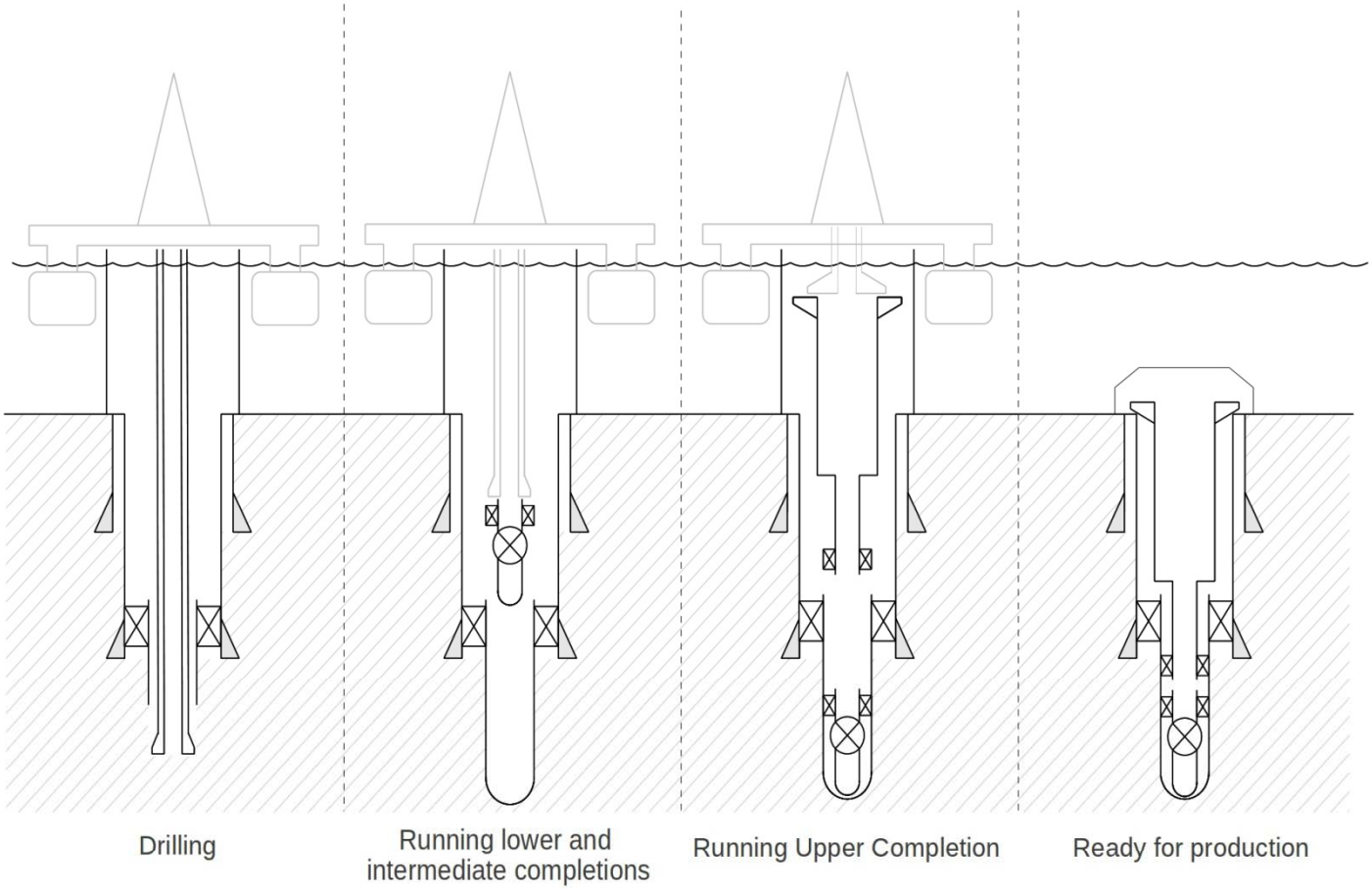


Modelling surge and swab in deep water wells while running completions

Aaron Boase
Mechanical Engineering Undergraduate
Curtin University

Background ~ Terminology & Operational context

The (highly simplified) Drilling and Completion of a well.



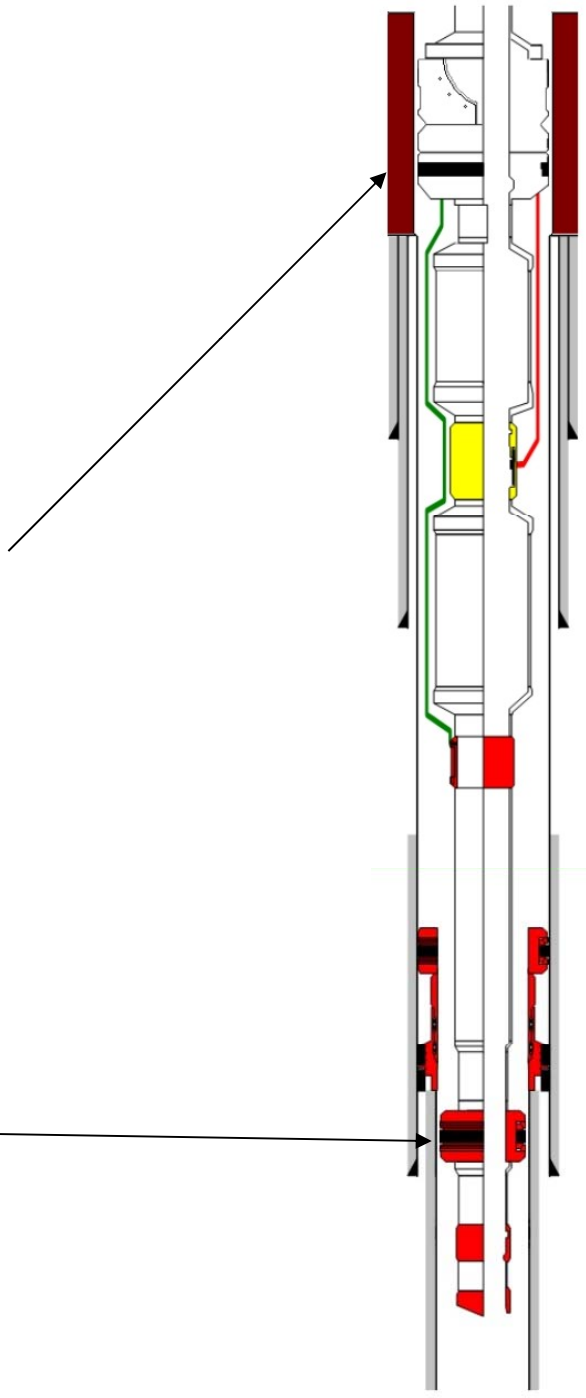
A Completion is a series of components, mainly comprised of production tubing, which are installed in a well before perforation. They are designed to act as the conduit through which hydrocarbons are transported to the top of the well once the well is operational and can include a range of gauges and valves to help control the well.

Background ~ Pressure surge and swab

Fluid flow is caused by the movement of the completion inside the well.

The radial clearance between Tubing Hanger (TH) and the internal diameter of the BOP is known to be as low as 2.5mm

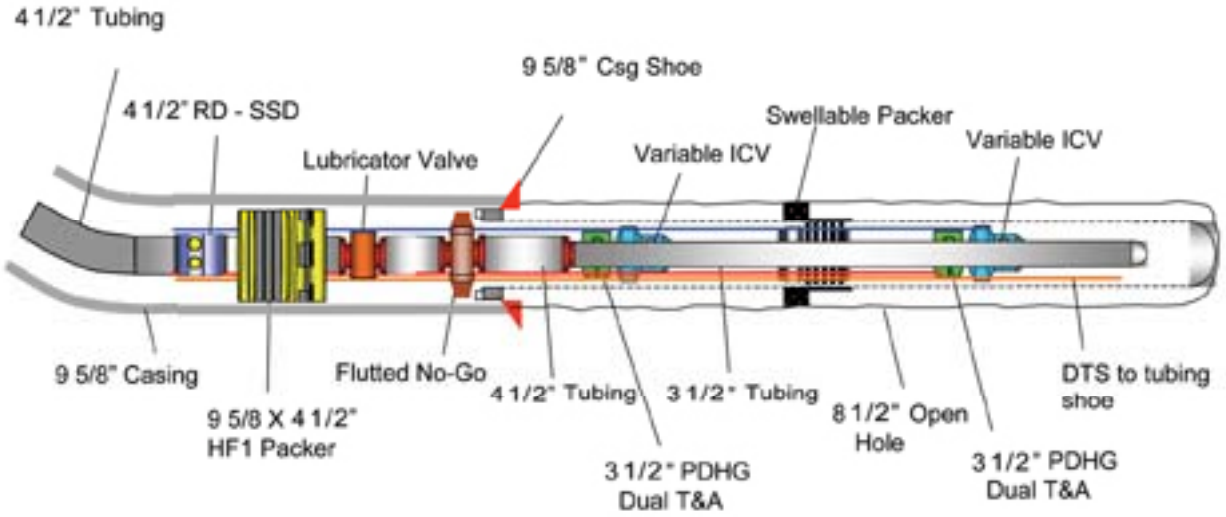
A typical radial clearance between packer and liner is 6mm.



Background ~ Motivation

1) There has been an increase in the use of intervention-less technologies. Many of these devices are activated from the surface by increasing down-hole pressure (Mason et al. 2001, 2).

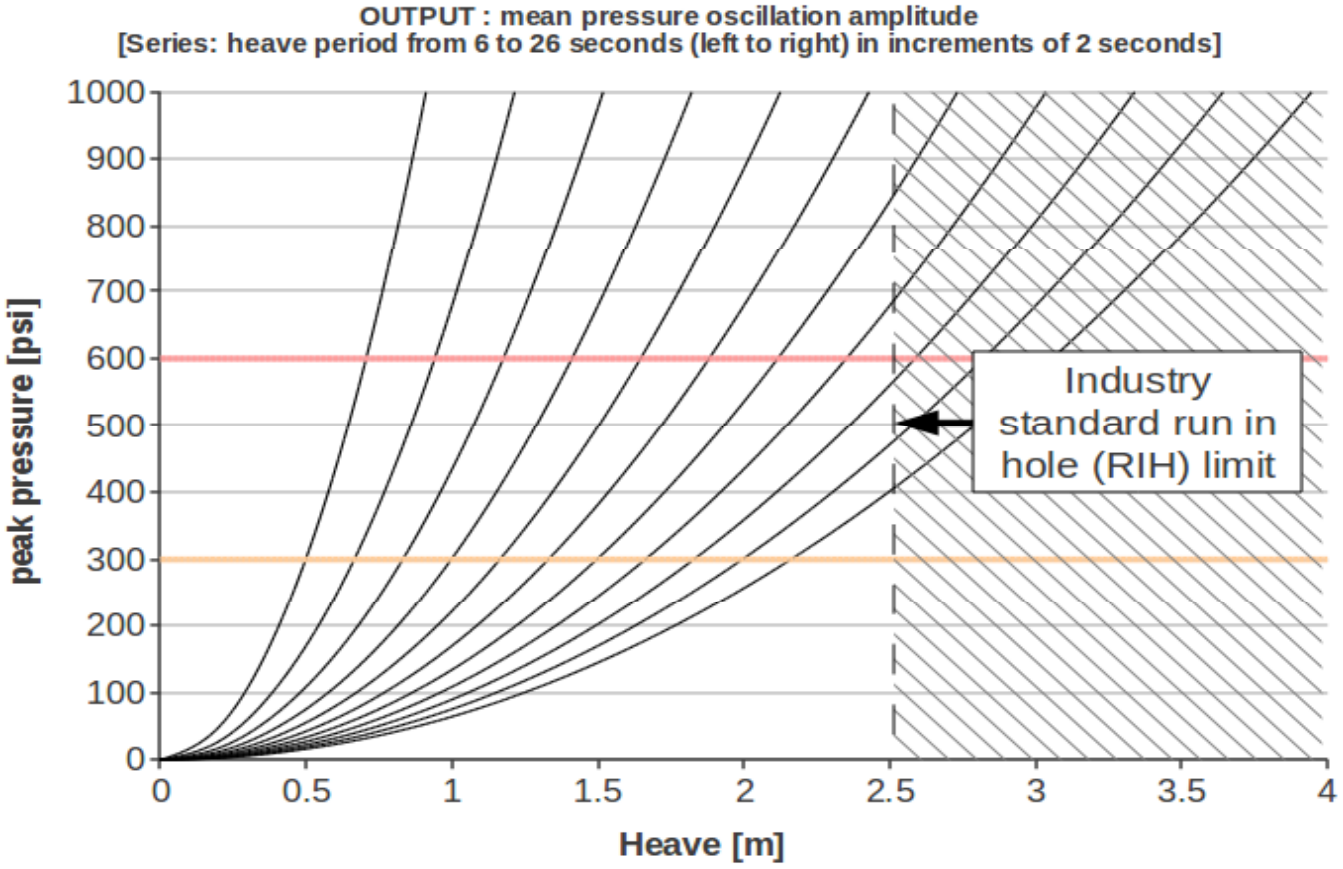
2) Smart completions. A relatively new type of completion which requires installation into a well with sections of exposed formation. Excessive surge or swab may breach the pore and fracture limits of the formation.



Smart-completion (epmag.com)

Approximate analytical model

The flow regime is modelled as though the fluid flows through an equivalent pipe network. The model assumes flow results purely from a pressure differential (Poiseuille flow) and that there is no relative movement of boundaries generating Couette flow.

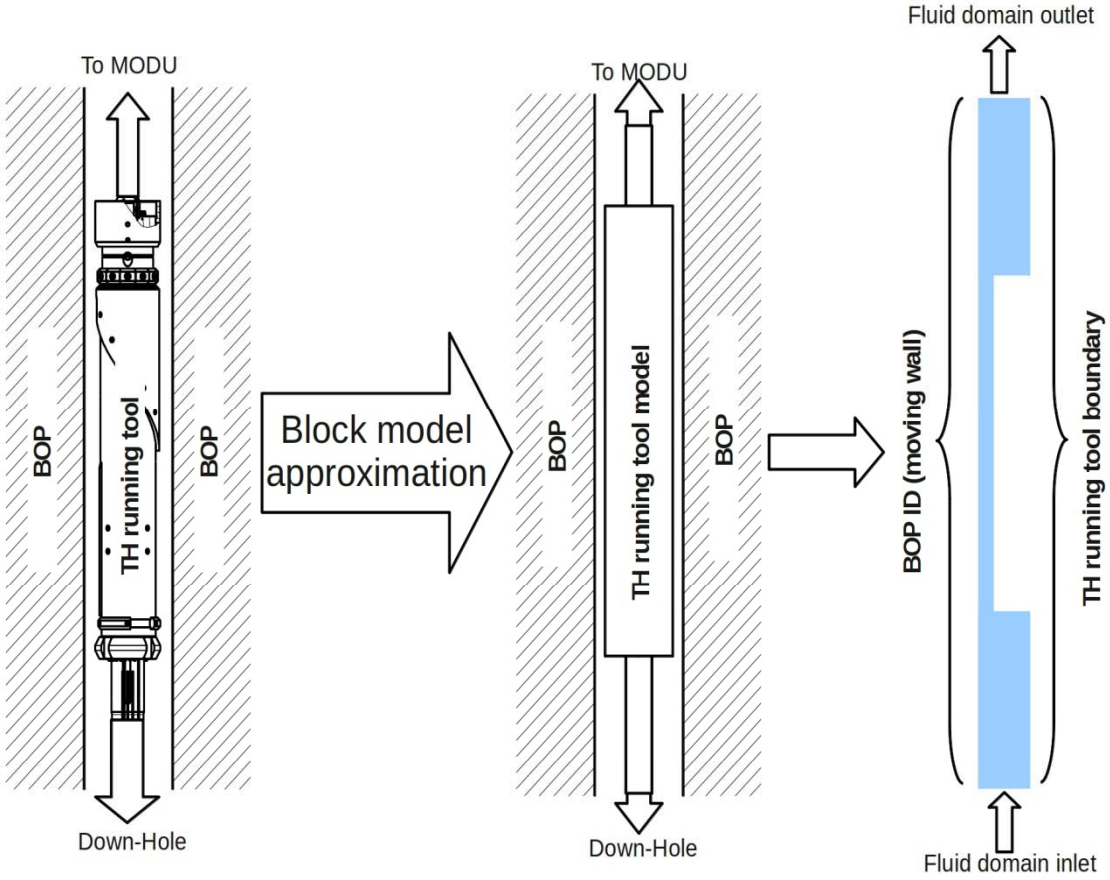


Typical output plot for the an approximate analytical solution

CFD investigation

Aims:

- Investigate whether a useful OpenFOAM model can be constructed.
- Compare CFD results with approximate analytical solution.
- Act as a reference for future modelling efforts.



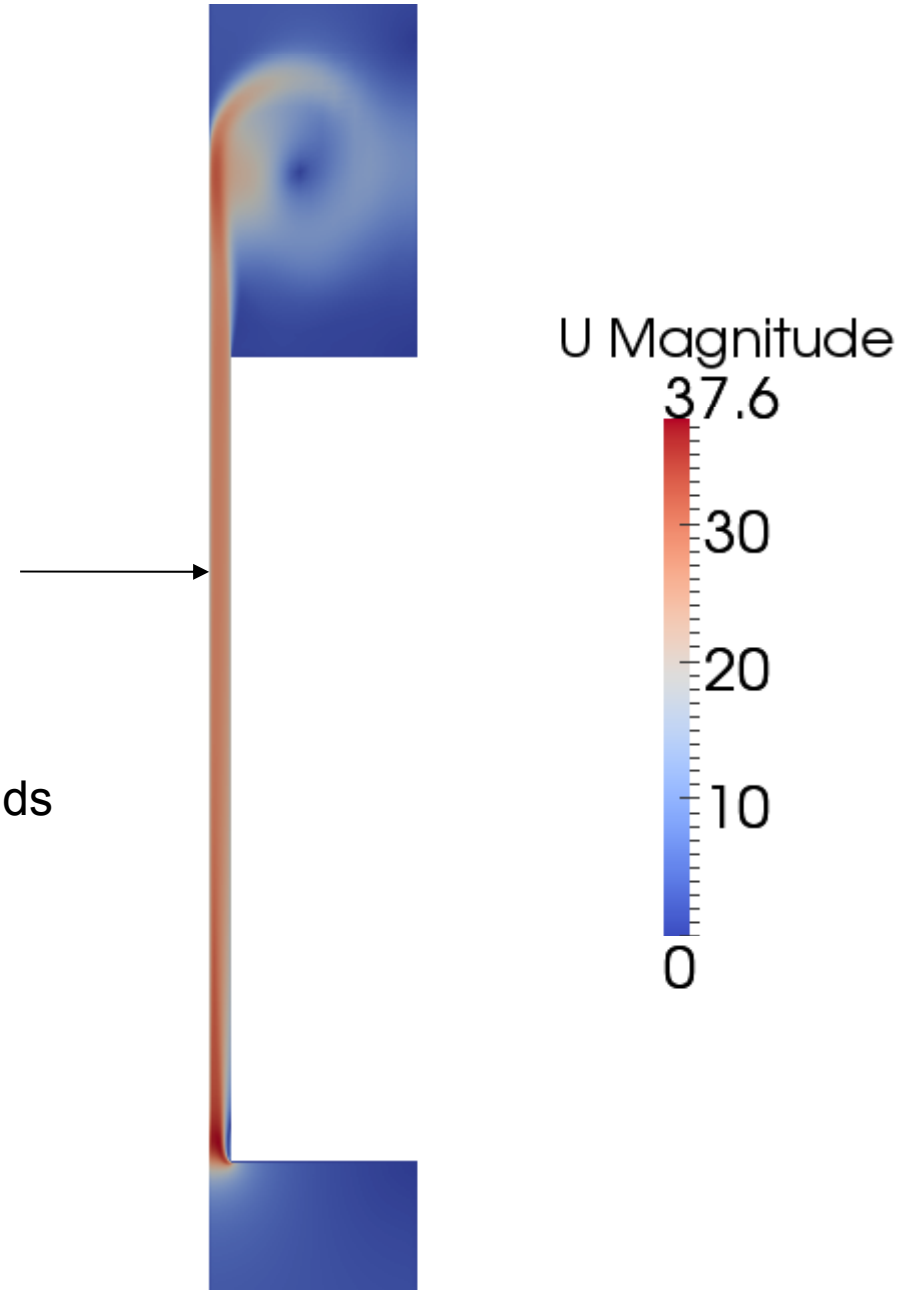
CFD investigation outcomes

The computational expense is excessive for full size and transient flow models

Transient flow models must have mesh small enough so that fluid does not pass through an entire cell in one time step.

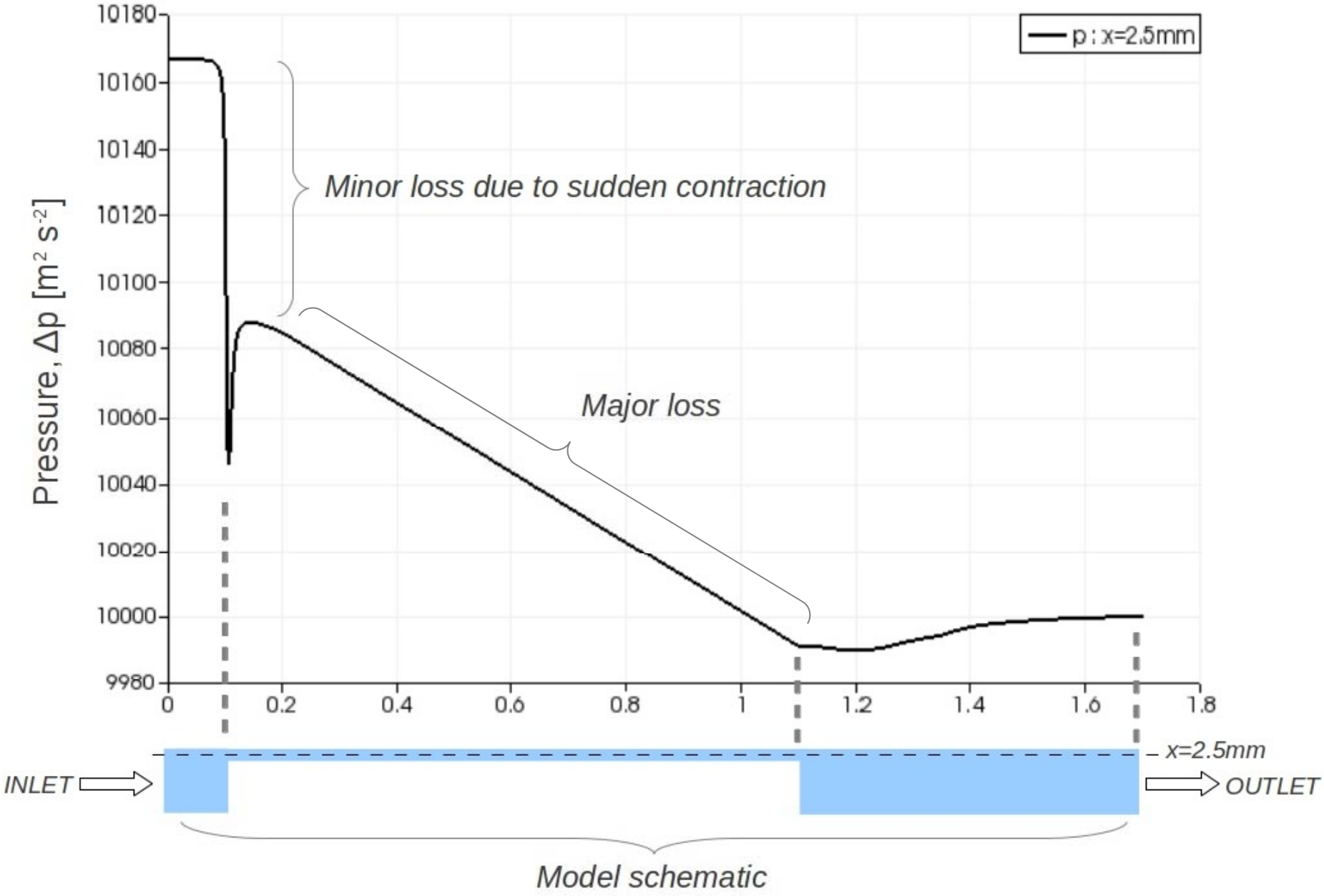
Small scale pseudo-2D model showing transient velocity:

- Simulation run time = 17 hours.
- Time simulated 0.5 seconds.
- Steady state reached at ≈ 0.1 seconds



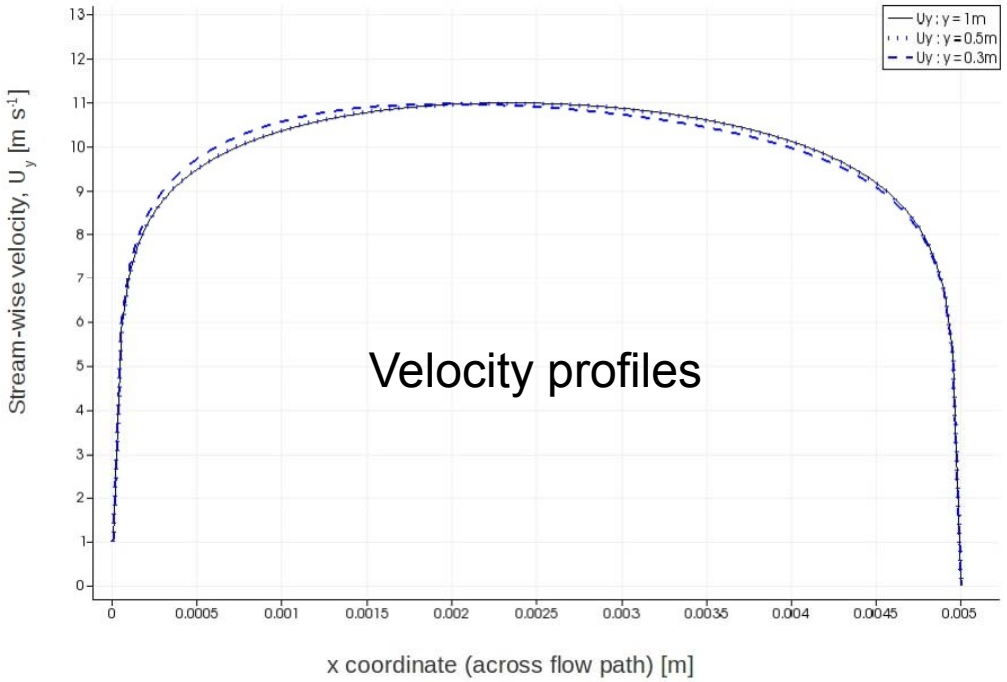
$$Co = \frac{dy\,|U|}{dx} \geq 1 \; ideally \geq 0.5$$

CFD results ~ Steady state model output

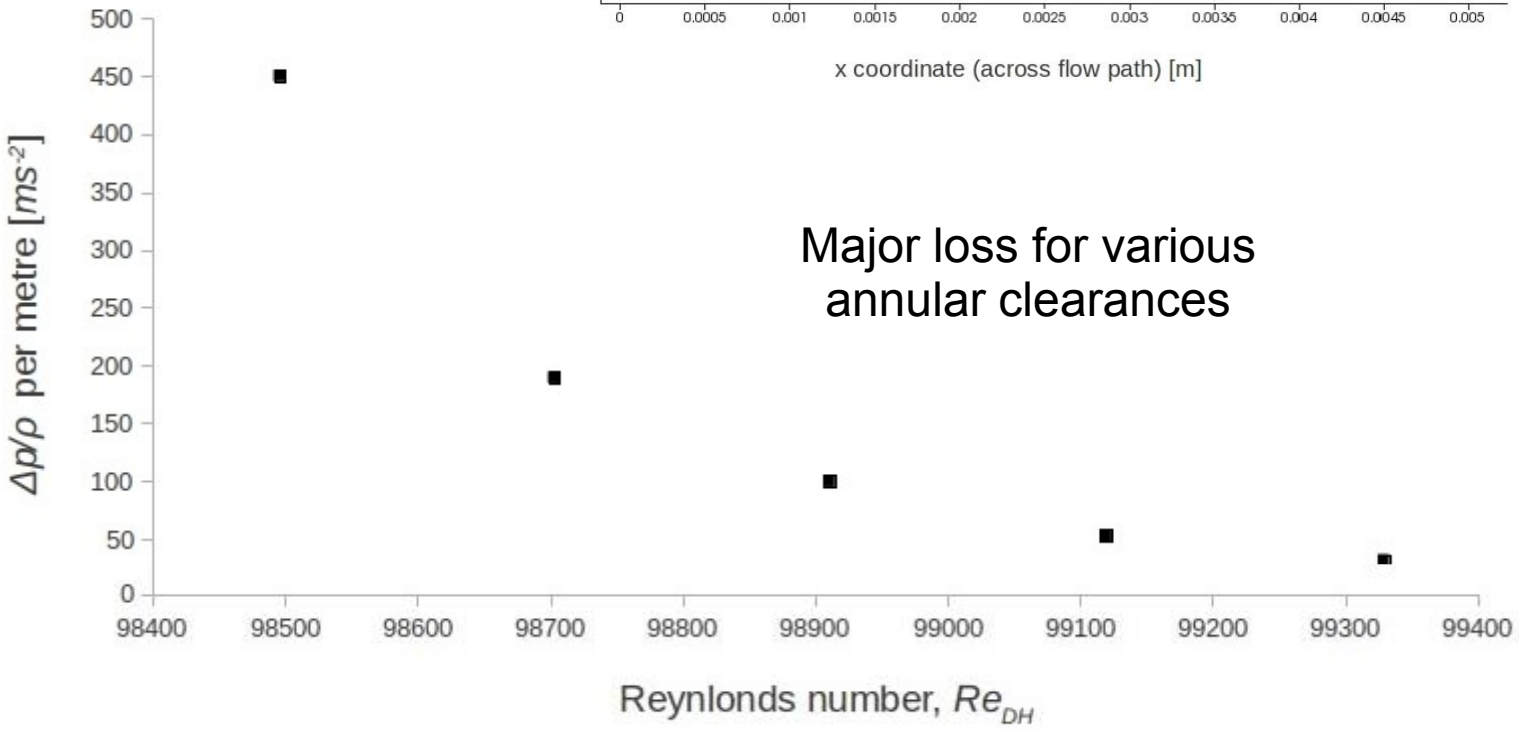


CFD results ~ Steady state model output examples

Graphical representations of flow field



Velocity profiles



Major loss for various annular clearances

Project outcomes

- Found that a CFD model can be constructed but it is only computationally efficient for steady state models of small sections of the completion
- Numerical model predicts higher pressure generation than estimated by the analytical method. The difference is significant at high flow rates (small annular clearances)
- Have created a base for a future CFD modelling.
- Numerical results can be used in a hybrid CFD/analytical system to give more cautious solutions than analytical alone.

Thank you