Vacuum Insulated Tubing Computer Simulation Project

Prepared for:

Mark Godin

PTAC

500 5 Ave SW, Calgary, AB T2P 3L5

Submitted by Shashank Karra

ANSYS Canada Ltd

March 2016

Table of Contents

Introduction	1
CFD & Mechanical Transient	
Thermal Domain and Mesh Setup	2
Boundary Conditions and Material	
Properties	8
Computational Methodology	8
Results and Discussion	10
Conclusions	10
Appendix	11

Vacuum Insulated Tubing Computer Simulation Project

Introduction

In SAGD, ~100% quality steam is injected through an injector string. Bare tubing is used, resulting in heat losses from the wellhead to the heel of the well. Therefore, less than 100% quality steam is delivered to the horizontal section. Vacuum Insulated Tubing (VIT) could replace the bare tubing and significantly reduce heat losses, resulting in higher quality stream and more heat delivered to the formation.

SAGD operators have experimented with VIT in recent years but adoption is uneven. This is due, in part, to the difficulty of precisely assessing the energy efficiency and the economic benefits of VIT vs. bare tubing. Past computer modeling efforts have used basic simulation packages and simplistic geometries. This simulation in this project used sophisticated CFD and Mechanical Transient Thermal modeling software by ANSYS.

The objective of this project was to perform a comparative analysis between VIT and bare string tubing configurations. The simulations provided a comparison of heat losses in both the tubing configurations.

The simulations in Tasks 1 to 4 were steady state fluid flow and heat transfer simulations. These simulations modeled advection of steam through the tubing, three modes of heat loss namely conduction, convection and radiation, and condensation of steam into water in the tubing. The simulations in these four tasks were performed using "ANSYS CFX CFD" software. Task 6 was cancelled to increase the scope and number of simulations in Task 3.

The simulations in Task 5 and 7 were transient thermal conduction only simulations. These simulations did not include any fluid flow and modeled just the transfer of heat due to conduction. The simulations in task 5 and 7 were performed using "ANSYS Mechanical Transient Thermal" software.

Simulation Work Summary

This report summarizes the CFD and transient thermal simulation work spread over six different tasks. A total of 12 simulations were carried out in all the tasks. The report covers details about the CFD and mechanical transient modeling methodology, simulated domain, computational mesh and summary of results and conclusions from the work. The domains created in tasks 1 to 4 were 3D. The domain created in tasks 5 & 7 were 2D.

1) The four steady state CFD simulations in Tasks 1 and 2 involved studying the geometric sensitivity of both bare and VIT tubing using steady state CFD simulations. Both bare and VIT tubing were modeled in concentric and eccentric positions (with respect to the casing).
2) The four steady state CFD simulations in Task 3 involved studying the effect of pseudo-transient heating of the rock/overburden on heat losses in the tubing. This was carried out in pseudo-transient manner by changing the distance between the casing and the boundary location of formation temperature.
3) The two steady state CFD simulations in Task 4 involved studying the effect of varying K value of VIT tubing on heat losses.
4) The two transient thermal mechanical simulations in Tasks 5 and 7 involved studying the effect of transient heating of rock/overburden to have a comparison of heat loss vs. time for both bare and VIT tubing.

The results of each of the above tasks was provided to PTAC and all participating members in the form of PowerPoint reports.

CFD & Mechanical Transient Thermal Domain and Mesh Setup

A typically CFD and Mechanical transient thermal simulation setup involves creation of a fluid and solid geometry respectively representing the actual domain dimensions. Meshing the fluid and solid volume, setting up the simulation, running it to convergence, and finally analyzing the results.

In this project, ANSYS Design Modeler was used to create the simulation domain. ANSYS Meshing was used to generate a mesh on computational domain, and ANSYS CFX and ANSYS Mechanical transient thermal were used to run simulations and post process results.

Geometry Creation

The CFD simulation domain created in Tasks 1 and 2 is shown in Figures 1, 2 and 3. The domain for bare tubing in both concentric and eccentric positions is shown in Figure 1. The domain for VIT tubing in both concentric and eccentric positions is shown in Figure 2. Rock temperature boundary condition was applied at 0.01 inch radial distance away from the casing. The dimensions of the domain simulated in Tasks 1 and 2 are provided in Table 1 in Appendix A.

The contact area between the tubing and casing in eccentric position was measured by zEroCor Tubulars Inc. in their lab. For bare tubing, contact area was measured to be 7.1% of the circumference of the ID of the casing and for VIT tubing contact area was measured to be 7.9% of the circumference of the ID of the casing. These values were used in building the eccentric geometries in all six tasks executed in this project.

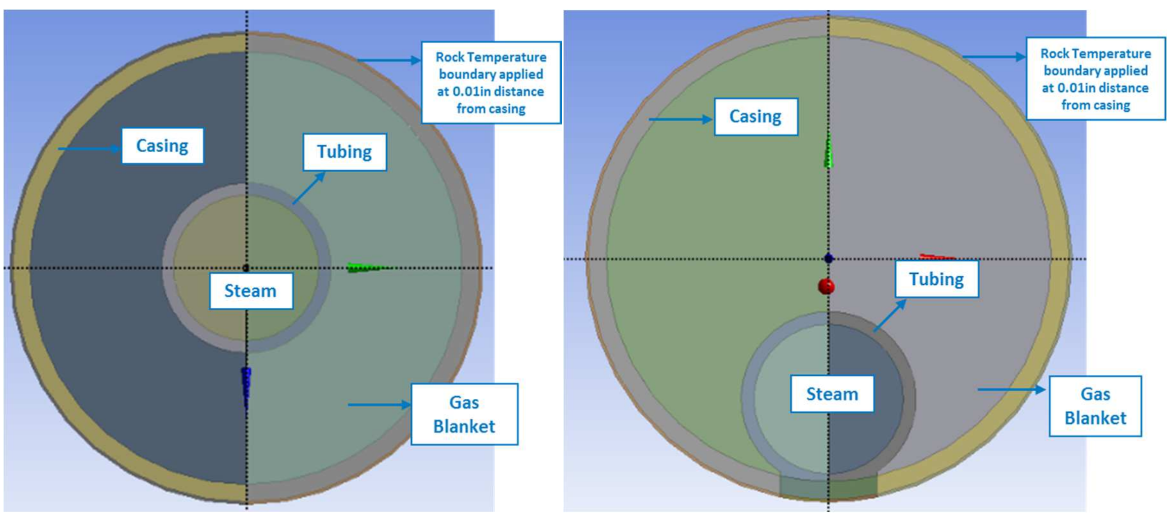

Figure 1: Cross Section of Bare Tubing in Concentric Position (left); Cross section of Bare Tubing in Eccentric Position

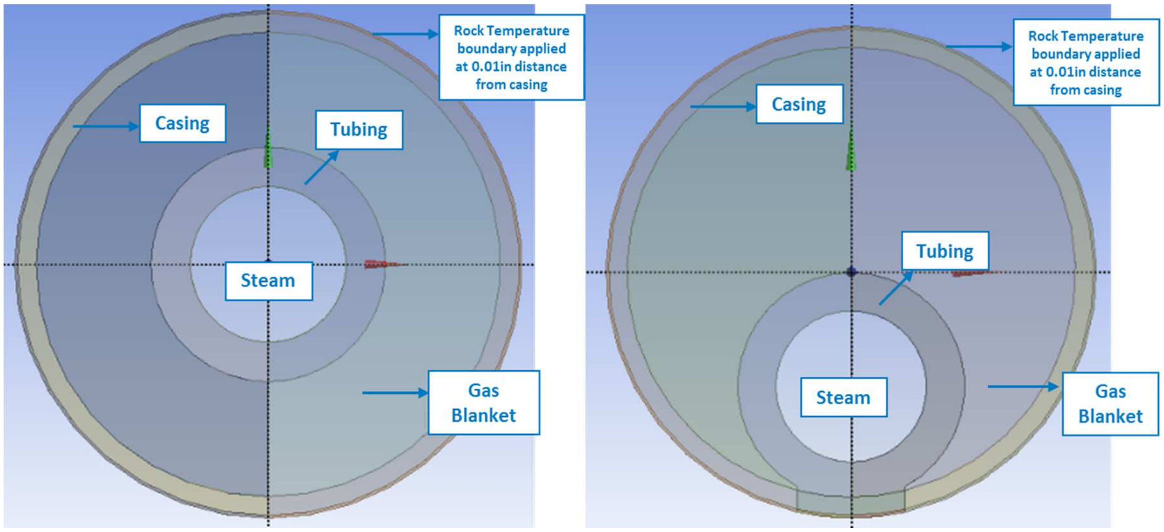

Figure 2: Cross Section of VIT Tubing in Concentric Position (left); Cross Section of VIT Tubing in Eccentric Position

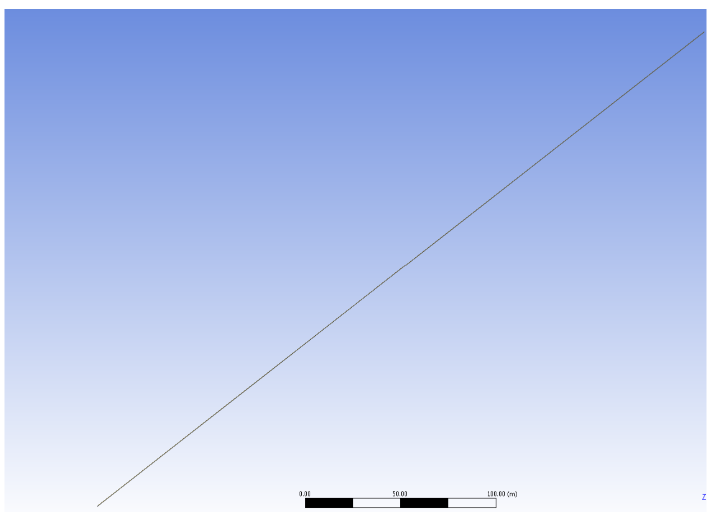

Figure 3: 450m Long Simulation Domain containing Tubing, Gas Blanket, Casing and Rock for both Bare and VIT Tubing Simulations

The simulation domain created for Tasks 3 and 4 is shown in Figures 4 and 5 respectively. Figure 4 shows domain for bare and VIT in eccentric position with rock temperature applied at 10m radial distance away from the casing. Similarly Figure 5 shows domain for bare and VIT in eccentric position with rock temperature applied at 50m radial distance away from the casing. The dimensions of the domain simulated in these two tasks are provided in Table 1 in the Appendix A.

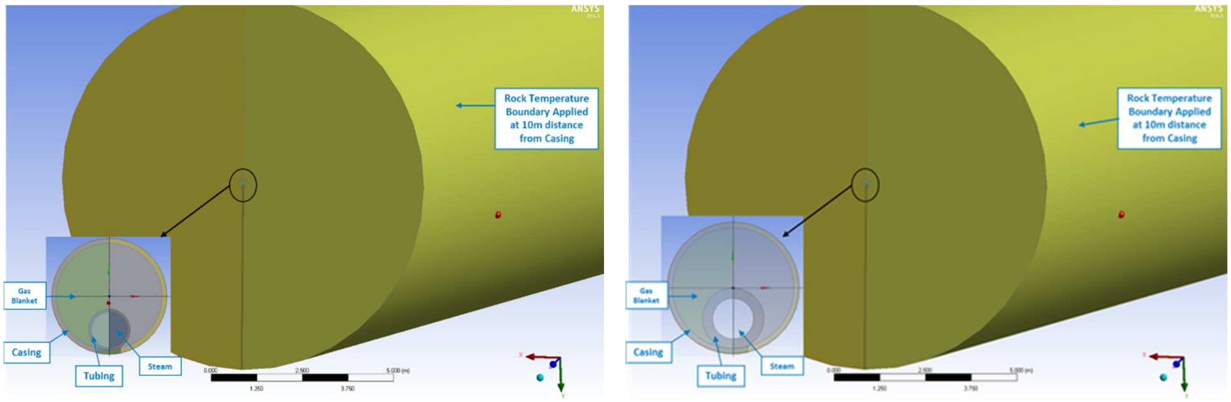

Figure 4: Simulation Domain of Bare Tubing with Formation Temperature applied at 10m Distance from Casing (left); Simulation Domain of VIT tubing with Formation Temperature applied at 10m Distance from Casing - Task 3

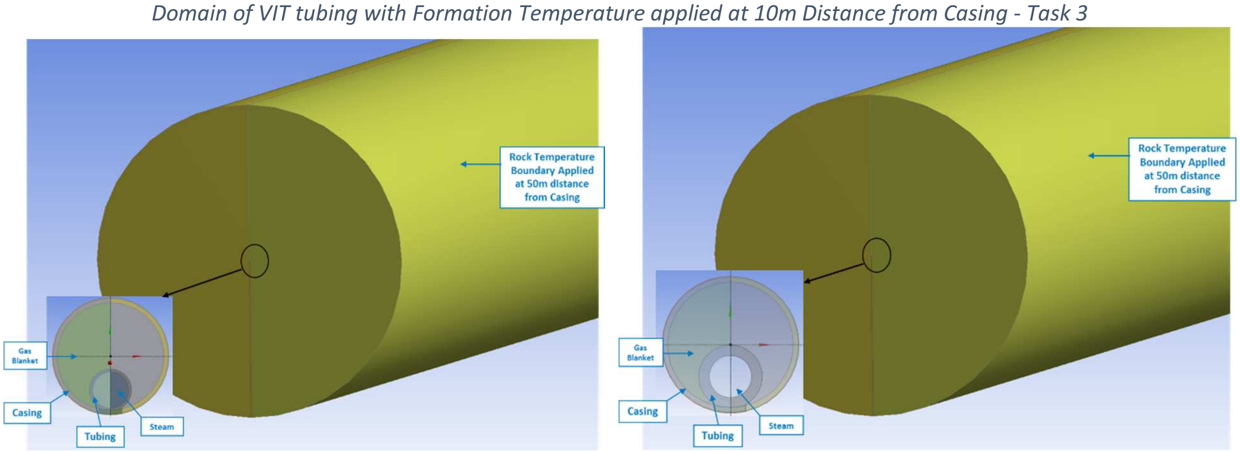

Figure 5: Simulation Domain of Bare Tubing with Formation Temperature applied at 50m Distance from Casing (left); Simulation Domain of VIT tubing with Formation Temperature applied at 50m Distance from Casing - Task 4

As mentioned earlier in this report Task 5 and 7 of this study were 2D mechanical transient thermal simulations. The simulation domain created for Tasks 5 and 7 is shown in Figures 6 and 7. These simulations were run in ANSYS transient mechanical solver as mentioned in earlier section. A 2D planar strain domain as created for these simulation. For dimensions of the domain used in these two tasks please refer to Table 1 in the Appendix A.

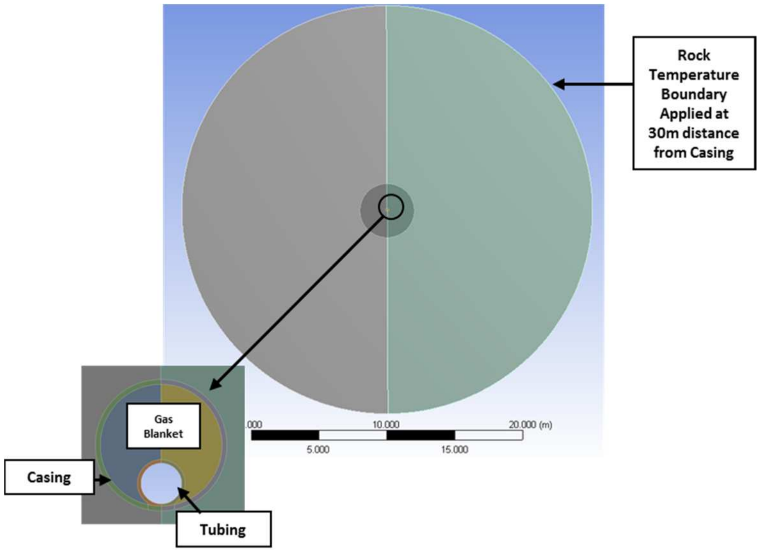

Figure 6: Simulation Domain of 2D Mechanical Transient Simulation. The Figure shows Bare Tubing with Formation Temperature Applied at 30m Distance from Casing

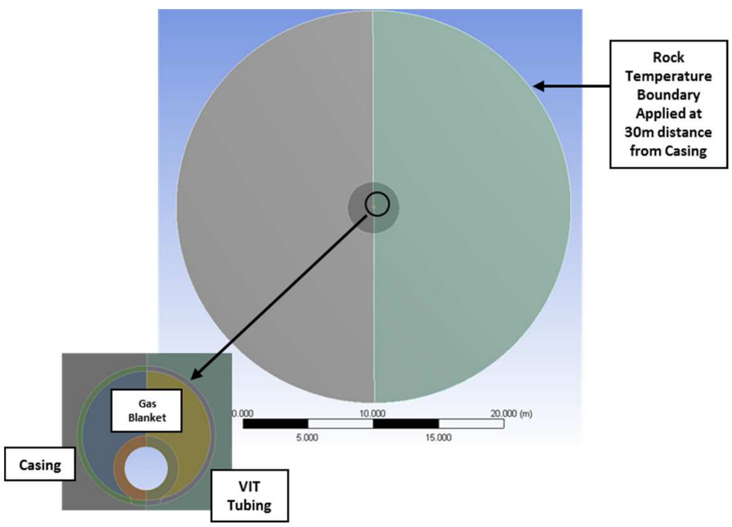

Figure 7: Simulation Domain of 2D Mechanical Transient Simulation. The Figure shows VIT Tubing with Formation Temperature Applied at 30m Distance from Casing

CFD Mesh Creation

The general CFD mesh created in Tasks 1 and 2 is shown in Figures 8 and 9. A hexahedral mesh was generated for all the domains namely tubing, gasblanket, casing and rock. Boundary layers were created on tubing wall to capture flow frictional losses due to non-slipping walls. The tubing wall surface was assumed to be smooth. Mesh was suitably refined near casing and tubing contact area for eccentric domain cases.

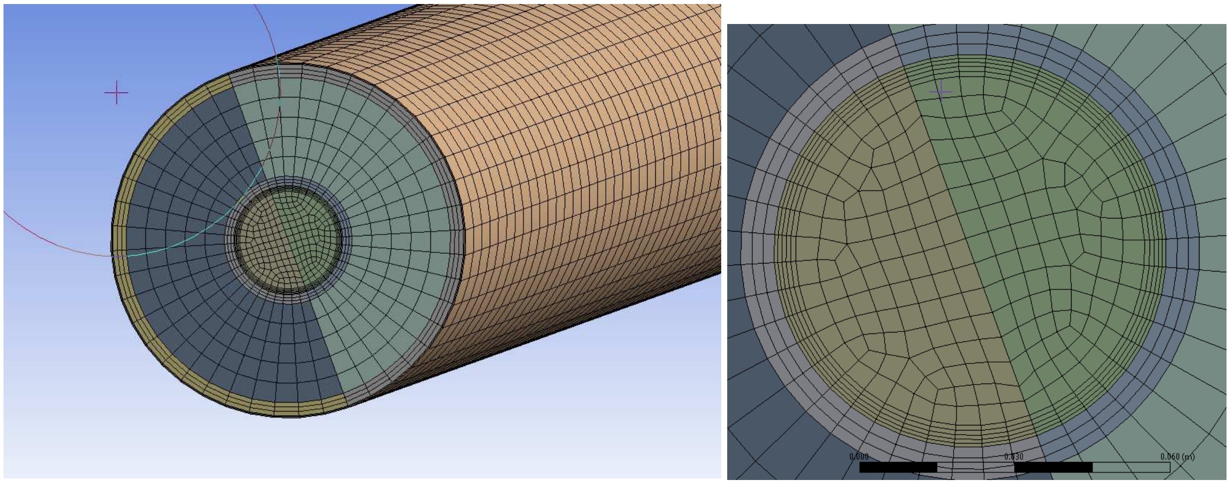

Figure 8: The Figure shows Computational Mesh for Concentric Bare Tubing

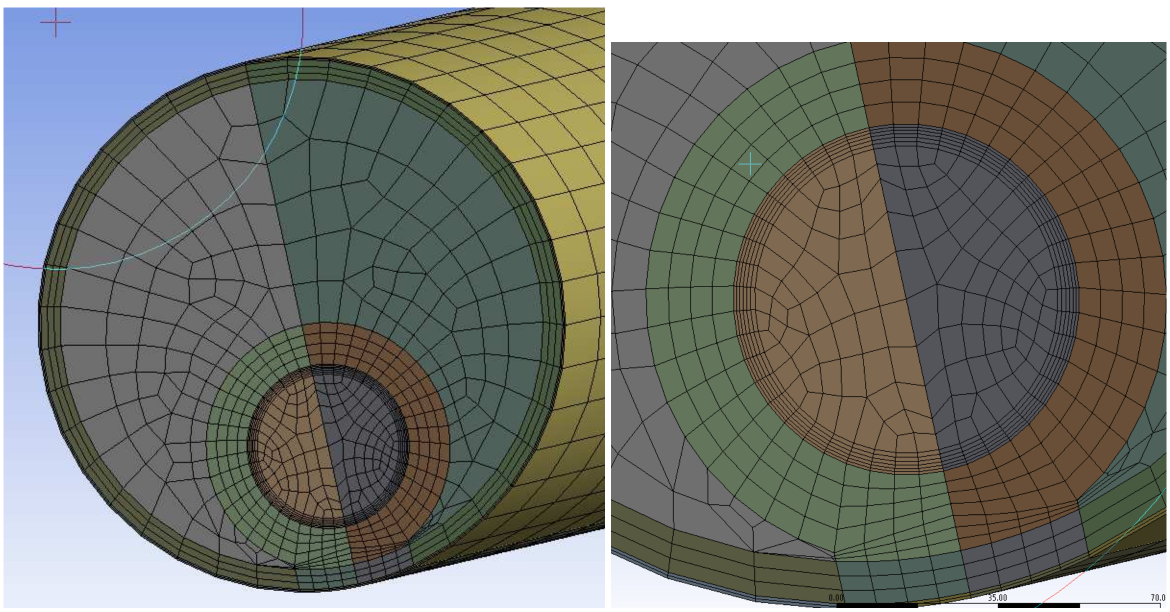

Figure 9: The Figure shows Computational Mesh for Eccentric VIT Tubing. Similar mesh was generated for Eccentric Bare Tubing Domain

The CFD mesh created in Tasks 3 and 4 is shown in Figure 10. Similar to the meshes generated in the first two tasks a hexahedral mesh was generated for all the domains namely tubing, gasblanket, casing and rock. Boundary layers were created on tubing wall to capture flow frictional losses due to non-slipping walls. The tubing wall surface was assumed to be smooth. 10 and 50 m rock domain was modeled in Tasks 3 and 4 respectively. Mesh was suitably refined near casing and tubing contact area for eccentric domain cases.

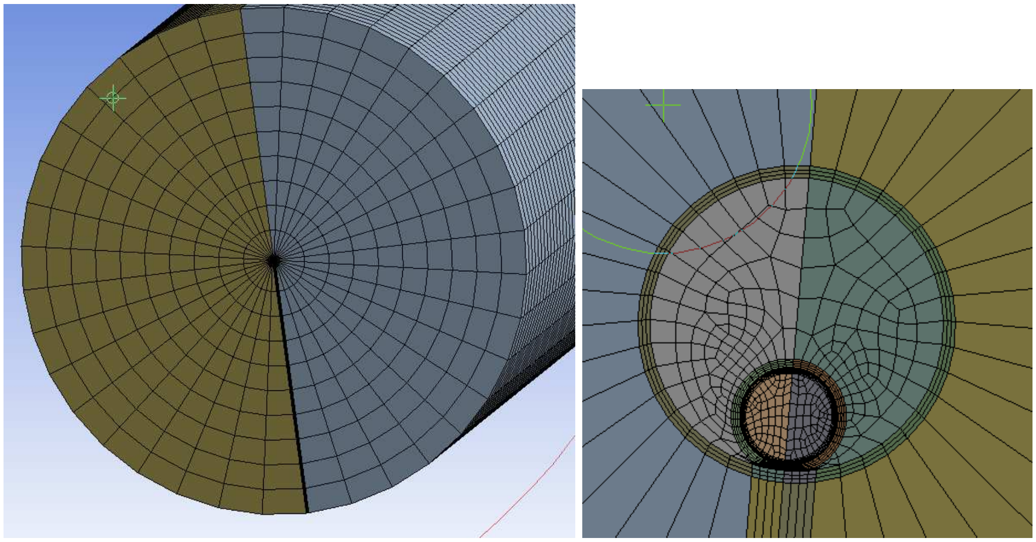

Figure 10: The Figure shows Computational Mesh for Eccentric VIT Tubing with 10m of Rock Modeled. Similar meshes were generated for 50m case as well

The 2D transient structural thermal mechanics mesh created in Tasks 5 and 7 is shown in Figure 11. A quadrilateral mesh was generated for all the domains namely tubing, gasblanket, casing and rock.

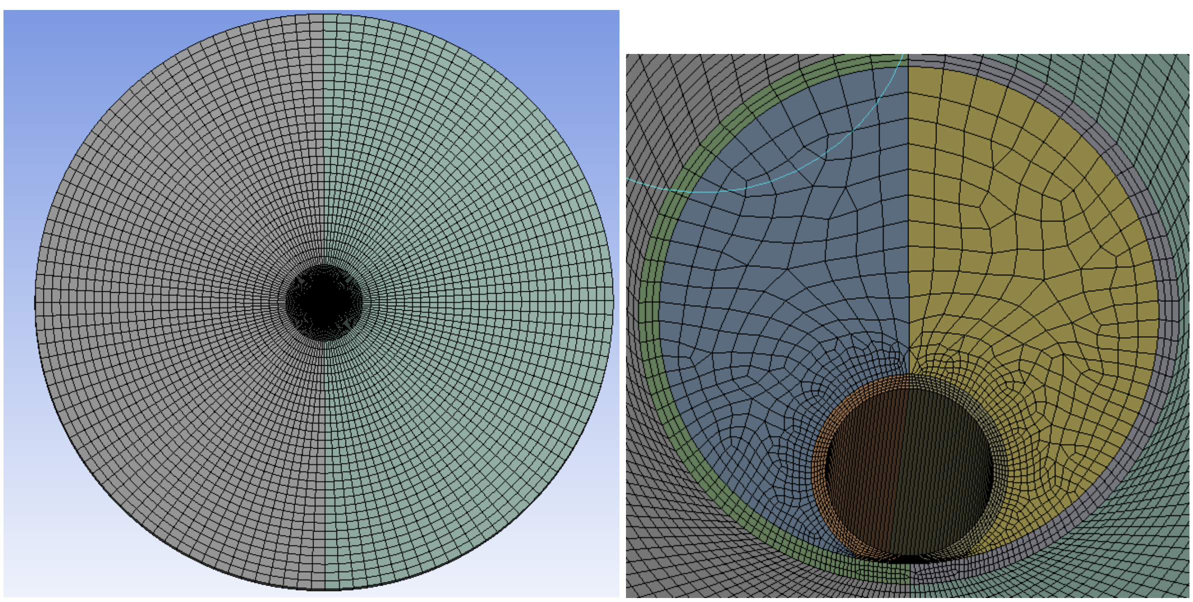

Figure 11: The Figure shows Computational Mesh for 2D Bare Tubing with 30m of Rock modeled. Similar Mesh was generated for VIT Tubing as well

Boundary Conditions and Material Properties

Tasks 1 and 2

Four CFD simulations were executed in Tasks 1 and 2. These simulations involved modeling of steam injection through the tubing and heat loss through conduction, convection and radiation. The simulations also included modeling of condensation of steam into water in the tubing. The boundary conditions used in Tasks 1 and 2 in this project are given in Table 3 in Appendix B. The material properties used in Tasks 1 and 2 are given in Table 4 in Appendix B.

The key point to note here is that in these simulations 0.01inch thickness of rock domain was modeled.

Tasks 3 and 4

Six CFD simulations were executed in Tasks 3 and 4. These simulations involved modeling of steam injection through the tubing and three of heat loss through conduction, convection and radiation. The simulations also included modeling condensation of steam into water in the tubing. The boundary conditions used in Task 3 and 4 in this project are given in Table 3 in Appendix B. The material properties used in Tasks 3 and 4 are given in Table 4 in Appendix B

The key point to note here is that in these simulations 10m and 50m thickness of rock domain was modeled.

Tasks 5, 7

Two mechanical transient thermal simulations were executed in Task 5 and 7. These simulations involved modeling of transient heat conduction/losses through the tubing into the surrounding rock domain. These simulation did not include any steam flow modeling. The boundary conditions used in Task 5 and 7 in this project are given in Table 5 in Appendix B. The material properties used in Tasks 5 and 7 are given Table 6 in Appendix B.

The key point to note here is that in these simulations 30m thickness of rock domain was modeled.

Computational Methodology

Tasks 1, 2, 3 & 4

ANSYS CFX, the CFD solver used for Tasks 1, 2, 3 & 4, uses a finite volume based discretization of the conservation equations of mass, momentum and energy. The simulation in these tasks involve multiphase flow of steam and water through the tubing. Steam is modeled as a primary continuous phase in the domain and water is modeled as a dispersed droplet phase. The conservation equations for mass, momentum and energy for phase α in multiphase flow are written as:

$$\frac{\partial}{\partial t}(r_\alpha \rho_\alpha) + \nabla \bullet (r_\alpha \rho_\alpha U_\alpha) = \sum_{\beta=1}^{N_P} \Gamma_{\alpha\beta}$$

Where "$\Gamma_{\alpha\beta}$" is the interphase mass flow rate per unit volume between phase α and phase β

$$\frac{\partial}{\partial t}(r_\alpha \rho_\alpha U_\alpha) + \nabla \bullet (r_\alpha (\rho_\alpha U_\alpha \otimes U_\alpha)) = -r_\alpha \nabla p_\alpha + \nabla \bullet (r_\alpha \rho_\alpha (\nabla U_\alpha + (\nabla U_\alpha)^T)) + \sum_{\beta=1}^{N_P} (\Gamma^+_{\alpha\beta} U_\beta - \Gamma^+_{\beta\alpha} U_\alpha)$$

Where "$\Gamma^+_{\alpha\beta} U_\beta - \Gamma^+_{\beta\alpha} U_\alpha$" represents the momentum transfer induced by interphase mass transfer

As mentioned earlier, the current simulation accounts for condensation of steam into water. To account for steam condensation in the tubing, the Thermal Phase Change model in ANSYS-CFX was used. The wall condensation heat and mass transfer rates were determined by a total heat balance in the wall-adjacent cell assuming thermal equilibrium between the phases. This approach utilised the near-wall vapour temperature along with local saturation and wall temperatures, which allowed the usage of the thermal phase change model for wall driven condensation. While this approach does not account for the nucleation physics happening near the wall or detailed liquid wall film effects, it provides for a good approximation for the complex process. This approach is best suited for a comparison study, between different design configurations.

ANSYS-CFX considers the interphase heat transfer due to thermal non-equilibrium across phase interfaces. The phasic total energy equation is written as

$$\frac{\partial}{\partial t}(r_\alpha \rho_\alpha h_{\alpha,tot}) - r_\alpha \frac{\partial P}{\partial t} + \nabla \bullet (r_\alpha \rho_\alpha U_\alpha h_{\alpha,tot}) = \nabla \bullet (r_\alpha \lambda_\alpha \nabla T_\alpha) + r_\alpha \nabla \bullet (U_\alpha \bullet \tau_\alpha) + \sum_{\beta=1}^{N_P} (\Gamma^+_{\alpha\beta} h_{\beta s,tot} - \Gamma^+_{\beta\alpha} h_{\alpha s,tot})$$

Where "$\Gamma^+_{\alpha\beta} h_{\beta s,tot} - \Gamma^+_{\beta\alpha} h_{\alpha s,tot}$" represents the heat transfer induced by interphase mass transfer

For radiation heat transfer, the Monte Carlo model was used in ANSYS-CFX. The Monte Carlo model assumes that the radiation intensity is proportional to the differential angular flux of the photons and the radiation field as a photon gas

Tasks 5&7

ANSYS Mechanical, the FEA solver used for Tasks 5 & 7, uses a finite element based discretization of the conservation equation of energy. ANSYS Mechanical solves the thermal energy equation of the following form

$$\rho c \left[\frac{\partial T}{\partial t} + v_x \frac{\partial T}{\partial t} + v_y \frac{\partial T}{\partial t} + v_z \frac{\partial T}{\partial t}\right] = \ddot{q} + \frac{\partial}{\partial x}\left(K_x \frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(K_y \frac{\partial T}{\partial y}\right) + \frac{\partial}{\partial z}\left(K_z \frac{\partial T}{\partial z}\right)$$

Where T = temperature (=T(x, y, z, t), ρ = density, c = specific heat, t = time and $\ddot{q}$ = heat generation per unit volume

Results and Discussion

The four CFD simulation in Tasks 1 and 2 included bare and VIT tubing in concentric and eccentric configurations with respect to the casing. The simulations results showed that both in an idealistic concentric position and in a realistic eccentric position the bare tubing had more heat losses compared to the VIT tubing. Bare tubing in eccentric position had the highest heat loss. This is because in eccentric configuration the tubing is in contact with the casing and heat conduction occurs through the contact area resulting in heat loss and reduction in injected steam quality. On the other hand for VIT tubing heat losses were minimal in the eccentric position due to the low conductivity material of VIT tubing.

In Task 3 the rock temperature boundary condition was applied at 10m and 50m distance respectively to investigate the effect of surrounding rock/overburden heating up on heat loss from the injection tubing in a pseudo transient manner. However the distances of 10m and 50m were too far away and the results were inconclusive from these simulations. This resulted in modification of simulation methodology which was executed in Tasks 5 and 7 through transient thermal conduction only simulations.

Task 4 involved studying the effect of VIT K-value on heat loss through injection tubing. It was seen that decrease in K-value of VIT tubing results in increased heat loss.

The two mechanical transient heat conduction simulations in Tasks 5 and 7 included bare and VIT tubing in eccentric positions. These transient simulations were run for a total physical run time of five years. Temperature and heat loss values at different circumferential positions and radial distance (from the casing) along the rock were recorded with respect to time. The results from these simulation showed that the heat loss from bare tubing was higher than VIT tubing over the five year period.

Conclusions

All participating producer companies gained valuable insights from the simulations over the performance of VIT and bare tubing. The simulations showed the importance of considering eccentric placement of tubing inside the casing and its effect on heat loss. Valuable data from the simulations showing the temperature and heat loss over five period of time from both bare and VIT tubing was provided to all participating members of this project.

As next steps all the participating producer companies plan to calculate the economic impact of from heat loss in the tubing. Future simulation work could involve studying the effect of various parameters such as tubing diameter, injection steam temperature, rock heterogeneity etc.

Appendix A

Tubing and Casing Domain dimensions simulated in Task 1, 2, 3 and 4

	TVD (m)	Inclination (degrees)	OD (in)	ID (in)
Bare Tubing	450	10	3.5	2.99
VIT Tubing	450	10	4.5	2.99
Casing	450	10	9.625	8.9

Table 1: Domain Dimensions for Casing and Tubing used in CFD simulations in Tasks 1 to 4

Task 5 and 7 were 2D mechanical transient thermal simulations. The dimensions for casing and tubing used in these simulations are given below

	OD (in)	ID (in)
Bare Tubing	3.5	2.99
VIT Tubing	4.5	2.99
Casing	9.625	8.9

Table 2: Domain Dimensions for Casing and Tubing used in Transient Thermal Simulations in Tasks 5 and 7

Appendix B

Boundary Conditions used for CFD simulation in Tasks 1, 2, 3 and 4 are given below

Steam Injection Flow rate (m3/day)	250
Steam Injection pressure (Mpa)	3
Steam Injection Temperature (C)	434
Steam Quality	0.957
Rock Temperature (C)	10

Table 3: Boundary Conditions used in CFD simulations in Tasks 1 to 4

Material properties used in CFD simulations in Task 1, 2 and 3 are given below

	Bare Tubing (Steel)	**VIT Tubing**	**Casing (Steel)**	**Gas Blanket (Nitrogen)**	**Athabasca Oil Sands ***
Density (kg/m^3)	7850	3000	7850	808.4	2073
Thermal Conductivity (W/mC)	44	0.01	44	0.05	2.5

Table 4: Material Properties used in CFD simulations in Tasks 1 to 4

For task 4 (which involved CFD simulations comparing different K-values of VIT tubing) K-values of 0.006 and 0.1 were used.

Boundary Conditions used in mechanical thermal transient simulation in Tasks 5 and 7 are given below

Steam Injection Temperature (C)	250
Rock Temperature (C)	10

Table 5: Boundary Conditions used in Transient Thermal Simulations in Tasks 5 and 7

	Bare Tubing (Steel)	**VIT Tubing**	**Casing (Steel)**	**Gas Blanket (Nitrogen)**	**Athabasca Oil Sands ***
Density (kg/m^3)	7850	3000	7850	808.4	2073
Thermal Conductivity (W/mC)	44	0.01	44	0.05	2.009
Specific Heat Capacity (J/kgk)	434	750	434	1040	900

Table 6: Material Properties used in Transient Thermal Simulation in Tasks 5 and 7