NURETH14-152

THERMOSYPHON CIRCULATION INSIDE VERTICAL CONDENSER BUNDLE WITH THROUGH FLOW

K. Aly[1] and N. Idvorian[1]
[1] Babcock and Wilcox Canada, Cambridge, Canada

Abstract

The tube side intermittent flow in U-tube bundle of vertical condenser is numerically simulated using RELAP5 MOD3.4. The flow rate intermit between a maximum and minimum value. The results of the simulation shows that reverse flow is established inside a group of the tubes from the outlet to the inlet channel -head during the low flow rate period. The reverse flow is seized when the minimum flow rate exceeds certain value. A steady state numerical simulation predicts the minimum inlet flow rate to establish the reverse flow. The effect of temperature difference at the inlet of the tubes and the tube resistance on the limit of the reverse flow inception is also studied. The heat transfer from the condensing steam on the shell side is also taken into consideration.

Introduction

Intermittent and oscillating flow in heat transfer equipments can result in temperature fluctuation-induced fatigue and the failure of some parts of the equipments [1]. For vertical condenser with U-tube bundle, unexpected thermal fatigue is observed in the inlet tube-sheet of the bundle due to the intermittency in the flow rate and hence temperature variation of the tube-sheet and tube welds. In order to reduce the intermittent cooling and heating - a minimum forward trickle flow to maintain the tube sheet at constant temperature was used. The introduction of forward trickle flow did not stop the thermal fatigue cracking of the inlet tube-sheet. One explanation for the fatigue failure of the tube welds is that a reverse flow is established during the trickle flow period, through a group of tubes, between the inlet and outlet cover-head though part of the tube bundles. This reverse flow will create thermal boundary conditions that favour fatigue failure.

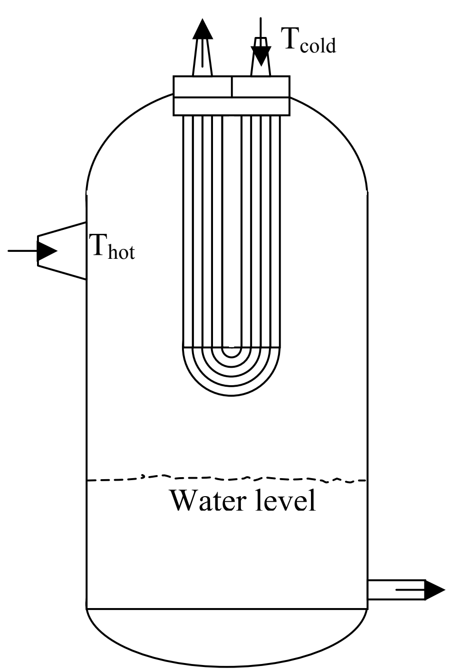
During natural circulation through recirculating steam generator, a reverse flow can be established in part of the inverted U-tube bundle [2-6]. De Santi et al [2] observed this phenomenon in the LOBI-MOD2 thermohydraulic integral test facility which represent a scaled down PWR system. Kukita et al [3] and Tasaka et al [4] observed similar behaviour in there scaled-down steam generator. Their results show that reverse-flow is established in the long tubes. Sander [5] developed a mathematical model describing this phenomenon for single phase. Jeong et al [6] expanded the mathematical model to describe this phenomenon for two-phase flow regime. The model shows that there is a lower bound of the flow rate below which the flow is not stable. Below this limit a reverse flow can be established between the inlet and outlet. The reverse flow is driven by the pressure increase at the outlet port due to the buoyancy effect.

This phenomenon can take place in the vertical U-tube bundle of the condenser as in the inverted U-tube bundle of recirculating steam generator. The purpose of this paper is to develop a numerical model using RELAP5 MOD3.4 [7] to predict the existence of the reverse flow during intermittent flow rate through the condenser bundle. The model is also used to predict the lower flow rate bound to prevent the establishment of the reverse flow. The effect of non-uniform temperature and different tubes resistance on the lower flow bound is also examined.

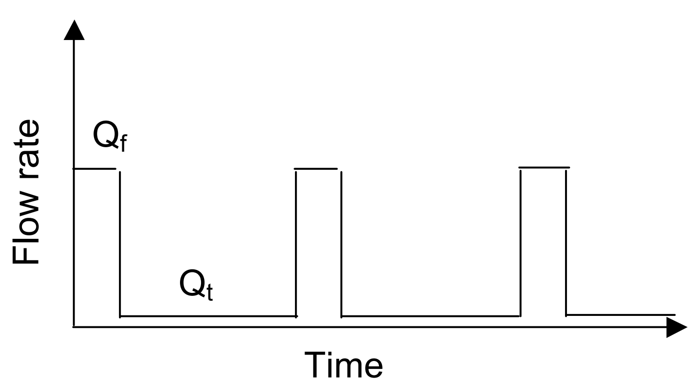
The paper is organized as follow. The first section of the paper describes the condenser geometry and the typical operating conditions. The RELAP5 model is described in the second section. The third section details the results of the transient analysis. The low bound flow rate limit is described in section four.

1. Condenser geometry and operating conditions

Figure 1a show a schematic of the condenser under investigation. The condenser is equipped with a U-tube bundle. The number of tubes of the bundle is relatively small such that the difference in the tubes length is small relative to the total length. The water level inside the condenser is maintained below the bundle at all time. The saturation pressure on the shell side is maintained constant at all time, and therefore the saturation temperature is maintained constant at T_{hot}. The cold water flows on the tube side at constant temperature T_{cold}. The flow rate is modulated with time as shown in Figure 1b. To keep the shell side pressure constant, the full flow (Q_f) in the bundle is introduced intermittently. However, to prevent thermal fatigue of the bundle tube sheet, a trickle flow (Q_t) is introduced to keep the bundle tube sheet at T_{cold}. The value of the trickle flow is originally determined based on the predicting heating rate of the tube sheet by the surrounding metal and fluid assuming a uni-directional flow.



a) Schematic of the condenser geometry

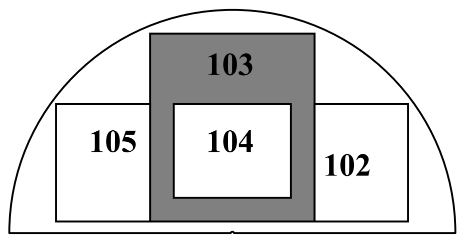


b) Typical tube side flow rate pattern

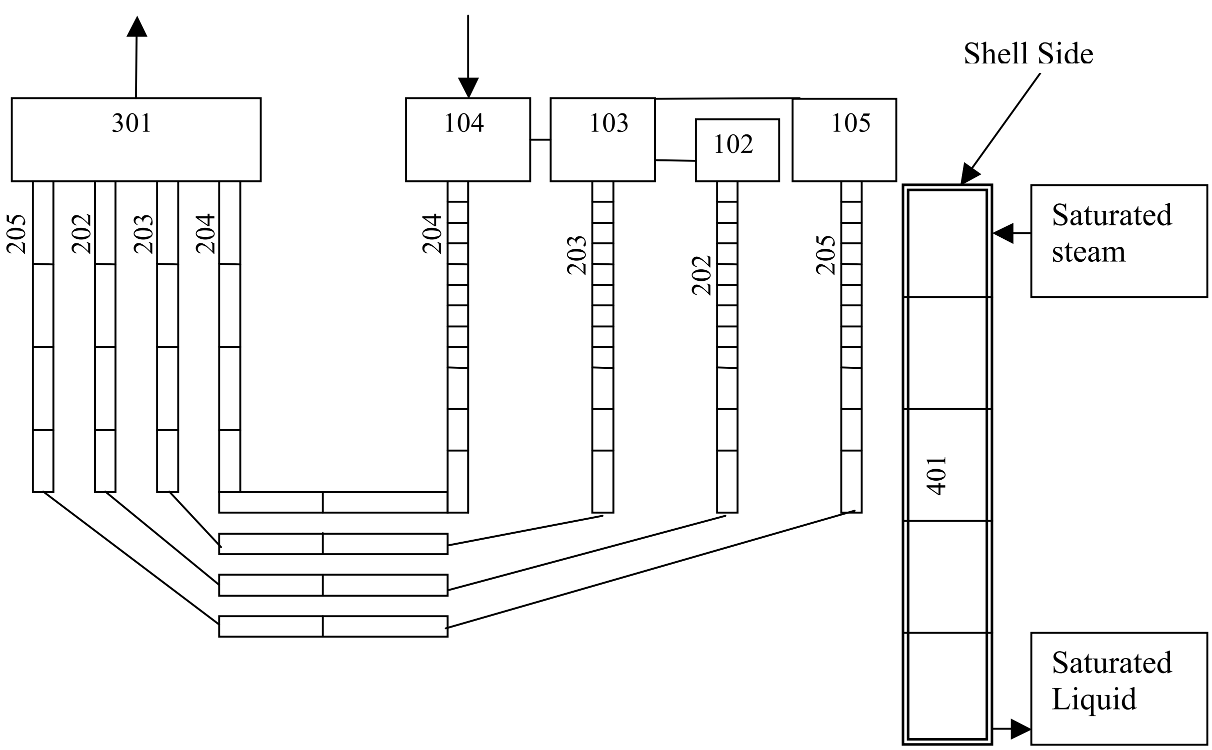
Figure 1 Condenser geometry and bundle flow rate

2. RELAP5 model

A one dimensional model of the condenser is developed. RELAP5 MOD3.4 code is used to perform the analysis. The model simulates both the tube side and the shell side of the bundle. The inlet and the outlet channel head is also modelled as shown in Figure 2. The shell side is divided into five control volumes. The bundle occupies only the first four control volumes. The condensed water is collected in the fifth control volume than drained to time dependent control volume maintained at the saturation pressure. Saturated steam is supplied to the second control volume from a time dependent volume to make-up for the condensate. This arrangement maintained the shell side at saturation condition all the time with the water level below the bundle.



a) Inlet channel head Nodalization



b) Nodal diagram of U-tube bundle and the shell

Figure 2 Nodal diagram of the RELAP5 model

The steady state adiabatic computational fluid dynamic (CFD) simulation of the inlet channel head showed that the flow is slightly mal-distributed as a result of the location of the inlet nozzle. Based on this analysis the channel head and the tube bundle are divided into four parts, as shown in Figure 2a&b. The inlet nozzle is connected to control volume (104). A small hydraulic resistance is specified between the different parts of the inlet quadrant to reproduce the CFD mal-distribution.

All the four sets of the u-tubes have the same length and the same nodal distribution. 87 nodes are used to represent each set of u-tubes. The inlet flow rate is specified using a time dependent junctions that dictates the inlet nozzle flow rate. On the outlet side, all the tubes are connected to one control volume which represents the outlet channel head. The outlet pressure of this quadrant is set by connecting the outlet nozzle to time dependent volume with the specified pressure. Heat structures are used to model the tubes metal and the heat transfer through them. The heat structures are connected to the tube side and the shell side. The heat transfer coefficient is specified by the program based on the flow regime. The material of the heat structures is inconel 600. The heat transfer between the inlet and outlet channel head is also simulated using heat structures that represents the channel head partition. The thermal capacity of the tube sheet metal and the channel head wall is not simulated in the developed model. Also, the difference in the u-tube length is not considered. This simplification can slightly affect the output.

3. Transient analysis results

The transient analysis is performed using the model developed in the previous section. The inlet flow rate is changed in a regular cycle where there is 10 second of full flow (Q_f) followed by 70 second of trickle flow (Q_t). The flow changes from one value to another over 2 sec to minimize any numerical oscillation due to sudden change in the flow.

Figure 3 shows flow rate through the four different U-tube subsets. The fluctuation of the flow is clearly repeatable after the third cycle which indicates that there is no effect of the initial conditions on the results. The positive value means that the flow is going into the tubes in the normal flow direction and the negative value means the flow is going in the reverse direction. During the 10 second of full flow, water is going in the normal direction in all the tubes. However, during the trickle flow period the flow is reversed in two of the four sets of U-tube. The flow is always reversed in the middle tubes (204) and normal in (202). The amount of the reverse flow is about 3 times the trickle flow. Changes in the inlet channel geometrical model do not stop the reversed flow but it can change the set where the reversed flow is established (e.g 202 instead of 204).

Figure 4 shows the temperature at the inlet of the four different set of the U-tube. During the full flow, the temperature is equal to T_{cold} in the tubes. During the trickle flow period, the temperature rises close to T_{Hot} in the U-tubes where reverse flow is established. In the rest of the tubes, the temperature rises to lower temperature due to the mixing of the inlet flow and the circulating flow.

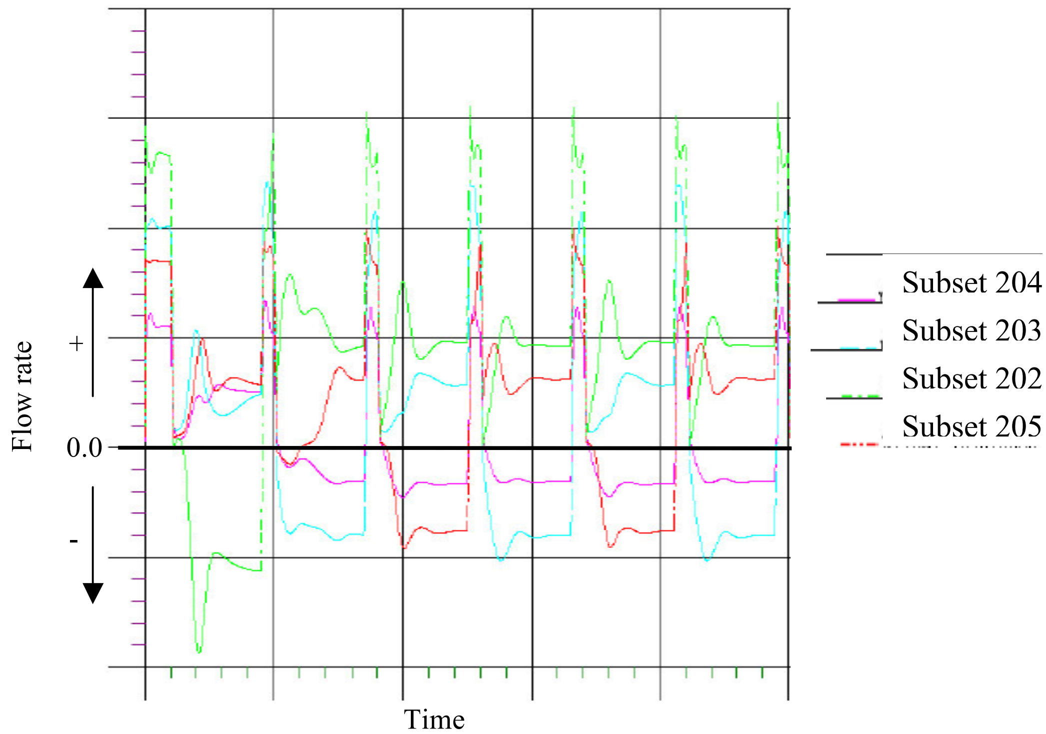


Figure 3 The mass flow rate in the different sets of the bundle in case of Q_t trickle flow

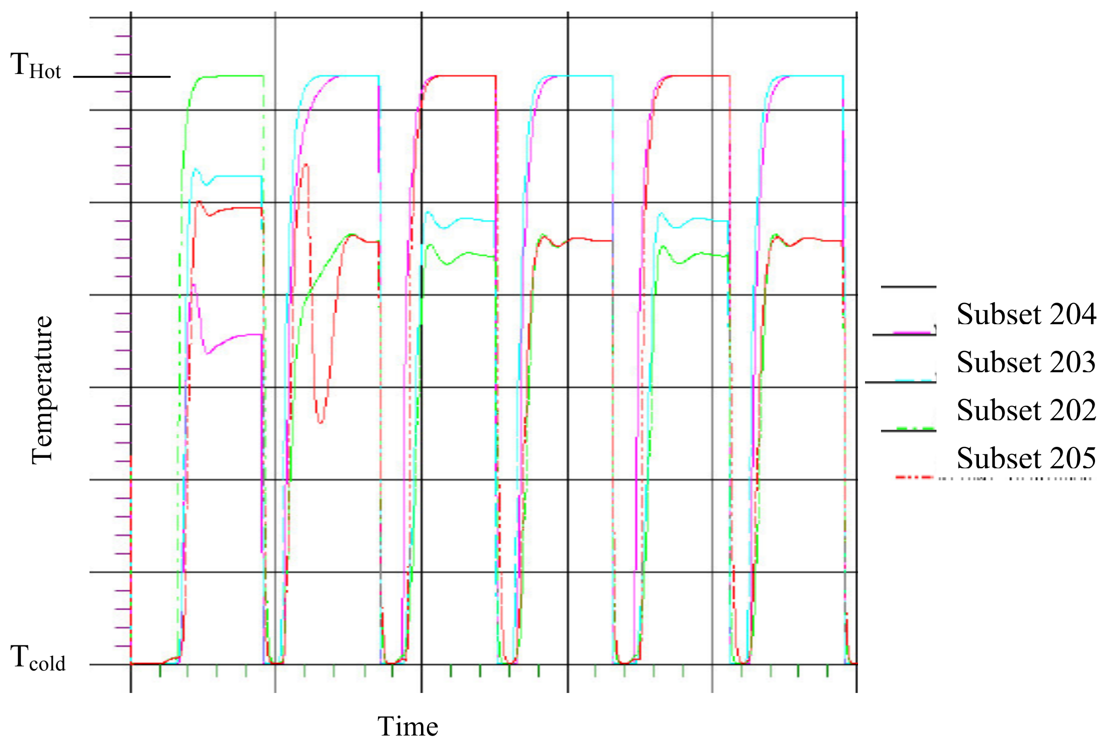


Figure 4 Temperature at the inlet of the different sets of the bundle with Q_t trickle flow

Another transient run is performed for the case of zero trickle flow. The results show that circulation is also established in case of zero trickle flow. However, the amount of circulating flow is much smaller. However, the temperature jumps from T_{cold} f to T_{Hot} in all tubes every time the flow stops. To eliminate the circulation of the flow in the U-tube during the trickle flow period, the trickle flow rate has to increase by about 7 folds. Figure 5 shows the flow rate through the four different u-tube sets for 7 Q_t trickle flow. It shows no reverse flow in any of the tube sets. For this condition the temperature is always T_{cold} at tubes inlet.

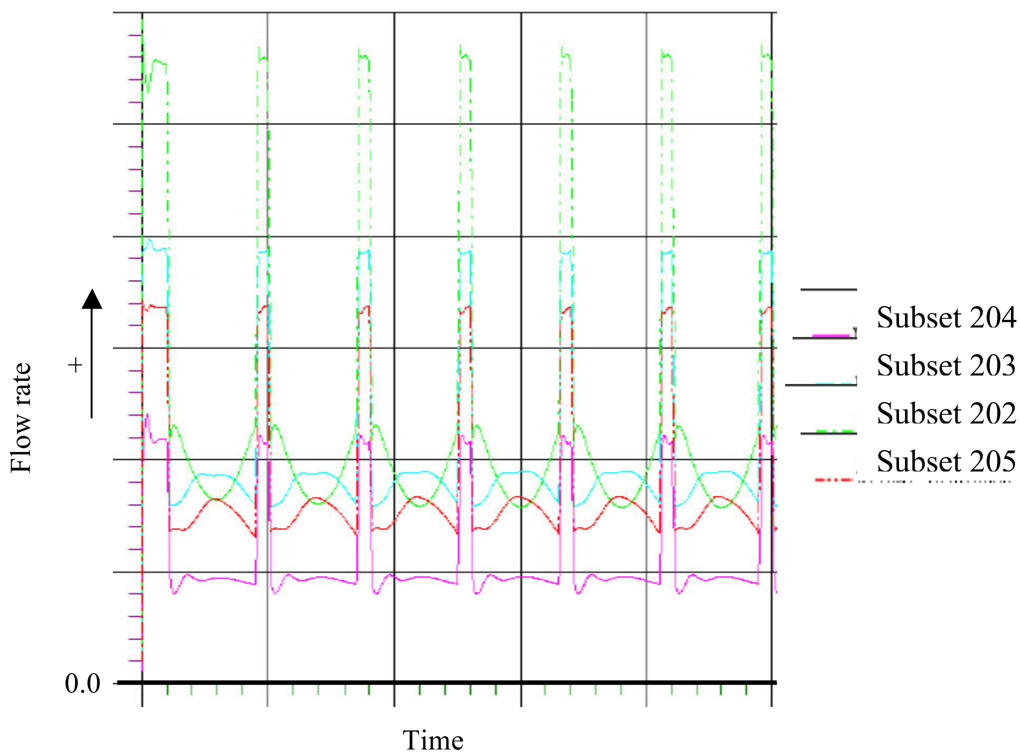


Figure 5 the mass flow rate in the different sets of the bundle in case of 7 Q_t trickle flow

4. Reverse flow limit

The reverse flow is driven by the pressure increase at the outlet port due to the buoyancy effect [5 and 6]. For the current case, the hydrostatic pressure gain in the downward leg of the U-tube is higher than the hydrostatic pressure drop in the upward leg. This is the result of the continuous heating of the water inside the tube which leads to a lower average water density in the upward leg compared to the downward leg. Over a range of flow rate the hydrostatic pressure gain is higher than the irreversible pressure drop due to friction and form losses. In this case, the pressure at the outlet channel head is higher than the pressure in the inlet channel head. A single U-tube analysis is performed using the RELAP5 to calculate the outlet-inlet channel head pressure difference at different flow rate as shown in Figure 6. The pressure difference increases with the flow up to about 6.25 Q_t. After that the pressure difference drops with the increase of the flow rate as the increase in the friction pressure drop is more substantial.

To understand the mechanism behind the circulation phenomenon, let us assume average flow rate of $3Q_t$ with mal-distribution such that a group (A) of the tubes will get what is equivalent to $3.5Q_t$ and a group (B) will get what is equivalent to $2.5Q_t$. Based on Figure 6, the pressure difference of group (A) is higher than group (B) which will cause the flow of group (A) to accelerate and the flow of group (B) to decelerate to the point when the flow in group (B) will stop and reverse flow will be established.

If the average flow is higher than $6.25Q_t$, it can be seen from Figure 6 that in case of flow mal-distribution the difference in pumping pressure will decelerate the higher flow rate and accelerate the low flow rate to reach more uniformly distributed flow and no reverse flow can be established. Therefore, the circulation will only be established if the flow rate is lower than $6.5Q_t$. This is in agreement with the transient analysis that predicts $7Q_t$ as minimum trickle flow to avoid circulation during the cycle operation. The difference between 6.25 and $7Q_t$ is explained in the next sub-section.

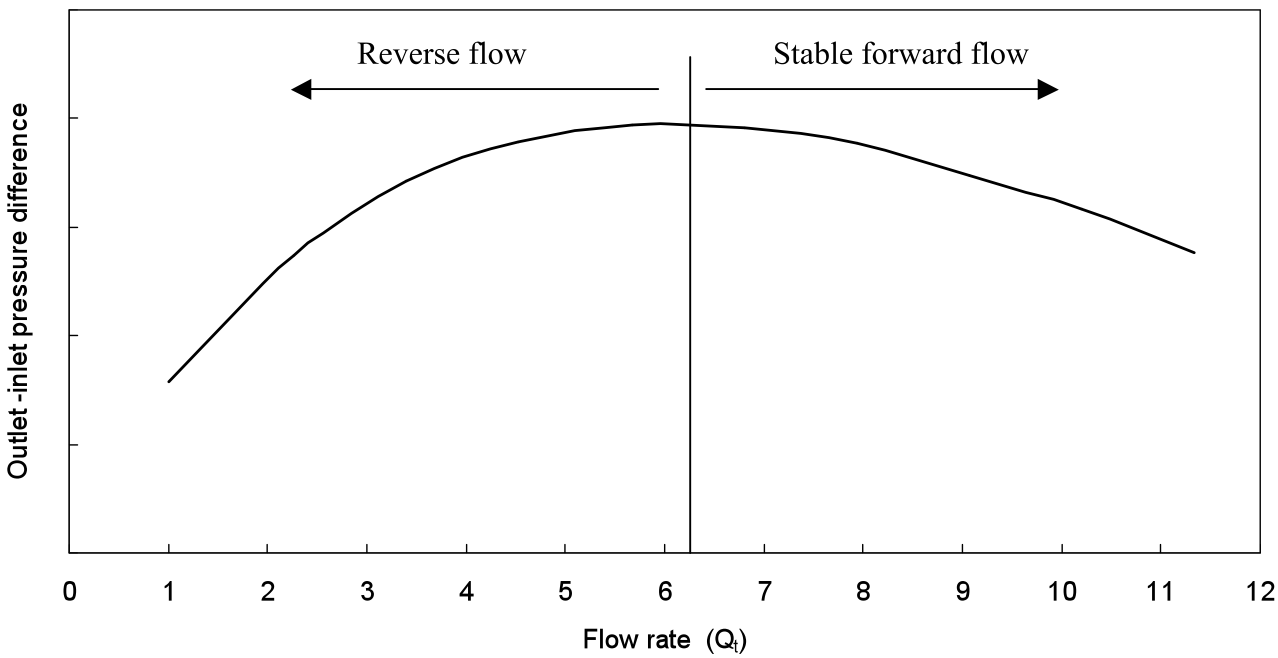


Figure 6 Outlet-inlet channel head pressure difference as function of the flow rate

4.1 Effect of temperature difference

In this section, the effect of the inlet temperature of the water flow on the circulation mechanism. Figure 7 shows the outlet-inlet channel head pressure difference for two different inlet flow temperatures. The higher temperature is $1/8(T_{Hot} - T_{cold})$ warmer than the T_{cold}. It is clear that the increase in the inlet temperature reduces the outlet-inlet channel head pressure difference, but without significantly changing the trend or the flow rate at which the outlet-inlet channel head pressure difference peaks. When the flow in some of the tubes is heated more than the other tubes, the outlet-inlet channel head pressure difference of this group of the tubes will be different than the rest even if the flow rate is the same. Difference in the inlet temperature can be the result of thermal stratification or heating by the adjacent walls. For this case, the flow rate above which no circulation will occur is the vertical dash-dot line in Figure 7. It is located somewhere along the horizontal dashed line. The reason is if the flow rates in the two groups of tubes are at the end of the dashed line, any increase in the flow rate of one side will cause a reduction of its outlet-inlet channel head pressure difference and subsequently reducing the flow rate to the original value at the end of the dash line. In the case of the transient simulation, the flow going into the periphery tubes are slightly wormer than the middle tubes due to the heat up by the channel head walls. Therefore, the trickle flow has to increase to $7Q_t$, which is higher than $6.5Q_t$, to stop the circulation.

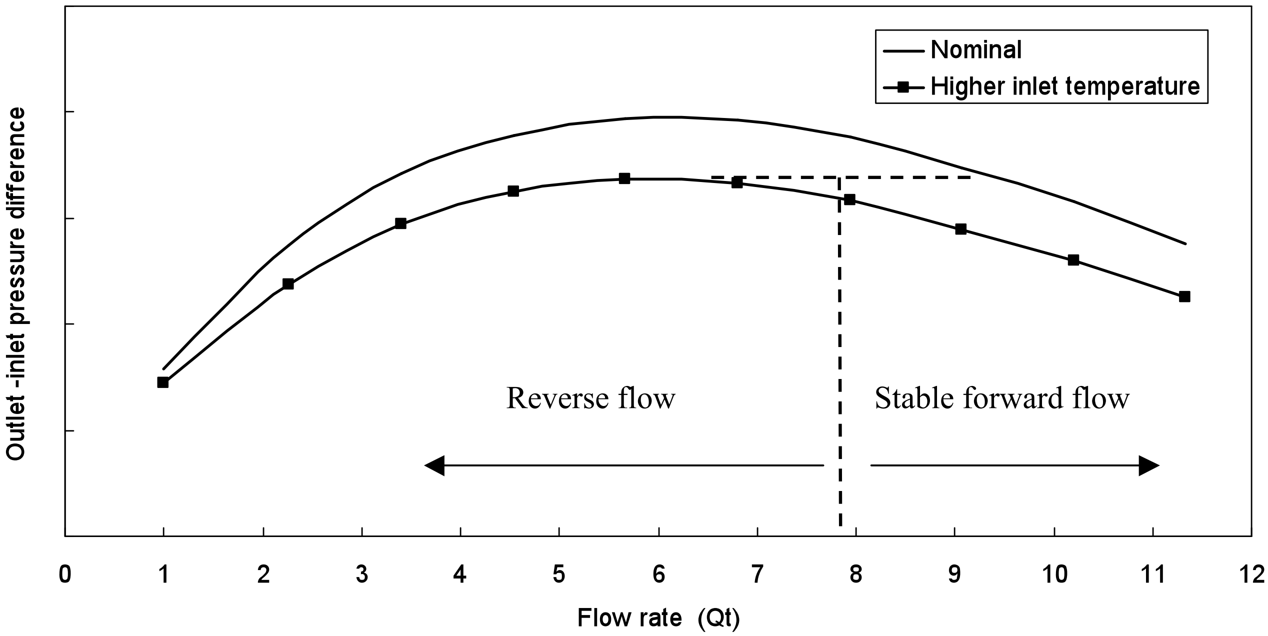


Figure 7 Effect of changing the bundle inlet temperature on the outlet-inlet channel head pressure difference.

4.2 Effect of hydraulic resistance

Figure 8 shows the effect of introducing a hydraulic resistance at the end of the U-tube on the outlet-inlet channel head pressure difference. The inlet temperature in both cases is T_{cold}. As expected when a resistance of loss coefficient is introduced the outlet-inlet channel head pressure difference drops as part of it being consumed to overcome the hydraulic resistance. However, more importantly the curve shows that the peak of the pumping pressure moves to a lower flow rate. This means that the minimum flow rate needed to stop the circulation is lower when the hydraulic resistance increases.

However, the required hydraulic resistance is relatively high which will result in a high pressure drop during full flow operation. Thus, increasing the hydraulic resistance is not a practical solution of preventing the reverse flow during low flow operation. However, if the hydraulic resistance is different from one tube to another, this can promote reverse flow at higher flow rate similar to the effect of inlet temperature.

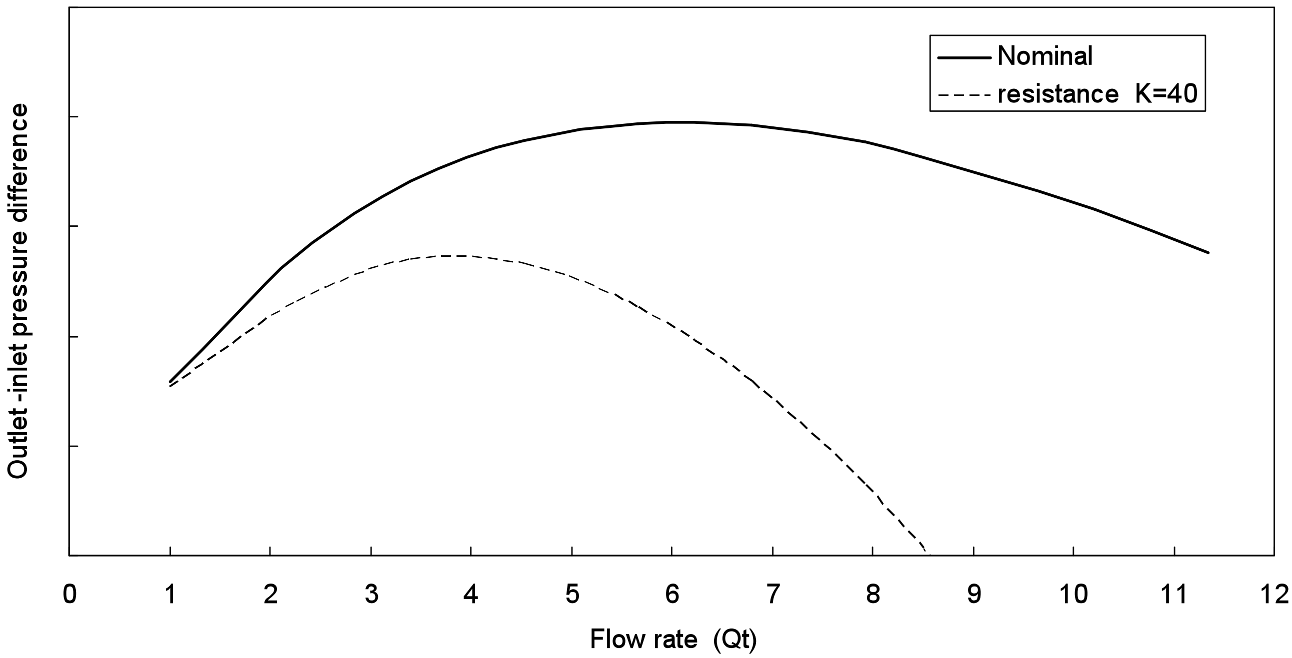


Figure 8 Effect of increasing hydrodynamic resistance on the outlet-inlet channel head pressure difference.

5. Conclusion

A numerical simulation is performed using RELAP5 MOD3.4 to investigate the flow pattern in U-tube bundle of vertical condenser during intermittent inlet flow. The simulation results show that reverse flow is established in some of the tubes during the low-flow period. During the high-flow period, the flow is in normal direction in all the tubes. The reverse flow pattern is seized to exist when the low-flow rate is higher than low bound flow rate.

A steady-state analysis is performed using the single U-tube RELAP5 model to determine the low bound flow rate. The model successfully predicts the trend and the phenomenon that induces the reverse flow. The effect of inlet temperature difference between tubes is investigated. The results show that a non-uniform inlet temperature results in a higher low bound flow rate to stop the reverse flow. A similar effect results from difference is the hydraulic resistance of the U-tubes.

6. References

[1] K. Iida, "A review of fatigue failures in LWR plants in Japan", Nuclear Engineering and Design, Vol. 138, 1992, pp. 297-312.

[2] De Santi, G. F., Piplies, L., and Sanders, J., "Mass flow instabilities in LOBI steam generator U-tubes array under natural circulation conditions", Proceedings of second international topical meeting on Nuclear power plant thermal hydraulics and operations, Tokyo Japan, 1986, pp. 2-158 to 2-165.

[3] Kukita, Y., Nakamura, H. and Tasaka, K., "Non-uniform steam generator U-tube flow distribution during natural circulation tests in ROSA-IV large scale test facility", Nuclear Science and Engineering, Vol. 99, 1988 289–298..

[4] Tasaka, T., Kukita, Y., Koizumi, Y., Osajabe, M. and Nakamura, H., "The results of 5% small-break LOCA tests and natural circulation tests at the ROSA-IV LSTF". Nuclear Engineering and Design, Vol. 180, 1988, pp.37–44.

[5] Sanders, J., "Stability of single-phase natural circulation with inverted U-tube steam generators", transaction of the ASME. J. Heat Transfer Vol.110, 1988, pp. 735–742.

[6] Jeong, J., Hwang, M., Lee, Y. J., and Chung, B. D., "Non-uniform flow distribution in the steam generator U-tubes of a pressurized water reactor plant during single and two-phase natural circulations", Nuclear Engineering and Design, Vol. 231, 2004, pp. 303–314.

[7] RELAP5 MOD3.4 Manual, ISS, 2010.