RELAP5-3D THERMAL HYDRAULICS COMPUTER PROGRAM ANALYSIS

COUPLED WITH DAKOTA AND STAR-CCM+ CODES

A Thesis

by

OSCAR ARTAJERJES RODRIGUEZ

Submitted to the Office of Graduate Studies of
Texas A&M University
in partial fulfillment of the requirements for the degree of

MASTER OF SCIENCE

Approved by:

Co-Chairs of Committee, Yassin A. Hassan
                        William H. Marlow
Committee Member,       Timothy J. Jacobs
Head of Department,     Jerry Caton

December 2012

Major Subject: Mechanical Engineering

ABSTRACT

RELAP5-3D has been coupled with both DAKOTA and STAR-CCM+ in order to expand the capability of the thermal-hydraulic code and facilitate complex studies of desired systems. In the first study, RELAP5-3D was coupled with DAKOTA to perform a sensitivity study of the South Texas Project (STP) power plant during steady-state and transient scenarios. The coupled software was validated by analyzing the simulation results with respect of the physical expectations and behavior of the power plant, and thermal-hydraulic parameters which caused greatest sensitivity where identified: inlet core temperature and reactor thermal power. These variables, along with break size and discharge coefficients, were used for further investigation of the sensitivity of the RELAP5-3D LOCA transient simulation under three difference cases: two inch break, six inch break, and guillotine break. Reactor thermal power, core inlet temperature, and break size were identified as producing the greatest sensitivity; therefore, future research would include uncertainty quantification for these parameters. In the second study, a small scale experimental facility, designed to study the thermal hydraulic phenomena of the Reactor Cavity Cooling System (RCCS) for a Very High Temperature Reactor (VHTR), was used as a model to test the capabilities of coupling Star-CCM+ and RELAP5-3D. This chapter discusses the capabilities and limitations of the STAR-CCM+/RELAP5-3D coupling, and a simulation, on the RCCS facility, was performed using STAR-CCM+ to study the flow patterns where expected complex flow phenomena occur and RELAP5-3D for the complete system. The code showed inability to perform

flow coupling simulations and it is unable, at this time, to handle closed loop systems. The thermal coupling simulation was successful and showed congruent qualitative results to physical expectations. The locations of large fluid vortices were located specifically in the pipes closest to the inlet of the bottom manifold. In conclusion, simulations using coupled codes were presented which greatly improved the capabilities of RELAP5-3D stand-alone and computational time required to perform complex thermal-hydraulic studies. These improvements show greatly benefit for industrial applications in order to perform large scale thermal-hydraulic systems studies with greater accuracy while minimizing simulation time.

## ACKNOWLEDGEMENTS

TABLE OF CONTENTS

LIST OF FIGURES

## LIST OF TABLES

CHAPTER I

INTRODUCTION

For over 40 years, the main area of research in the nuclear field has focused on

nuclear power plants' performance during accident conditions. In order to simulate the

behavior of water cooled reactors, the nuclear engineering community developed several

complex system thermal hydraulic codes. RELAP5-3D, developed by Idaho National

Laboratory, is one of the most used best-estimate thermal-hydraulic system codes. It is

capable of conducting steady-state, transient, and postulated accident simulations,

including loss of coolant accidents (LOCA) and a multitude of system transients, of

Light Water Reactors (LWRs) [1].

RELAP5-3D is a very powerful tool in the nuclear engineering industry, but it

contains some disadvantages common with complex thermal-hydraulic codes. The

results of the codes are affected by the variation from the nominal values of the

boundary and initial conditions. As a consequence, evaluation of the uncertainty of the

output parameters of such codes must be performed. Due to the relatively large number

of sources of uncertainties, a preliminary sensitivity analysis would provide the user with

important information about the simulated system response for selected input parameters

before proceeding with the uncertainty quantification. Using a system code such as

RELAP5-3D for a sensitivity analysis would require several simulations to analyze the

system response to each input parameter of interest with large computing time and

effort. In addition, thermal-hydraulic codes are designed to show the behavior of the full

system under investigation. In this effort, RELAP5-3D does not have the ability to

analyze localized flow phenomena where a detailed flow study is needed. Computational fluid dynamic (CFD) codes are the appropriate tools to analyze problems where a detailed fluid flow and heat transfer study is needed. CFD is a very computationally expensive analysis tool; therefore, it is only used in specific sections of a system.

In the present study, RELAP5-3D has been coupled with two separate codes in order to overcome these limitations. In the first chapter, a sensitivity analysis of the South Texas Project (STP) Pressurized Water Reactor (PWR) during steady state and transient conditions was performed using RELAP5-3D coupled with Design Analysis Kit for Optimization and Terascale Applications (DAKOTA) in order to facilitate input processing and data extraction of the study. The second chapter is a simulation of the Reactor Cavity Cooling System (RCCS) experimental facility using RELAP5-3D coupled with the CFD code STAR-CCM+, using STAR-CCM+ to better estimate the local effect of the fluid flow and RELAP5-3D to compute the thermal-hydraulic behavior of the remaining sections.

CHAPTER II

RELAP5-3D COUPLED WITH DAKOTA

**Introduction**

The containment building of a typical Light Water Reactor (LWR) is designed to contain the radioactive materials released during an accident and to facilitate the core cooling during a LOCA event. Under such postulated accidents, the Emergency Core Cooling System (ECCS) provides the required coolant flow to remove the decay heat from the reactor core and bring the system to the cold shutdown condition. During the first phase of the blowdown, the ECCS uses the cold water (nominal 85 ºF) contained in the Reactor Water Spent Fuel Tank (RWST) located inside the containment. The same water is discharged directly in the containment via containment sprays to keep its pressure under desired limits. Once all the water in the RWST is depleted, the ECCS begins to recirculate the water that has been dislodged through the break into the reactor containment and collected in the sump. The NRC requires such long-term cooling processes in the event of an accident. To protect the ECCS components from possible damage induced by the debris created during the blowdown phase and transported through the containment floor into the sump by the water flow, a set of screens are typically installed for each ECCS train, reducing the amount of debris that could be injected into the primary system and may impact the required core cooling.

In 1992, a steam line safety relief valve accidently opened in the Barseback nuclear power plant in Sweden. The jet ripped away insulation from neighboring pipes which caused debris to collect in the sump. The amount of scrap ejected by the jet stream

was enough to cause a blockage of the intake strainers. Though the accident posed no threat, under different circumstances it could have caused a failure of the recirculation system of the Emergency Cavity Cooling System. The incident drew the attention of the international community to look for a solution. Concerns arose over the generation of the debris during a LOCA in Light Water Reactors (LWR), transport in the containment from the generation site to the sump, and the effect that such debris may induce to the safety injection performances (in particular to the injection pumps) and to the core cooling capabilities that may be altered by the amount of debris that may bypass the sump strainers. These concerns relevant to the long term cooling associated with the ECCS during a postulated LOCA became a concern to the industry outlined by the Generic Safety Issue 191 (GSI-191)[2].

The evaluation of the overall behavior of a nuclear reactor during such a scenario is best performed using a thermal-hydraulic system codes; however, the results of system codes are affected by the variation from the nominal values of the boundary and initial conditions[3]. As a consequence, the evaluation of the uncertainty of the output parameters of such codes must be performed [4]. Due to the relatively large number of sources of uncertainties, a preliminary sensitivity analysis would provide the user with important information about the simulated system response for selected input parameters before proceeding with the uncertainty quantification. Sensitivity analysis, in turn, may require several simulations to analyze the system response to each input parameter of interest with large computing time and efforts.

DAKOTA (Design Analysis Kit for Optimization and Terascale Applications) was developed by Sandia National Lab to provide engineers and other disciplinary scientists with systematic and rapid means to obtain improved optimal designs or understand sensitivity/uncertainty using simulation-based models. DAKOTA can handle a variety of iterative methods, including optimization, sensitivity analysis, and uncertainty[5]. DAKOTA was coupled with RELAP5-3D, a thermal-hydraulic system code developed by Idaho National Laboratory capable of conducting steady-state, transient, and postulated accident simulations, including loss of coolant accidents (LOCA) and a multitude of system transients, of Light Water Reactors (LWRs)[1], in order to greatly facilitate sensitivity analysis and uncertainty quantification studies.

A RELAP5-3D input deck of the South Texas Project (STP) nuclear power plant was created in order to study the thermal-hydraulic behavior of the system during LOCAs of different sizes and locations. The coupled software was validated by analyzing the steady-state simulation results of a two inch break with respect of the physical expectations and behavior of the power plant, and thermal-hydraulic parameters which showed greatest sensitivity where identified. These variables, along with a few parameters specific to the LOCA, were used for further investigation of the sensitivity of the RELAP5-3D transient simulation. In addition, the parameters of interest for uncertainty quantification of the RELAP5-3D simulations for both the steady state and LOCA simulation were identified.

**Power Plant Description**

The nuclear power plant considered for this study is a typical 4-loop Westinghouse PWR Nuclear Steam Supply System (NSSS). The main features for the reactor designs are the following:

The main features of such system are summarized in the flowing bullets:

- Four independent primary loops, identified as A (loop with the pressurizer), B, C, and D;

- Three independent Safety Injection (SI) trains connected to loops B, C and D (no cross connection headers), with one High Pressure Safety Injection (HPSI), one Low Pressure Safety Injection (LPSI) pump, one accumulator and one sump strainer per train;

- Safety injection in cold legs, downstream the Reactor Coolant Pumps (RCP) with possible manual switchover to hot leg (after 5.5 hours from the beginning of the long-term cooling phase);

- One Residual Heat Removal (RHR) exchanger connected to each LPSI (downstream) activated during the long-term cooling phase;

- Large dry containment with two fan coolers per SI train (six in total) and 3 containment sprays trains with independent pumps;

- Automatic safety features actuated during a LOCA, including reactor scram, containment sprays and fan coolers actuation;

During a LOCA, some specific Plant Operating Procedure (POP) may be actuated. This includes:

- The reset (isolation) of one or more SI trains to avoid rapid core cooling (pressurized thermal shock), including the isolation of the accumulators when such condition is met;

- RCP shutoff at low pressure when SI are confirmed to be actuated;

- Shutoff of one or more containment spray trains based on the containment pressure level;

- Manual isolation of the RWST injection line when the low-low level of the water in the tank is achieved (switchover to the sump injection is automatic);

- Manual switchover to hot leg injection of two SI trains (one is maintained to cold leg injection); the selection of the trains to be switched to hot leg injection depends on the break location;

**RELAP5 Description**

The full RELAP5-3D nodalization for the study is shown below, Figure 1. The primary loops, identified with number 1(loop D), 2 (loop A), 3 (loop B), and 4 (loop C), were simulated independently to account for the flow asymmetry during the phases of the injection. All hydrodynamic components are identified by a three digit code with the initial number identifying the loop. The following sections will give a detailed description of major components of the system and the techniques adapted to model specific parts of the plant. Values of geometrical dimensions, setup points and initial conditions are proprietary information of the power plant used as reference and cannot be disclosed.
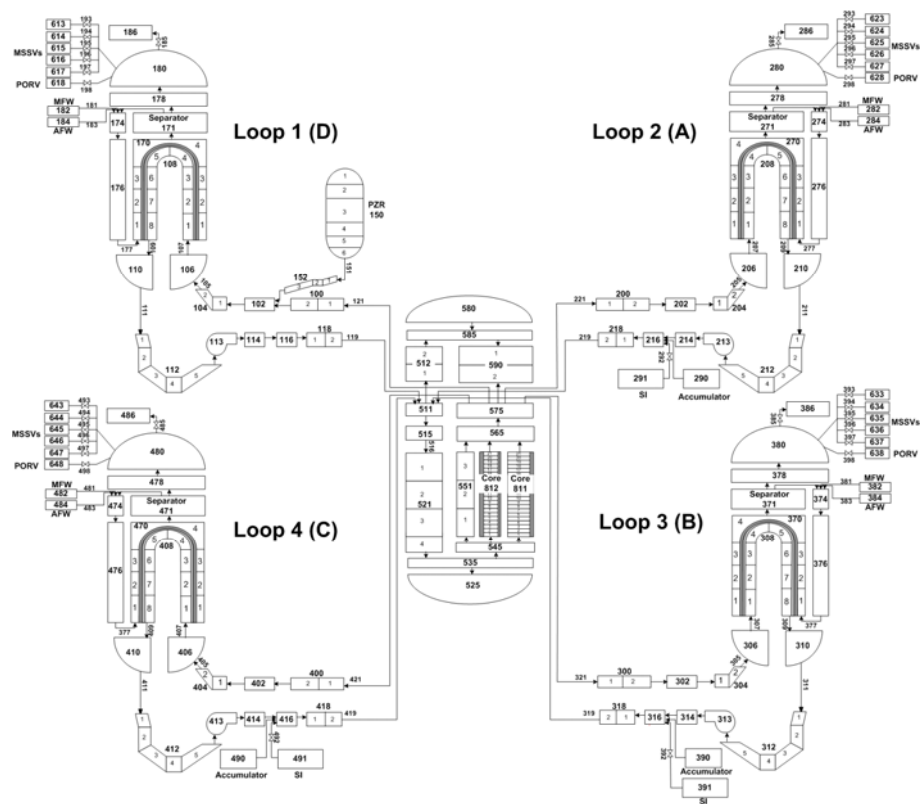
Figure 1. RELAP5-3D STP PWR Nodalization

The reactor pressure vessel nodalization is shown below, Figure 2. The cold legs are

connected to component (511) which in turn is connected with the upper volume of the

downcomer (515). The lower plenum consists of an annular region with 5 axial

subvolumes (521). The lower plenum is modeled by two components a single volume

(525) and branch component (535). Careful consideration was made in order to correctly

capture the dimensions and shape of the lower plenum using increasing cross-sectional

area in the axial direction. The core inlet channel consists of a branch component (545)

which divides the flow accordingly to the core bypass, hot channel, and average channel

of the core. The reactor bypass is modeled using a single pipe component(551) with

three axial sub-volumes. Water is distributed to four hot legs through a branch

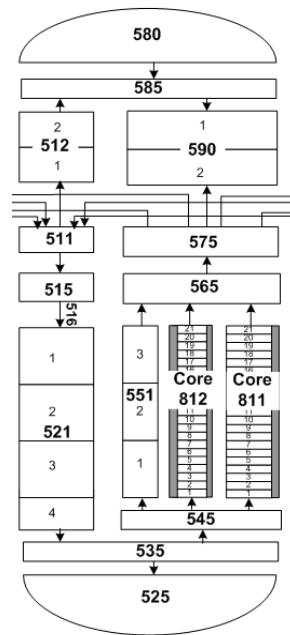component (575). The upper dome consists of a single volume (580) and branch (585).



Figure 2. PWR Reactor Pressure Vessel Nodalization

As also shown in Figure 2, the reactor core was modeled using two one-

dimensional pipe components. The pipe 811 simulates the average channels, where 191

assemblies were lumped together, and the pipe 812 was used to simulate the hot channel.

One heat structure was connected to the pipe 811. Two heat structures were connected to

the pipe 812 to account for the hot rod and for the remaining rods in the hot channel.

Appropriate radial peaking factors were defined to distribute the total power of the

reactor within the average channels and the hot channel (hot rod and average rods in hot

channel). Twenty-one axial nodes were used for both the hydrodynamic components and

corresponding heat structures. The chopped cosine shape was imposed as axial power shape. The point reactor kinetic was selected to model the neutron fission power generation in the core. Appropriate factors were defined to account for the fraction of the thermal power produced in the fuel rods and the one released to the coolant. The ANS-1973 decay heat model was selected to calculate the decay heat power generated after the reactor shutdown.

Both the primary and secondary side of the steam generator loop nodalization is shown below, Figure 3. All loops have the same nodalization, and only the initial number in the identifying code is changes according to the loop number.
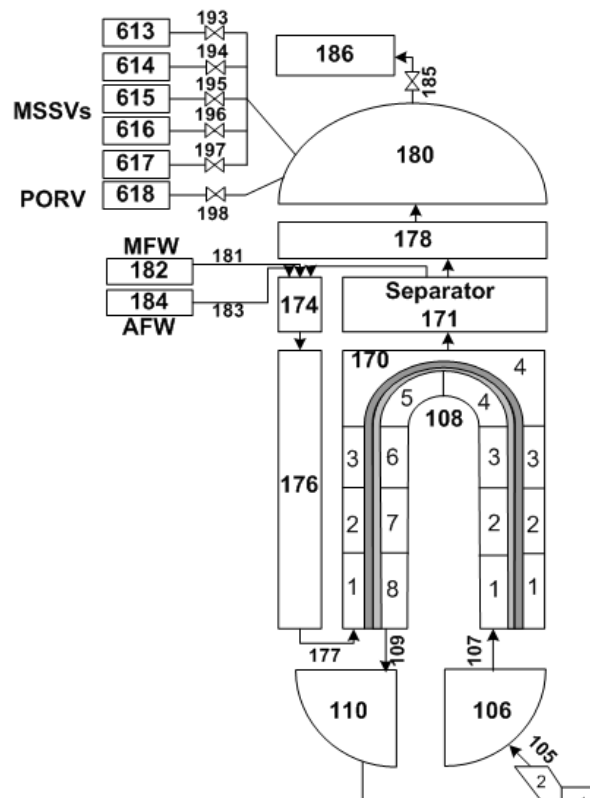


Figure 3. Steam Generator Loop 1 Nodalization

The inlet and outlet to the primary side steam generators are modeled using single volumes (X06 & X10) which are connected to the hot and cold leg, respectively. A single pipe (108) is used to represent the lumped number of steam generator tubes. The elevation change of the tube sub-volumes are set according to plant specifications. The secondary side contains the shell portion of the heat exchanger (X70), the separator (171), and the steam dome (178 & 180). Each generator has a Main Feedwater (MFW) and Auxiliary Feedwater(AFW) system represented by time-dependent volume to setup the boundary conditions of the injected water (temperature and pressure), and a time-dependent junction to impose the mass flow rate. For the purpose of these simulations the steam line was not modeled because during the initial steady-state calculations, a time-dependent volume (X86) is used to impose the secondary system pressure in order to achieve the design primary coolant average temperature. This is done by a dedicated integral control variable. During the accident, the Main Steam Isolation Valve (MSIV, X85), is closed and the secondary side is isolated. MFW and AFW injection activation times are setup according to the nominal conditions provided by STP. A heat structure was defined to simulate the steam generator tubes wall. Such heat structure is connected to pipe X08 (left boundary) and pipe X70 (right boundary).

The technique used to model the break is the one proposed in the RELAP5-3D user manuals suggested for different break sizes. For small (2 inch) and medium (6 inch) break sizes, a trip valve component was used to model the break, which stays closed during the steady-state phase and opens at the time of the break initiation. Such junction

was connected to one of the hydrodynamic components available in the nodalization of the cold and hot legs. As previously mentioned, the break was assumed to be located near the reactor vessel for both cold and hot leg break cases. The abrupt area change option was enabled to account for the additional pressure losses due to the sudden contraction and expansion of the flow at the break. The default chocked flow model was enabled since chocked condition is expected at the break during the phases of the accident. Similar technique was followed to model the DEG. In this case two trip valve components were used to simulate the flow from each side of the broken leg. These valves are closed during the steady-state phase and open simultaneously at the time of break initiation. To block any flow between the two sections of the broken leg (which are supposed to be completely shifted apart during the DEG scenario), a third trip valve was connected to both ends of the broken leg which is open during the steady-state phase and closes at the time of the break initiation. For all the break scenarios, the discharge volume (reactor containment) was simulated with a time-dependent volume. For a more accurate evaluation of the reactor system response, a table of the containment pressure as a function of the time for the entire simulation time was defined in the time-dependent volume. Such pressure was extracted by previous simulations performed by STP for each break condition under investigation.

**Dakota Description**

DAKOTA (Design Analysis Kit for Optimization and Terascale Applications) was developed by Sandia National Lab to provide engineers and other disciplinary

scientists with systematic and rapid means to obtain improved optimal designs or understand sensitivity/uncertainty using simulation-based models. DAKOTA was first released as an easy-to-use interface between simulation codes and optimization algorithms. More recently, DAKOTA capabilities were expanded to interface with other types of iterative analysis methods such as uncertainty quantification with nondeterministic propagation methods, parameter estimation with nonlinear least squares solution methods, and sensitivity/variance analysis with general purpose design of experiments and parameter study capabilities[5].

For the purpose of this study, DAKOTA functions as a black box relationship with RELAP5-3D. DAKOTA is executed using commands that are supplied in an input file which specify the type of analysis to be performed. The direct interface, or coupling, is considered to be "semi-intrusive" because it requires modification to the simulation code. Each case is contained in a separate subfolder. Once the RELAP5-3D simulation is finished DAKOTA extracts the data which it then outputs to a file. Therefore, the main advantage to using DAKOTA is that it offers access to a broad range of iterative capabilities through a single, relatively simple interface between DAKOTA and RELAP5-3D while allowing parallel processing for the simulations. A diagram showing the communication is shown in Figure 4.
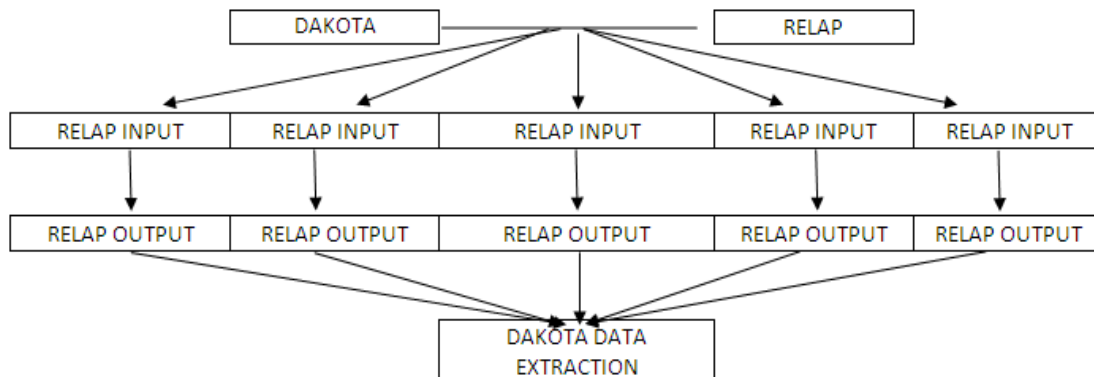
Figure 4. DAKOTA/RELAP Communication Diagram

To run the coupled software five different input files were necessary such as:

- Main Input file, to setup the type of analysis to be run and and the methodology

- Parameter file, where input parameters of interest are stated

- User Code Command file, containing the commands to run the case

- User Code Input file, which contains the RELAP5-3D coupled input deck

- Data Collection file, where the list of selected outputs is provided

The main input file is separated into five different sections required for the coupling:

- Method

- Variables

- Interface

- Responses

- Strategy

The method section in DAKOTA main input file specifies the name and controls of the iterator. The iterators include optimization, uncertainty quantification, least squares, design of experiments, and parameter study iterator. Each iterator has a certain input requirements unique for each study. Two different methods were used in order to perform the sensitivity analysis on RELAP5: list parameter study and multidimensional parameter study. Multidimensional parameter study allows the user to input a lower and upper bound and specify the number of partitions for the variable that is desired to study, which is done in the variable section. This is a very fast consistent method of performing the sensitivity analysis. The list parameter study requires the user to manually input every value desired to perform the study. This enables the user to have full control of the study. The variables section indicates the parameter set to be used by the particular method selected. These parameter sets can be continuous or discrete. Continuous and discrete range types include a lower and upper bounds, and discrete set types include the set values. The interface portion of the input file specifies how the function evaluation will be performed in order to map the input set of parameters into a set of responses. Function evaluation may be performed using algebraic mapping, interfaces to simulation codes, or a combination of the two. For the coupling of the software, interface to simulation codes was the only option in effect. The responses section is used to specify the output data set. Finally, the strategy section specifies the top level technique which will govern the management of the iterators and models.

The parameter file is identified in the main input file. The parameter file helps to keep track of the parameter sets and both output and input data sets. The user code command file is an executable file that specifies the executable analysis program. The data code input file is a copy of the RELAP5-3D input deck with modifications for the parameter study. The location in the input deck for the desired variable for study is replaced by the variable descriptor contained in brackets, e.g. {x1}, specified in both the main input file and parameter file. The data collection file is used to specify the RELAP5-3D execute command. In addition, it contains commands to extract the desired data from the RELAP output file.

**Steady State Method**

For the steady state sensitivity analysis, the methodology for DAKOTA was chosen as a multidimensional parameter study and the information such as lower and upped bounds and number of partitions for each particular case study was specified in the main input and parameter files. Once the correct interface was set up, the coupled software changed the input variable in the study and automatically ran the RELAP5-3D simulations. The software also extracted the defined output variables of interest to accelerate data analysis through the data collection file. The ability to run in multiple processors and the automated process saved a significant amount of time for simulations and data collection.

The following input parameters were identified to have an impact on the system behavior and were selected for the sensitivity analysis[6]:

- Reactor Thermal Power

- Steam generator pipe blockage due to sediment

- Core Inlet Temperature

- Primary Side Pressure

The output parameters of interest where the following:

- Secondary Side Pressure

- Mass Flow Rate

- Peak Clad Temperature

- Core Outlet Temperature

The variation of each parameter was set according to reasonable variations achieved during the nuclear reactor nominal operation. In particular reactor power was assumed to change within 0.2% of the nominal power. The fouling factor in the steam generators heat structures was varied between 0.9 and 1.0 to account for a maximum allowed design tube blockage of 10%. Core inlet temperature and primary pressure range of change was taken from STP power plant safety reports.  Table I summarizes the ranges used for the sensitivity analysis.

Sensitivity coefficient (S) represents the percentage of change on the system's response due to the percentage change of a system's input. The "sensitivity" of a parameter can be calculated using the following:

$$S = |(\delta w/w)/(\delta u/u)| \tag{1}$$

where 'w' is the output nominal value, and u is the input nominal value. The quantities

δw and δu are the divergence from the nominal value of w and u, respectively. A value

for S less than one means that the parameter is not very sensitive (a variation in u will

produce a small variation in w). In particular, when S is equal to zero, the parameter is

insensitive (any variation of u will not affect w). A sensitivity greater than one identifies

very sensitive parameters (a small change in u will results in a large change in w)[7].

Table I
Input Parameter for Steady-State Simulation Deviation from the Nominal Value

| Input Parameter | Deviation from nominal value |
|---|---|
| Reactor Thermal Power | +/- 0.2 % |
| Fouling Factor | 0.9 - 1.0 |
| Core Inlet Temperature | +/- 1.0 % |
| Primary Side Pressure | +/- 5.0 % |

**Steady State Results**

*Validation with uncoupled RELAP5-3D*

As a preliminary step to the sensitivity analysis, the coupled code was validated

by comparing the sensitivity analysis results for primary side pressure input parameter

during steady state with a parallel analysis using RELAP5-3D stand alone. The

comparison of the sensitivity, computed using (1), is shown in Table II. The results

obtained confirmed that the coupled software does not affect the thermal-hydraulic code

outputs and completed the validation phase of the coupled version.

*Table II*
*Sensitivity Comparison: Coupled Vs Uncoupled Software*

| Output<br><br>Primary Pres. | Secondary Side Pressure | Mass Flow Rate | Peak Clad Temperature | Core Outlet Temperature |
|---|---|---|---|---|
| *Coupled* | 0.008 | 0.053 | 0.060 | 0.016 |
| *Uncoupled* | 0.008 | 0.053 | 0.060 | 0.016 |
| *Error (%)* | 0.00 | 0.00 | 0.00 | 0.00 |

*Sensitivity Analysis*

The results of the sensitivity analysis are summarized in Table III where the

sensitivity coefficient S, calculated using (1), is shown for the selected cases at steady-

state. Plots of the simulations are reported in Appendix A. All output parameters show

significant sensitivity to the core inlet temperature. In particular the secondary side

pressure can be identified as a very sensitive parameter since its sensitivity coefficient

S=8.885 is higher than 1. Among those parameters, the peak cladding temperature is not

very sensitive (S=0.268 < 1). This is physically reasonable since the core inlet

temperature and subsequently the primary system average temperature control different

reactor parameters. In particular the secondary pressure (controlled via a control variable

in the input deck) is expected to changes to establish a new steady-state heat transfer in

the steam generators with a different boundary condition. The mass flow rate is also

expected to change with the core inlet temperature to remove the same steady-state core

thermal power with a different inlet temperature condition. Lower sensitivity of the peak

clad temperature is expected since the heat transfer capabilities through the core were found to be changed to guarantee the total heat removal rate. The sensitivity parameters estimated at steady-state for all the output parameters of interest were found to be much less than 1 for any other input variables such as reactor thermal power, fouling factor and primary side pressure. In particular the fouling factor, which accounts for the steam generators heat transfer coefficient degradation, was found to be the input parameter with the least effect on the selected output. For this reason, the selected outputs can be considered as insensitive to the steam generators fouling factor. This is essentially due to the limited range of change of the factor (10%) and to the ability of the system to account for the heat transfer degradation by a small change only in the secondary side pressure. The sensitivity of the other parameter was confirmed to be zero since the parameters are thermal-hydraulically independent.

*Table III*
*Steady State Two Inch Break Sensitivity*

| Output / Input | Secondary Side Pressure | Mass Flow Rate | Peak Clad Temperature | Core Outlet Temperature |
|---|---|---|---|---|
| *Reactor Thermal Power* | 0.047 | 0.025 | 0.039 | 0.080 |
| *Core Inlet Temperature* | 8.885 | 1.542 | 0.268 | 1.438 |
| *Fouling Factor* | 0.002 | 0.0 | 0.0 | 0.0 |
| *Primary Side Pressure* | 0.008 | 0.053 | 0.060 | 0.016 |

**Transient Methodology**

DAKOTA was modified to perform the sensitivity analysis of the LOCA

transient simulation study. A discrete methodology was chosen in order to have a finite

set of input values defined by the user. A vector of study for each parameter was

imposed on the respective RELAP5 input deck location. The sensitivity analysis was

performed in the 2" break, 6" break, and double ended guillotine (DEG) break.

As identified by the sensitivity study in the steady-state simulation, core inlet

temperature and reactor power showed a significant effect on the output nominal

parameters of the reactor; therefore, a study on the effect of these parameters on the

transient simulation was also a performed. Again, the boundaries of the sensitivity

analysis for both parameters were kept the same as in the previous study. Break-size

together with two-phase discharge coefficient and subcooled discharge coefficient are

parameters that were also chosen to perform a sensitivity study in order to observe the

effect a slight change from the nominal values could do. Break size varied between   +/-

5.0%. Both discharge coefficient values changed from  0.9-1.0. Table IV summarizes the

deviation of each parameter from the nominal values.

*Table IV*
*Input Parameter for LOCA Simulation Deviation from the Nominal Value*

| **Input Parameter** | *Deviation from nominal value* |
|---|---|
| *Reactor Thermal Power* | +/- 0.2 % |
| *Core Inlet Temperature* | +/- 1.0 % |
| *Subcooled Discharge Coefficient* | - 10%, -5% |
| *Two-phase Discharge Coefficient* | - 10%, -5% |
| *Break Size* | +/- 5.0 % |

The output parameters that are important for this study with an explanation on their effect on the reactor behavior are the following:

- Total ECCS flow rate: it affects rate of depletion of the water in the RWST and the RWST-to-sump switchover time. During the long-term cooling phase, this parameter affects the debris accumulation and bypass through the sump strainers and the pressure drop through the strainers.

- Primary Pressure: it imposes the boundary condition of the water discharged from the break as well as the injection flow rates through the high and low pressure injection pumps of the ECCS trains

- Time of Recirculation: it represents the time at which water is depleted from the RWST and begins to inject water that is suctioned from the sump. At time of recirculation the debris accumulation into the strainers begins to affect the ECCS possibly damaging the long term cooling of the reactor.

**Transient Results**

The final value at which the simulation trips, at the time of recirculation, will be recorded for all output parameters and analyzed for sensitivity from the nominal value. The plots of the final values due to the change in nominal value and plots of these values over time are shown in Appendix A.2.  Something to keep in mind is that during a blow down phase, reactor parameters are affected by both physical oscillations of the reactor and numerical oscillations. Due to the small parameter range and the effect of the

oscillation, the behavior of the parameters is not congruent with physical expectations. This is, nonetheless, the reason a sensitivity study must be performed for a thermal-hydraulic system code; to analyze how sensitivity specific parameters are in the RELAP5 input file during such scenarios.

A summary of the sensitivity analysis results for the two inch LOCA are shown in Table V. The ECCS total mass flow rate is the most sensitive parameter to all input parameters. The sensitivity value ranges from S=2.03 by the change in subcooled discharge coefficient to S=336.81 for the variation of reactor thermal power. The only insensitive value is the primary side pressure due to variation in the subcooled discharge coefficient (S=0.55).

*Table V*
*Two Inch Break Sensitivity*

| Inlet ╲ Outlet | ECCS Total Mass Flow Rate | Primary Side Pressure | Time of recirculation |
|---|---|---|---|
| Reactor Thermal Power | 336.81 | 79.70 | 54.53 |
| Core Inlet Temperature | 98.73 | 25.14 | 11.83 |
| Subcooled Discharge Coefficient | 2.03 | 0.55 | 2.02 |
| Two-phase Discharge Coefficient | 98.46 | 25.24 | 21.97 |
| Break Size | 9.28 | 2.37 | 3.64 |

All output parameters for the six inch break are very sensitive to reactor thermal power, core inlet temperature, and break size, shown in Table VI. Specifically, both primary side pressure and ECCS mass flow rate show very large sensitivity values to reactor thermal power and core inlet temperature. The time of recirculation shows little sensitivity to both the discharge coefficients (S<1.0).

*Table VI*
*Six Inch Break Sensitivity*

| Outlet / Inlet | ECCS Total Mass Flow Rate | Primary Side Pressure | Time of recirculation |
|---|---|---|---|
| Reactor Thermal Power | 104.26 | 211.62 | 15.93 |
| Core Inlet Temperature | 13.96 | 25.86 | 1.15 |
| Subcooled Discharge Coefficient | 2.32 | 4.8 | 0.17 |
| Two-phase Discharge Coefficient | 0.96 | 1.87 | 0.10 |
| Break Size | 3.12 | 4.31 | 1.59 |

In the double ended guillotine break, the break size was kept constant and was not a parameter in the sensitivity analysis. Table VII shows a summary of the sensitivity values. The time of recirculation has little to no sensitivity to input parameter variation; therefore, the time at which the RWST get drained is not affected by the input parameters under study. An important note, Reactor Thermal power and core inlet temperature still affect the ECCS total mass flow rate and primary side pressure greatly

(S>1), but the sensitivity of these parameters to the input values have decreased

significantly from the six inch break. Although, the primary side pressure sensitivity at

the full break is still larger than the two inch break.  All output parameters show very

small sensitivity (S<1) to the variation of subcooled discharge coefficient.

*Table VII*
*Double Ended Guillotine Break Sensitivity*

| Inlet \ Outlet | ECCS Total Mass Flow Rate | Primary Side Pressure | Time of recirculation |
|---|---|---|---|
| Reactor Thermal Power | 35.3 | 162.0 | 0.8 |
| Core Inlet Temperature | 1.2 | 10.2 | 0.5 |
| Subcooled Discharge Coefficient | 0.2 | 0.2 | 0.0 |
| Two-phase Discharge Coefficient | 0.4 | 1.8 | 0.0 |

CHAPTER III

RELAP5-3D COUPLED WITH THE CFD CODE STAR-CCM+

**Introduction**

The Very High Temperature Reactor (VHTR) was identified as one the designs

for the nuclear industry Next Generation Nuclear Plant (NGNP). One the key features of

the VHTR is the high temperatures at which the reactor operate, up to 1000 °C. The high

temperatures that are reached during operation allows for electricity production as well

as hydrogen generation. The VHTR is a helium-cooled, graphite moderated, thermal

neutron spectrum nuclear reactor. The VHTR operates with a direct Brayton cycle for

the generation of electricity[8]. Due to the high operating temperatures, heat is released

from the Reactor Pressure Vessel (RPV) into the containment building. New safety

systems have been implemented to ensure proper cooling of the structures surrounding

the vessel.

The Reactor Cavity Cooling System (RCCS) has been incorporated in the VHTR

design to guarantee the integrity of the fuel, Reactor Pressure Vessel (RPV), and

structure inside the cavity by removing heat from the surrounding area during normal

operation, shut-down mode, or accident conditions. The RCCS is designed to remove up

to 1.5 MW during an accident scenario and 0.7 MW during normal operation. The main

source of heat dissipation from the reactor pressure vessel has been confirmed to be

through thermal radiation[9].The RCCS consists of many pipes or rectangular ducts

surrounding the reactor pressure vessel (RPV), shown in Figure 5, and is organized by

panels each containing nine risers. Currently, there are different designs under

consideration for the RCCS: water cooled and air cooled. For this study, the water cooled RCCS design is used. In order to increase heat removal, several configurations were considered for the RCCS, including a metal panel welded to the back of the pipes (shield design) and mid-plane metal panel (fin design)[10].

The RCCS is a passive heat removal system. The buoyancy forces induced by the coolant density gradient inside the risers drives the coolant which collects into the top hot manifold and reaches the water tanks consisting of eight different reservoirs cooled by a heat exchanger to an external heat sink. Cooled water flows through the downcomer and back into the bottom manifold.
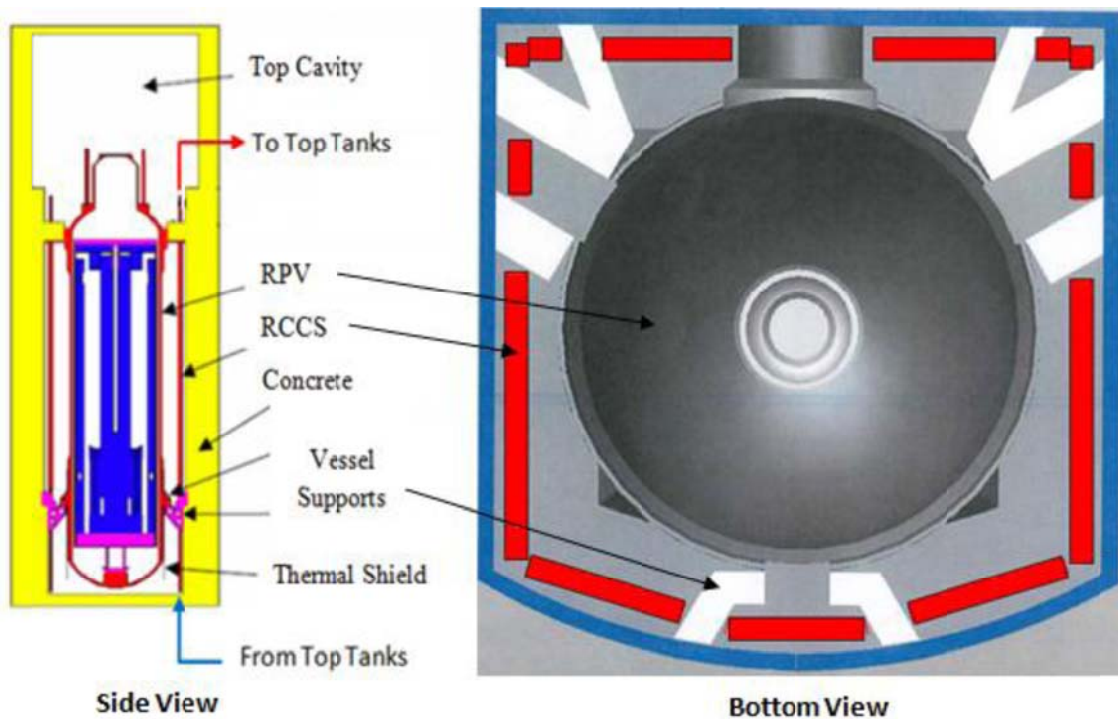


Figure 5. RCCS Design Surrounding the RPV

An experimental facility was designed at Texas A&M University to study the complex thermal-hydraulic behavior in the RCCS system, shown in Figure 6. The RCCS simulates a single mid-plane panel of the actual VHTR RCCS design consisting of nine risers, two manifolds (top and bottom) and one cylindrical water tank. The full experiment is designed to study the behavior of the system during both steady state and transient conditions, and due to the complex geometry of the RCCS, the flow behavior in the manifolds and risers are of interest.
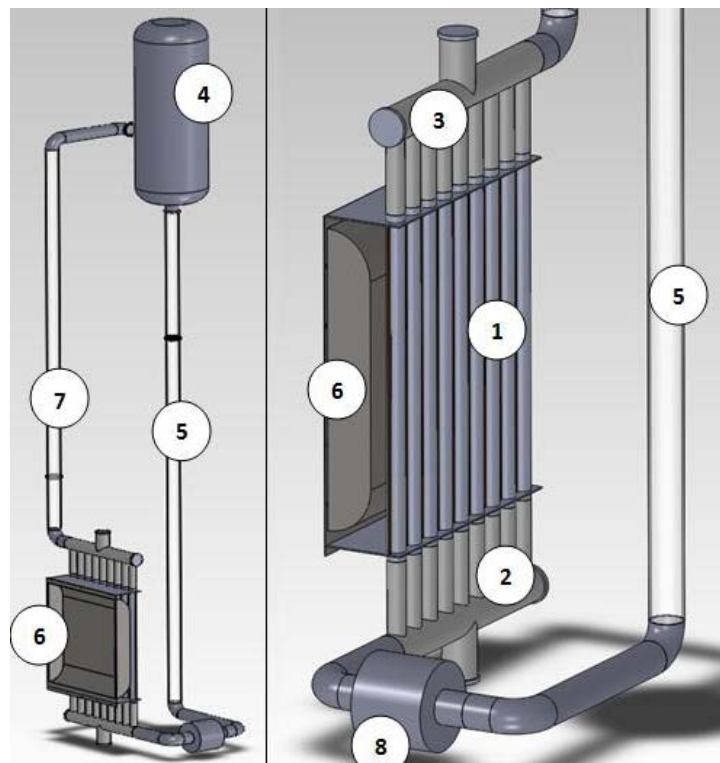


Figure 6. Texas A&M RCCS Facility Design

Due to the complexity of the expected thermo-hydraulic behavior of the experimental facility, a computer model of the system was desired. System codes are very powerful tools which allow full scale modeling of complex system which can run

long steady state or transient scenarios in a short time period, but complex geometries must be reduced to 1D representation; therefore, system codes do not have the ability to analyze localized flow phenomena where a detailed flow study is needed.

STAR-CCM+ is a commercially available Computational Fluid Dynamics (CFD) code by CD-ADAPCO. It features an elaborate user interface to facilitate model creation, meshing, solver execution, and post processing.  STAR-CCM+ is a three dimensional finite volume multiphysics code and can model detailed behavior of complex fluid flow and heat transfer problems in 3D components accurately. STAR-CCM+ can represent gaseous, liquid, solid, and porous media. Heat may be transferred through conduction, convection, and radiation. STAR-CCM+ can be used to study the flow behavior for both single and multi-phase flow, including boiling and cavitation phase changes. STAR-CCM+ is also capable of solving a multitude of turbulence models. Some of the available turbulence models include:

- k-epsilon (k-ε)

- k-omega (k-ω)

- Reynolds stress transport model

- Large-eddy simulation (LES

- Detached-eddy simulation (DES)

- Wall treatments

The user is able to model complex geometry and create surface and volume meshes. Complex input, output, and boundary conditions may be applied to surfaces to better

simulate elaborate systems. CFD codes are computationally very expensive, and the computational time only increases with complexity and size of the system[11].

Therefore, the coupling of both codes provides a very powerful computational tool. Coupling of the codes brings the greatest strengths of STAR-CCM+ (turbulence modelings, localized flow visualization, mulititude of heat transfer capabilities) together with the strengths of RELAP5-3D (fast computational time, ability to handle vast systems, accurate correlations). The RCCS system was chosen to test the capabilities and limitations for the coupling of STAR-CCM+ with RELAP5-3D.

**Coupling**

RELAP5-3D, version 3.0.0 Beta, was coupled with STAR-CCM+, Beta version 6.04. Both computer codes run as separate programs and have independent inputs/outputs. The RELAP5-3D Heat Structures, Time Dependent Volumes, and Junctions that are coupled with STAR-CCM+ are identified to RELAP5-3D using an extra input file. The CFD boundaries that are coupled with to the 1D-code are identified in the STAR-CCM+ GUI, and are matched up to the appropriate RELAP5-3D component.

There two types of coupling available between STAR-CCM+ and RELAP5-3D: Flow Coupling and Thermal Coupling. Thermal coupling couples a single side of a RELAP5-3D  heat structures to the wall surface of STAR-CCM+. Coupled surfaces must be defined as Fixed Temperature or Fixed Heat Flux. The coupling module collects data, temperature or heat flux boundary conditions, from RELAP5-3D and sends to

STAR-CCM+. It then collects data from STAR-CCM+ and saves into "oneDvalues" table. This data is sent back to RELAP5-3D when applying boundary conditions at the coupled boundary. The second type of coupling is flow coupling. The Coupling module collects Pressure Temperature data from the RELAP5-3D Time Dependent Volumes, and mass flux data from Time Dependent  Junctions and sends to STAR-CCM+.

The communication strategy commences when STAR-CCM+ is initialized. Automatically, RELAP5-3D begins running and establishes communication with the CFD code through sockets. Both codes run independently in the first time step and communicate consistently thereafter during each major time step. STAR-CCM+ has a much smaller time step requirement than RELAP5-3D due to the complexity of the code; therefore, RELAP5-3D takes the following time step once STAR-CCM+ time step is greater than the current RELAP5-3D time.

There are some limitations to the coupling that were identified through the testing of the code. First of all, the coupling only communicates information when both STAR-CCM+ and RELAP5-3D simulations are transient. Also, flow coupling is currently very limited. The coupled code is very powerful for compressible fluids such as air, but the code is not at stable when incompressible fluids are used. Furthermore, the flow coupling cannot be used for a closed loop system. In flow coupling, RELAP5 collects mass flow rate data from a Time Dependent Junctions (TDV) and fluid temperature from Time Dependent Volumes (TDV), but TDVs are used to specify hydrodynamic boundaries of a system where a fluid can enter or leave the system.

Therefore, a closed loop RELAP5-3D model, where Time Dependent Volumes are not used in the circuit, must be modified if flow coupling is desired.

**RELAP5-3D Nodalization and STAR-CCM+ Model**

*RELAP5Nodalization*

The initial RELAP5-3D nodalization is shown below, Figure 7. As noted, 300P constitutes the water tank above the experiment modeled as a pipe. The downcomer is modeled by 240B. It is connected by branches (245B and 250B) to the bottom manifold (255P). The nine risers are modeled as pipes 201-209 which are subdivided into 5 subvolumes to match the portion that is inside the cavity (heated by the vessel) and outside the cavity. The top manifold was modeled using horizontal pipes (225-227P) which exits to branches 230 and 235. These branches lead directly back to the water tank (300P). Figure 8 shows a detailed view of the heat structures representing the vessel (HS101), cavity walls(HS115), and the front and rear risers (HS10X0 and HS10X1, respectively). The view factors for the radiation enclosure between the vessel, cavity walls, and front riser heat structures were calculated using NEVADA. The cavity is represented by a single volume (100V) filled with air, and the space between the back cavity heat structure and the containment wall is represented by single volume 400Vfilled with air.
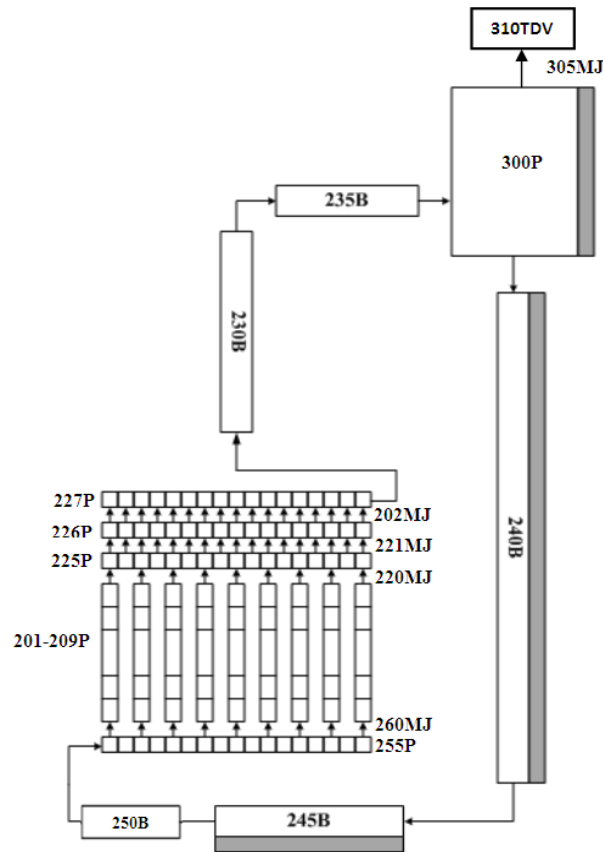
Figure 7. Full RCCS Nodalization
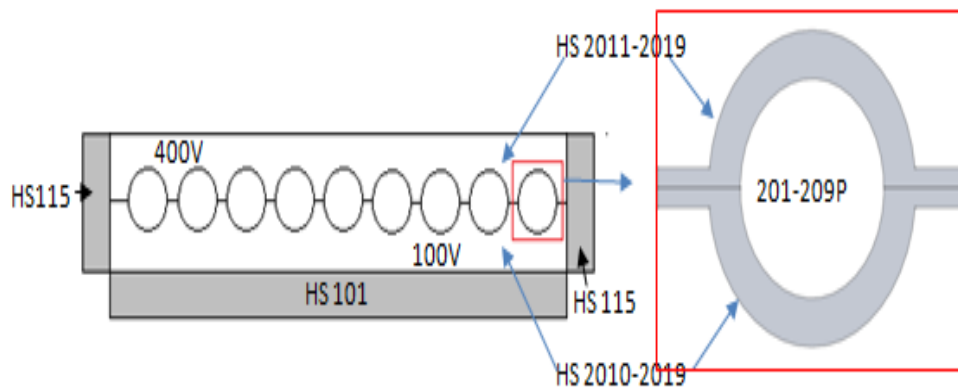


Figure 8. Top View of the RCCS Nodalization

Due to the flow coupling limitations that the coupled code currently has, the above nodalization was modified to enable the possibility of both thermal and flow coupling. The downcomer, water tank, and other branches were removed and replaced with time dependent volumes and connected to the bottom and top manifolds using time dependent junctions. The final nodalization for the simulation is shown in Figure 9. The boundary conditions of the time dependent volume were obtained by running a steady state simulation using the full scale experiment RELAP5 model and obtaining the boundary conditions at the entrance and exit of the manifolds.
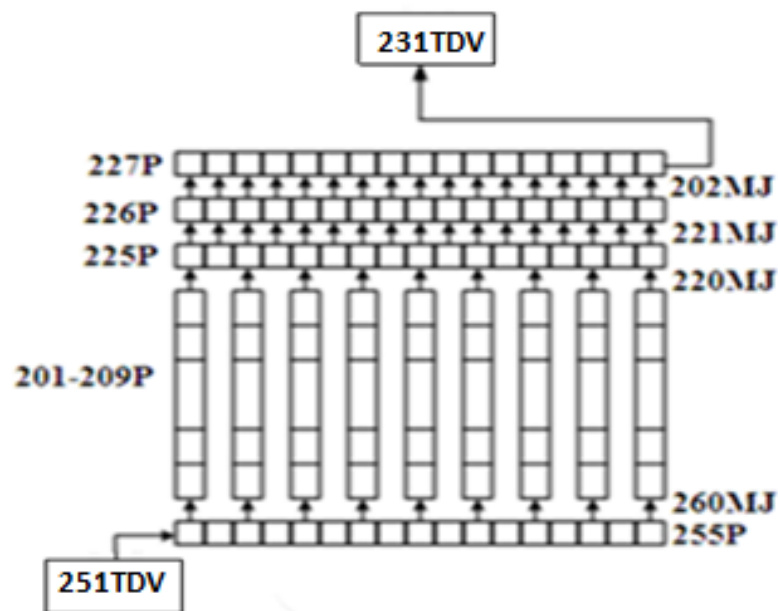


Figure 9. Modified Nodalization for RCCS

During the first time steps of RELAP5-3D, the simulation encounters numerical oscillations until it reaches a stable state. Until stability is reached, the simulation can

run for several seconds. The problem arises in the coupling because the STAR-CCM+ simulations require a very long time for a few seconds of real time data. Therefore, it is only communicating the numerical oscillations which are not accurate. In order to avoid this problem, the coupling was modified by CD-ADAPCO to enable restart files from RELAP5-3D to be coupled. This allowed the STAR-CCM+ simulation to get accurate data as soon as the simulation is initialized.

*STAR-CCM+ Model Description*

The STAR-CCM+ model is shown below, Figure 10. The bottom manifold, top manifold, and nine risers are represented by the flow field. The model was made using SolidWorks, 3-D computer aided drafting (CAD) tool. In order to represent the front and rear boundaries of the RCCS, each riser was modeled by half a pipe in the CAD model. The model was then imported into STAR-CCM+ where the inlet, outlet, and wall regions were specified. The model was meshed using trimmer, re-mesher, and prism layer. The model uses five prism layers in order to have a better prediction near wall laminar boundary region and trimmer models with a base size of five millimeters. Mesh convergence was reached with 1,020,808 cells.

Temperature-dependent water density was used as the primary fluid. The Realizable k-epsilon turbulence was selected with the Reynolds-Average Navier-Stokes equation. Segregated flow was the main solver dealing with the pressure and density equations. The simulation in STAR-CCM+ must be a transient simulation (Implicit Unsteady) in order for it to communicate with the thermal hydraulic code. Also, the 1D

Coupling and RELAP5 options must be selected. For thermal coupling stand-alone, the

boundary conditions at the inlet of the manifold where specified, shown in Table VIII.
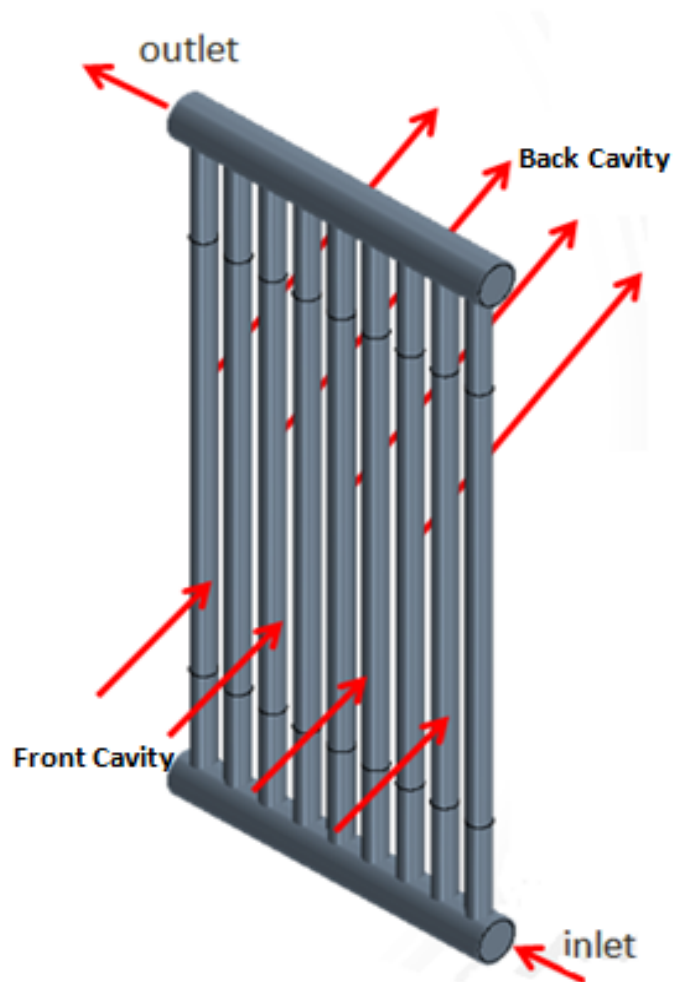


Figure 10. STAR-CCM+ RCCS Geometry

*Table VIII*
*Inlet Boundary Conditions for STAR-CCM+*

| | | |
|---|---|---|
| Inlet | Mass Flow Rate | 0.875 kg/s |
| | Temperature | 318 K |

When the RELAP model is turned on in STAR-CCM+, all boundaries have a new physics conditions option named "1D Coupling Specification". This is done in order to identify coupled and uncoupled surfaces. If the modeled is selected as "Coupled", all thermal properties disappear, but the user must still specify what type of information is being communicated from RELAP5: Temperature or Heat Flux. All front and back risers' physics conditions were changed to coupled surfaces selecting temperature as the boundary condition. In the tools section a RELAP5 option appears where the path to the RELAP5 input file and coupling input file are identified. Once the path is identified the user can select the "Start RELAP" option which initializes the coupling by reading the coupling input file and identifying the heat structures, time dependent volumes, and junctions which will be used in the simulation. Once the components are identified, a list in them appears as a subfolder. Here, the user identifies which STAR-CCM+ boundary will be coupled to the RELAP5 heat structure. Also, the orientation of the heat structure must be identified in terms of the STAR-CCM+ model axis. For my model, the orientation was set to (0.0, 1.0, 0.0) because the y-axis is in the same orientation as the surface that is coupled.

In order to overcome the transient limitation and obtain results for a steady-state simulation, STAR-CCM+ must be coupled with RELAP5-3D and ran as a transient simulation for a few time steps. Soon after STAR-CCM+ may be paused and the steady-state simulation may be specified and ready for processing. All thermal boundary conditions from RELAP5-3D are retained in the CFD boundaries. This process can be repeated until an acceptable convergence level.

**Results**

As shown in Figure 11, the temperature boundary imposed by the coupling is working properly. A qualitative analysis shows the front facing riser walls vary in temperature, and the center pipes have the highest temperature. This is expected with the radiation view factor calculations where the center boundaries receive the most thermal radiation. The RCCS boundaries facing the cavity wall are at a much lower temperature as expected. The largest velocity is located in the left most pipe and gradually decreases towards the inlet of the bottom manifold, shown in Figure12.
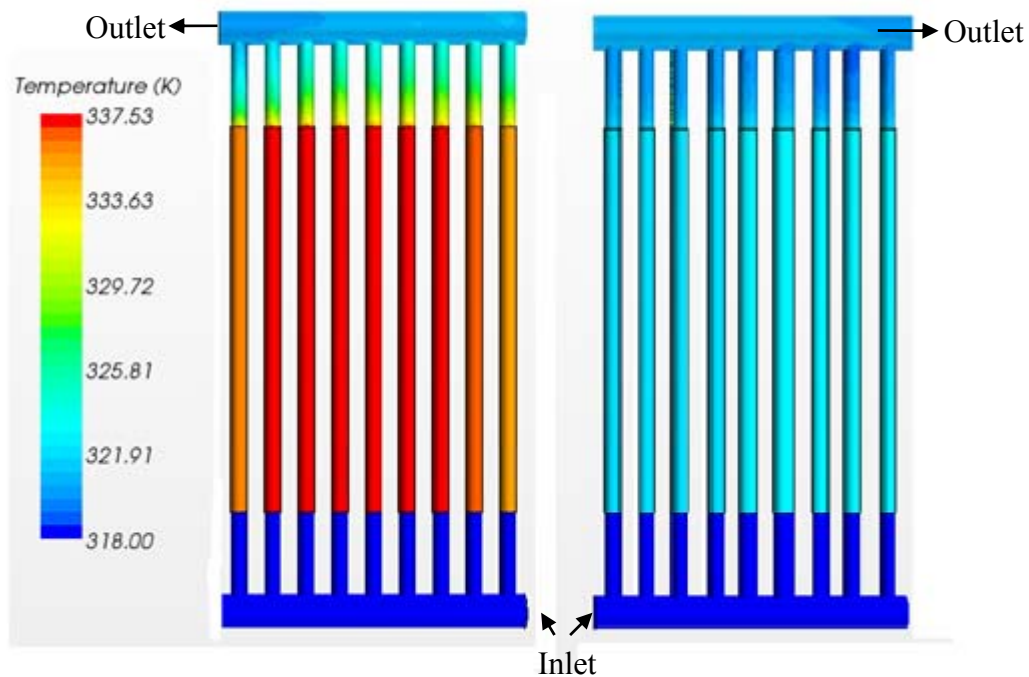


Figure 11. Front (left) and Rear (right) View Temperature Profile at RCCS Boundary

The temperature profile may be observed in Figure 12. There are large vortices in the riser closest to the inlet located in the bottom manifold, shown in Figure 13. This phenomenon may cause a higher local pressure drop and reduce the mass flow rate

through these risers. Also small vortices can be observed in the top manifold where the

risers meet the very top plenum of the manifold
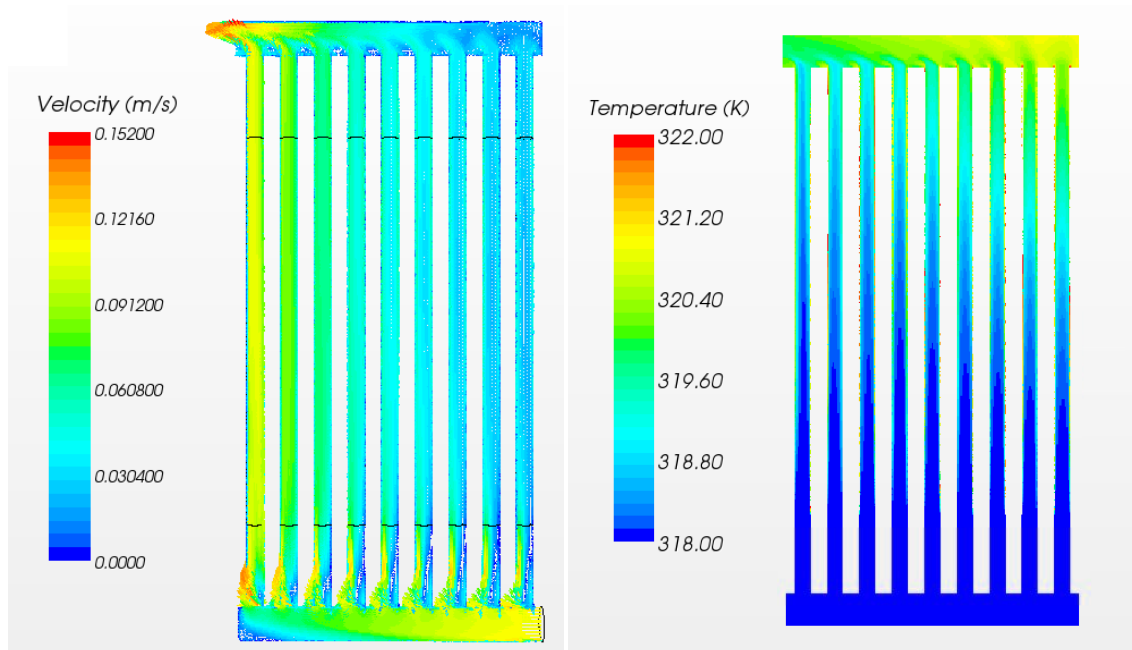


Figure 12. Full Geometry Velocity (left) and Temperature (right) Profile
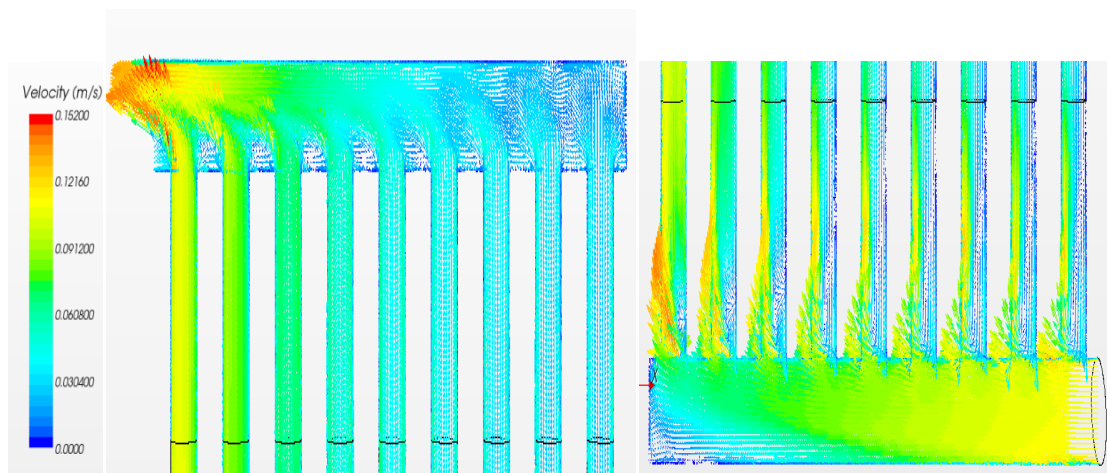


Figure 13. Velocity Profiles of the Top and Bottom Manifolds

CHAPTER IV

CONCLUSION & FUTURE WORK

RELAP5-3D thermal hydraulic system code has been successfully coupled with both

DAKOTA and STAR-CCM+ CFD code. Coupling with DAKOTA allowed for easier

input modification and data extraction of both steady-state and transient simulations. It

also allowed running simulation in parallel. The coupled code was verified simply

comparing the sensitivity coefficient results for a selected case with the stand-alone

RELAP5 version. The results of the steady state sensitivity analysis yielded the most

sensitive parameters and the input values which affected the output the most. These

output parameters were selected for further transient analysis. The results of the

sensitivity analysis of the transient cases showed a big difference depending on the break

size. In almost all parameters sensitivity would decrease as break size increases, with the

only exception being primary side pressure between the two inch break and six inch

break. The results of the sensitivity study helped to identify the most sensitive

parameters among the selected ones that could be used for future uncertainty

quantification for both steady-state and transient analyses. A successful steady-state

simulation using Thermal Coupling was done using RELAP5-3D coupled with STAR-

CCM+. Several limitations were identified throughout the study. These limitations are

currently under review by CD-ADAPCO. Locations where flow vortices are prominent

were also identified. Future work includes using the code to model a closed loop system

with future versions of the software.  In conclusion, simulations using coupled codes

were presented which greatly improved the capabilities of RELAP5-3D. RELAP5-3D

has been coupled with two separate codes: DAKOTA and STAR-CCM+. RELAP5 was coupled with DAKOTA in order to facilitate input processing and data extraction. Also, RELAP5-3D coupled with STAR-CCM+ helped to better estimate the local effect of fluid flow while performing a full nuclear reactor system simulation. The coupled engineering tools showed great improvements capability and computational time required to perform complex thermal-hydraulic studies. These improvements show greatly benefit for industrial applications in order to perform large scale thermal-hydraulic systems studies with greater accuracy while minimizing simulation time.

REFERENCES

[1] RELAP5, 1995. The RELAP5 Code Development Team, 1995. RELAP5/MOD3 Code Manual, vol. 1, Code Structure, System Models, and Solution Methods, NUREG/CR-5535, INEL-95/0174

[2] ECD/Nuclear Energy Agency (2004),*Debris Impact on Emergency Coolant Recirculation: Workshop Proceedings, Albuquerque NM, USA, 25-27 February 2004*, Nuclear Safety, OECD Publishing.

[3] Alessandro Petruzzi and Francesco D'Auria, "Thermal-Hydraulic System Codes in Nulcear Reactor Safety and Qualification Procedures," *Science and Technology of Nuclear Installations*, vol. 2008, Article ID 460795, 16 pages, 2008.

[4] A. Srivastava, H. G. Lele, A. K. Ghosh, H. S. Kushwaha, *Annals of Nuclear Energy* **35**, 323-334 (2008).

[5] Adams, B.M., Bohnhoff, W.J., Dalbey, K.R., Eddy, J.P., Eldred, M.S., Gay, D.M., Haskell, K., Hough, P.D., and Swiler, L.P.,"DAKOTA, A Multilevel Parallel Object-Oriented Framework for Design Optimization, Parameter Estimation, Uncertainty Quantification, and Sensitivity Analysis: Version 5.0 User's Manual," Sandia Technical Report SAND2010-2183, December 2009. Updated December 2010 (Version 5.1)

[6] V Vileiniskis, A Kaliatka, Uncertainty and sensitivity analysis of MCPs' trip events at Ignalina NPP, Nuclear Engineering and Design, Volume 224, Issue 2, September 2003, Pages 213-225, ISSN 0029-5493, 10.1016/S0029-5493(03)00103-1.

[7] B. L. Broadhead, B.T. Rearden, 2004, "Sensitivity and Uncertainty-Based Criticality Safety Validation Techniques", Nuclear Science and Engineering, 146, 340-366.

[8] D. Chapin, S Kiffer, J Nestell, "The Very High Temperature Reactor: A Technical Summary", *MPR Associates, Inc*, June 2009.

[9] H.J. van Antwerpen, G.P. Greyvenstein, "Evaluation of a detailed radiation heat transfer model in a high temperature reactor systems simulation model", *Nuclear Engineering and Design* 238 (2008) 2985–2994.

[10] R. VAGHETTO, " Simulation of a Simple RCCS Experiment with RELAP5-3D System Code and Computational Fluid Dynamics Computer Program," NURETH14, Toronto, Canada (2011)

[11] CD-ADAPCO, "User Guide: STAR-CCM+ Version 2.10.017," CD-ADAPCO (2007).

APPENDIX A.1

**Additional Plots Steady State Simulation**

*Reactor Thermal Power*
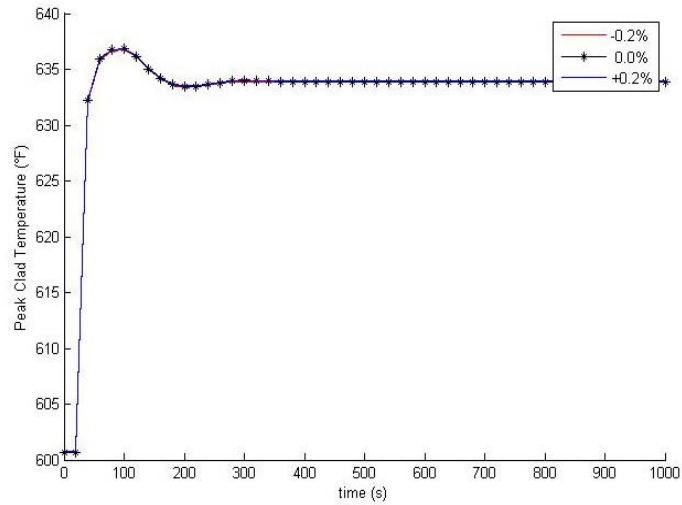


Figure A1. Peak Clad Temperature as a Function of the Reactor Thermal Power
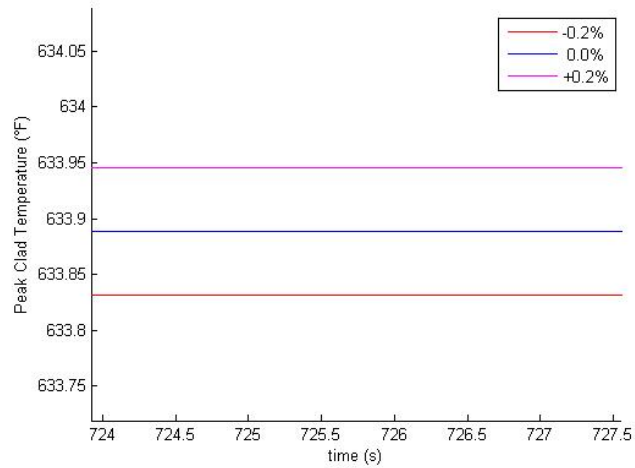


Figure A2. Peak Clad Temperature as a Function of the Reactor Thermal Power-
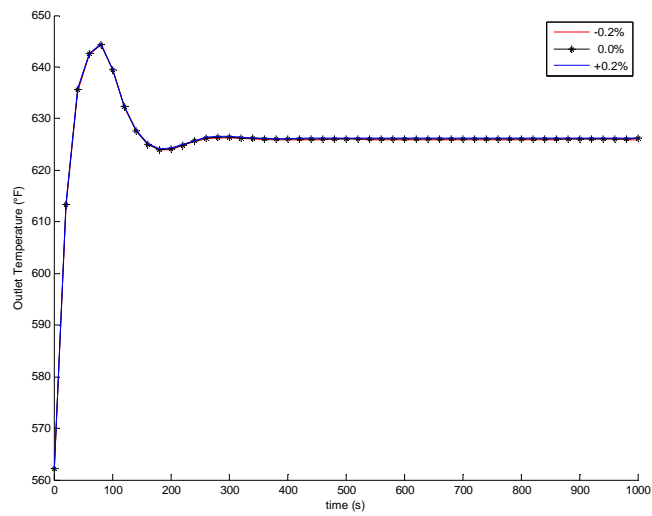
(Magnified)

Figure A3. Core Coolant Outlet Temperature as a Function of the Reactor Thermal
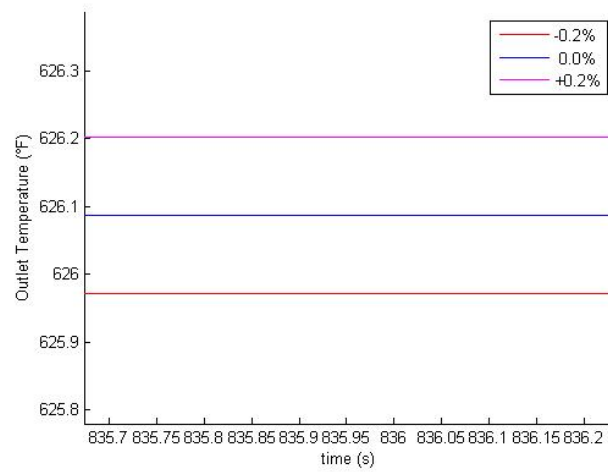
Power.



Figure A4. Core Coolant Outlet Temperature as a Function of the Reactor Thermal

Power (Magnified)
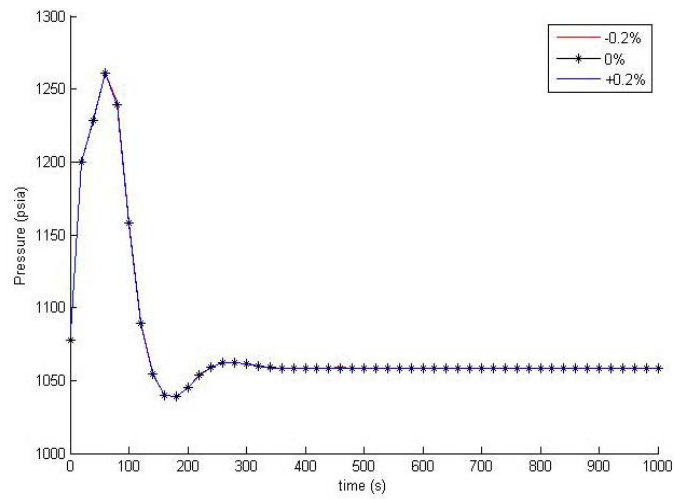
Figure A5. Steam Generator Secondary Side Pressure as a Function of the Reactor
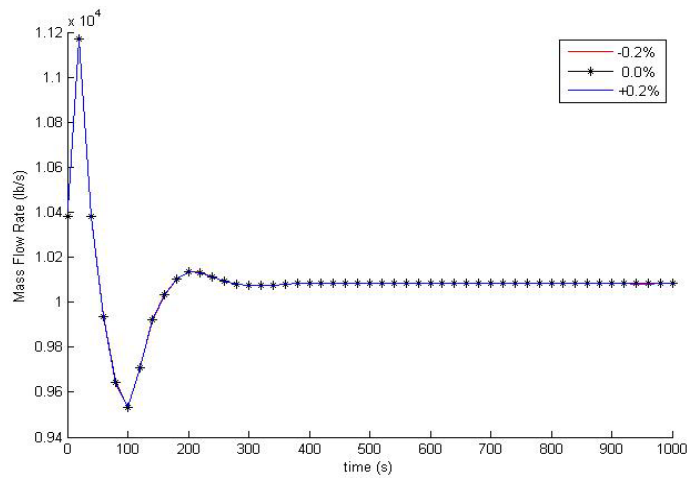
Thermal Power.



Figure A6. Hot Leg Mass Flow Rate as a Function of the Reactor Thermal Power.
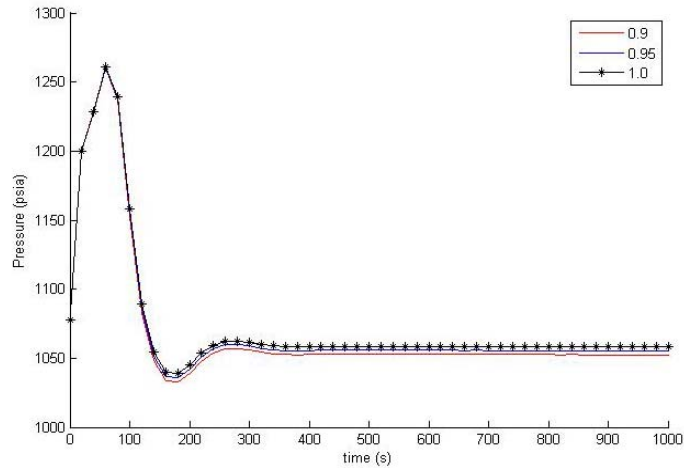
*Fouling Factor*



Figure A7. Steam Generator Secondary Side Pressure as a Function of the Fouling
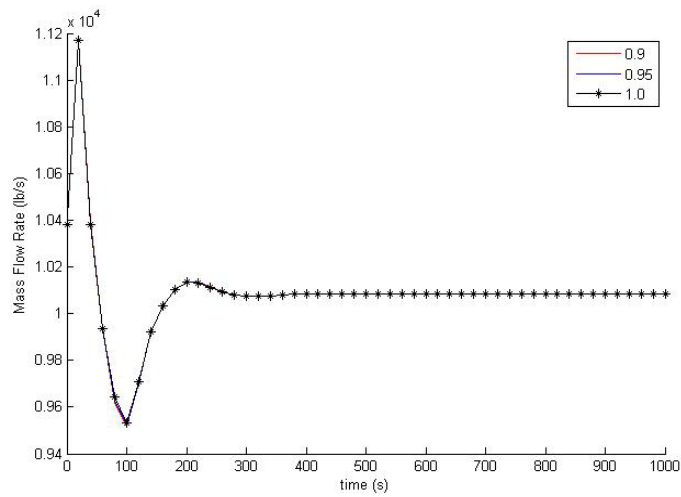
Factor



Figure A8. Hot Leg Mass Flow Rate as a Function of the Fouling Factor
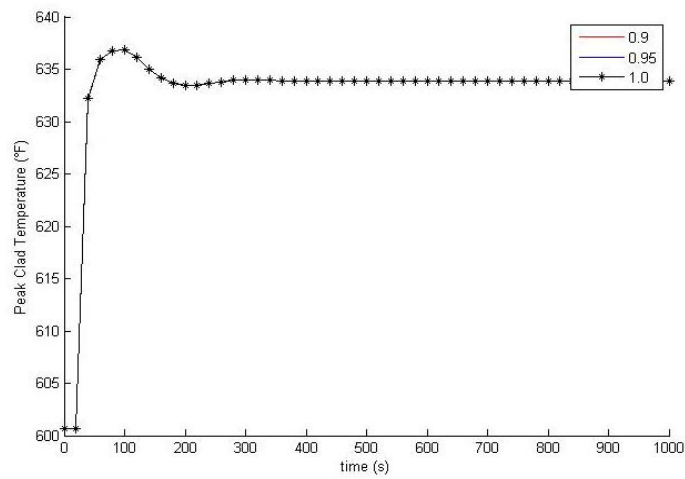
Figure A9. Peak Clad Temperature as a Function of the Fouling Factor
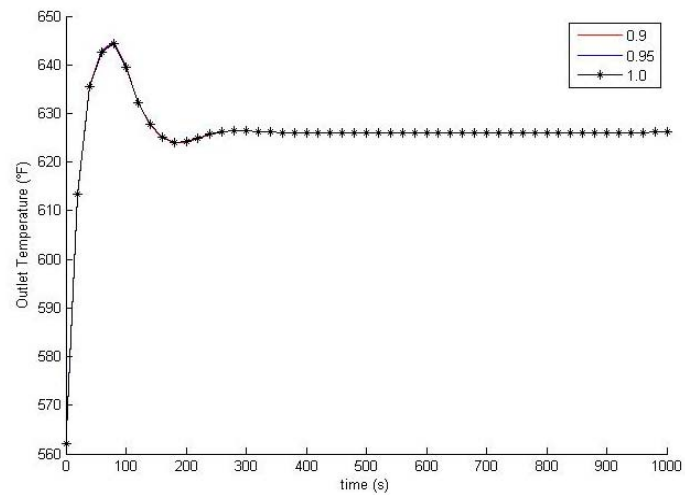


Figure A10. Core Coolant Outlet Temperature as a Function of the Fouling Factor

*Core Coolant Inlet Temperature*
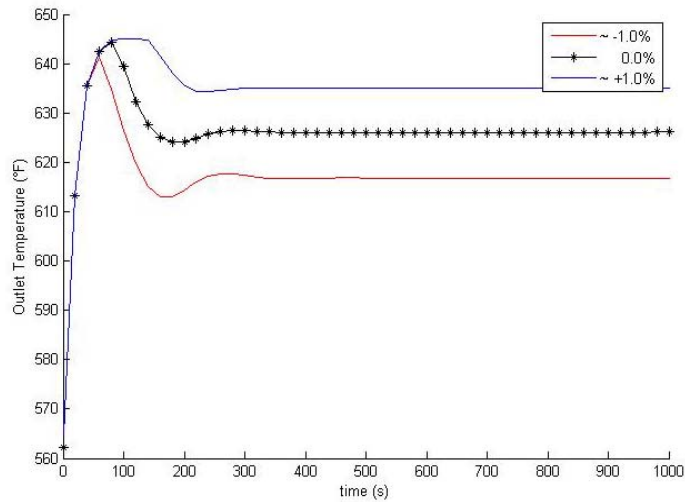


Figure A11. Core Outlet Fluid Temperature as a Function of the Core Coolant Inlet
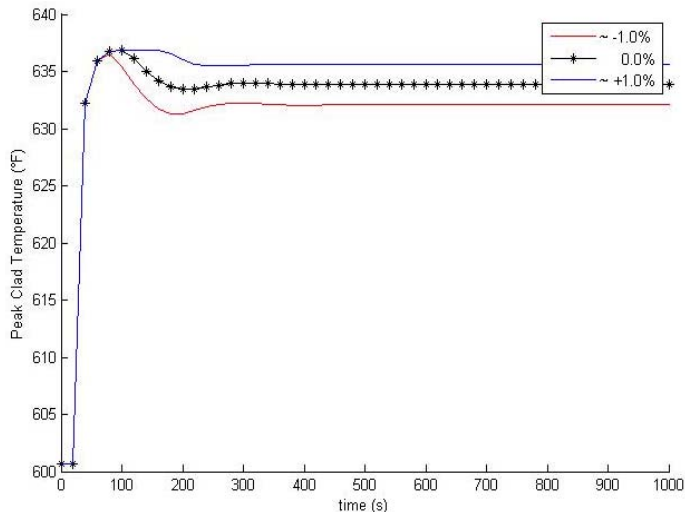
Temperature



Figure A12. Peak Clad Temperature as a Function of the Core Coolant Inlet
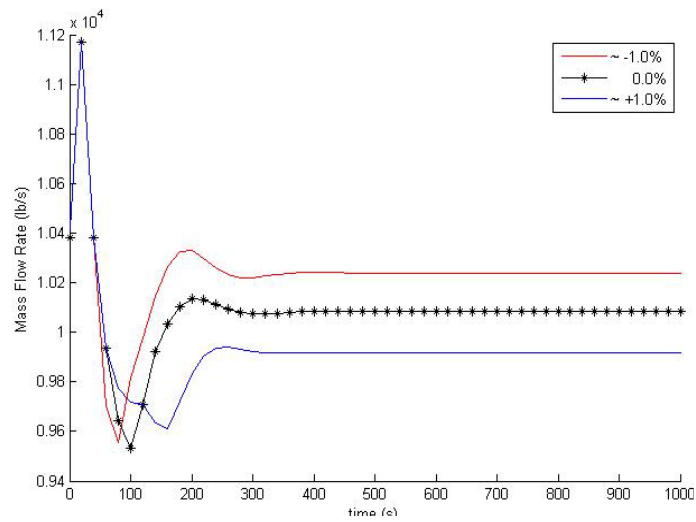
Temperature

Figure A13. Hot Leg Mass Flow Rate as a Function of the Core Coolant Inlet
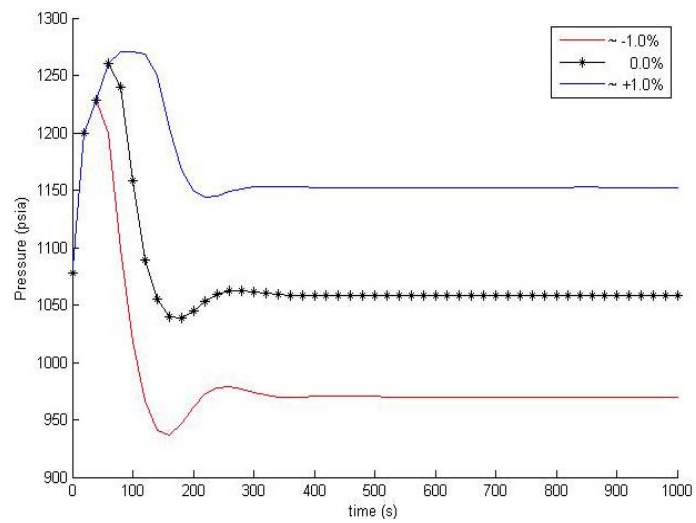
Temperature



Figure A14. Steam Generator Secondary Side Pressure as a Function of the Core

Coolant Inlet Temperature

*Primary Side Reactor Pressure*
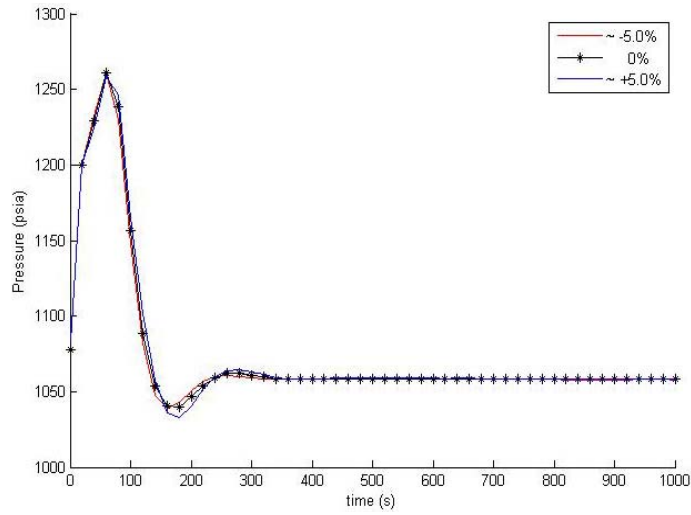


Figure A15. Steam generator secondary side pressure as a function of the primary side
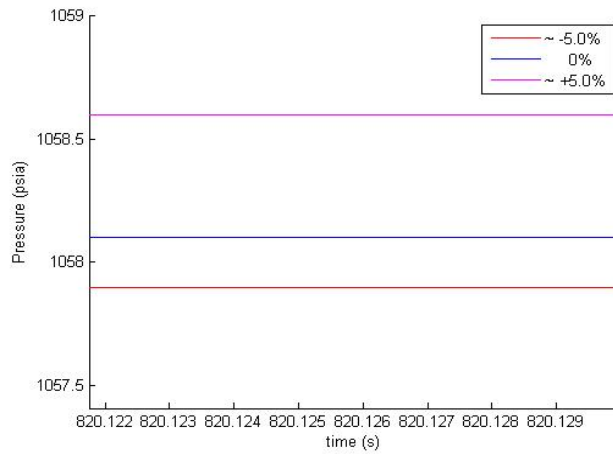
pressure.



Figure A16. Steam Generator Secondary Side Pressure as a Function of the Primary Side

Pressure (Magnified)
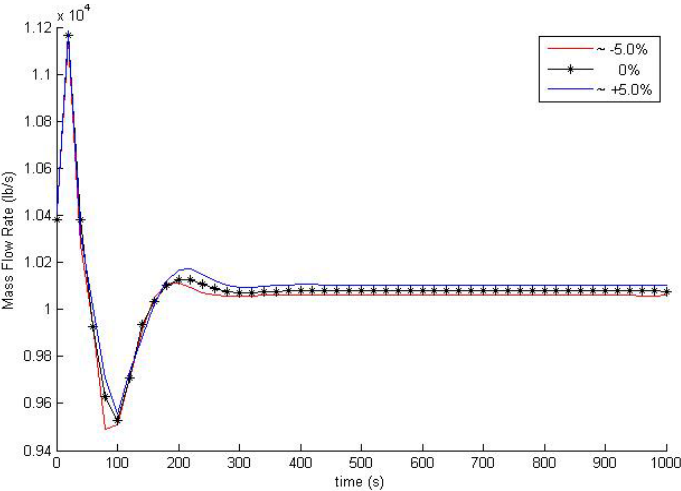
Figure A17. Hot Leg Mass Flow Rate as a Function of the Primary Side Pressure
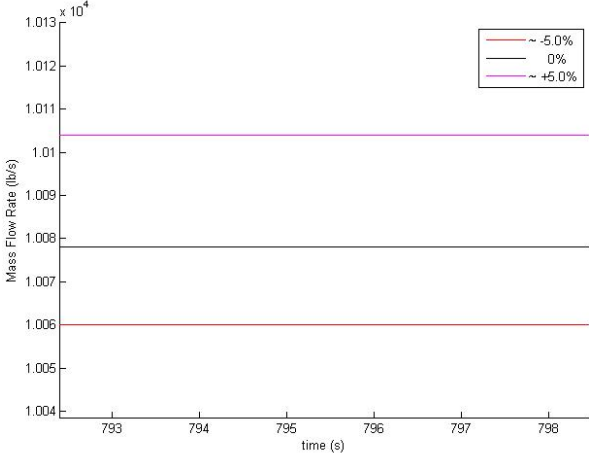


Figure A18. Hot Leg Mass Flow Rate as a Function of the Primary Side Pressure
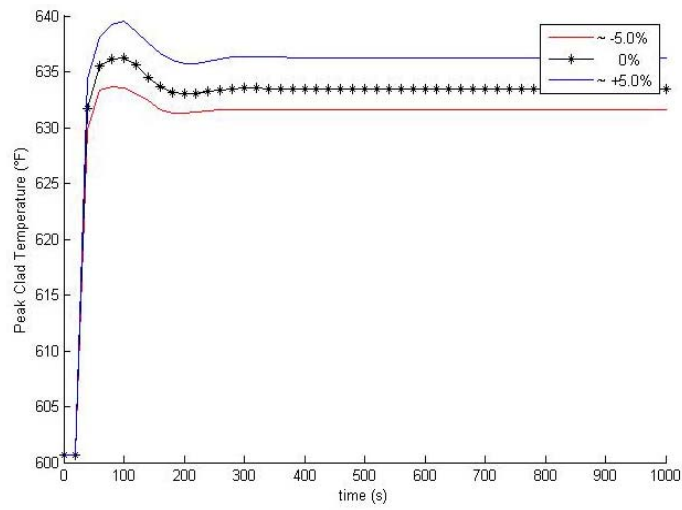
Figure A19. Peak Clad Temperature as a Function of the Primary Side Pressure
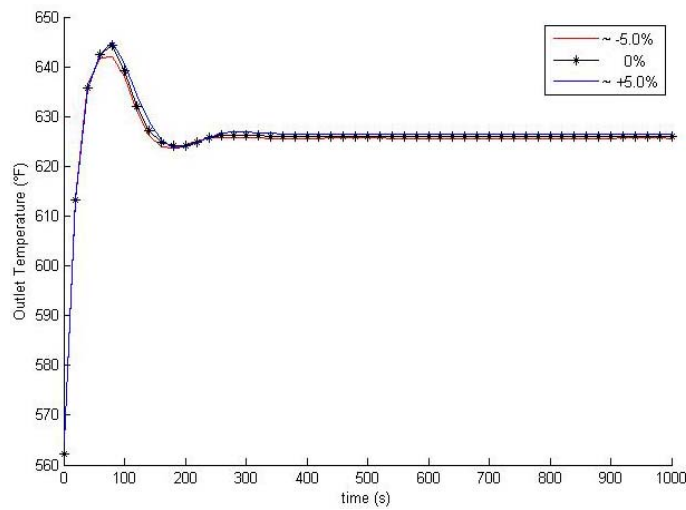


Figure A20. Core Outlet Fluid Temperature as a Function of the Primary Side Pressure

APPENDIX A.2

**Full Break Parameter Plots**
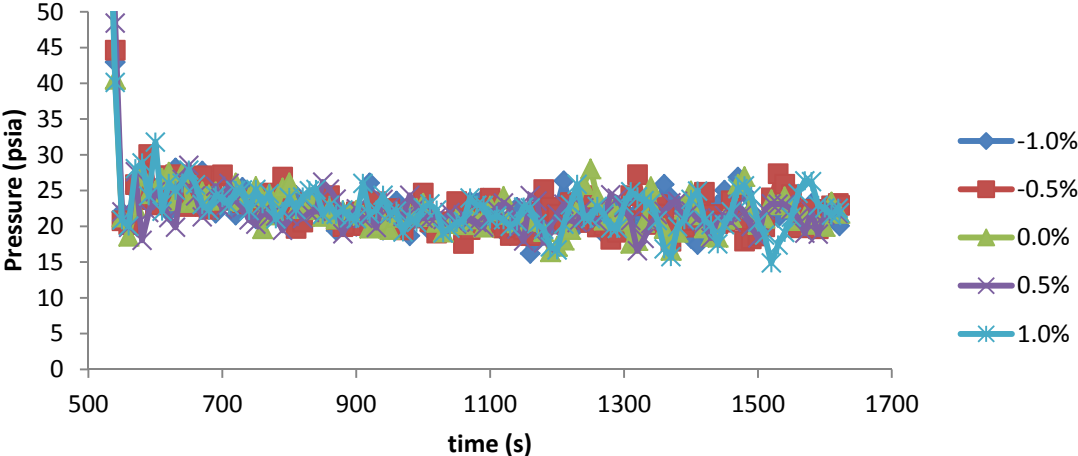


Figure A21. Primary System Pressure Over Time Oscillations Due to Variation of Core
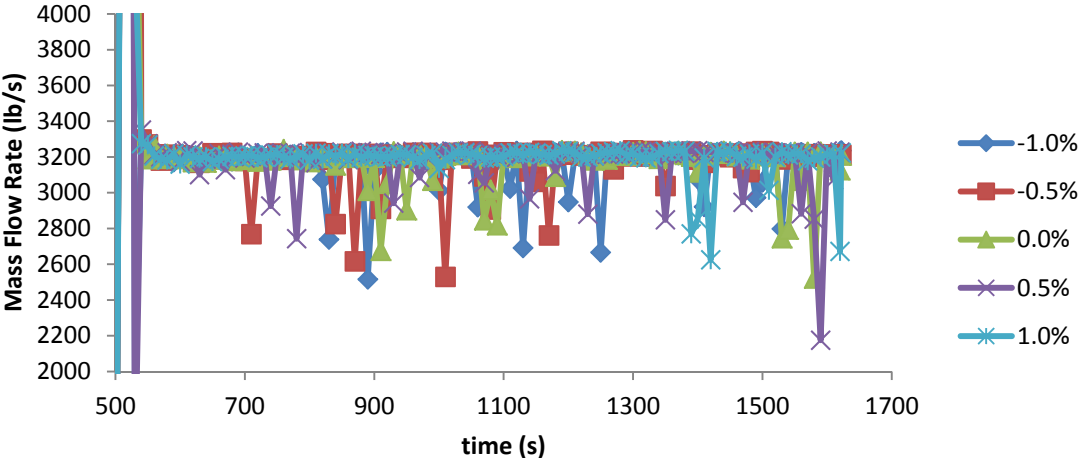
Inlet Temperature



Figure A22. ECCS Total Mass Flow Rate Over Time Oscillations Due to Variation of

Core Inlet Temperature
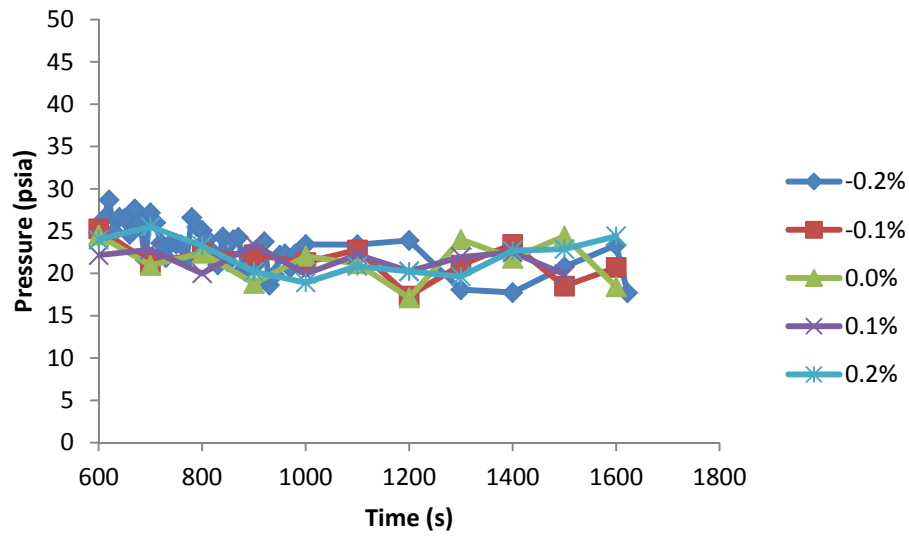
Figure A23. Primary System Pressure Over Time Oscillations Due to Variation of
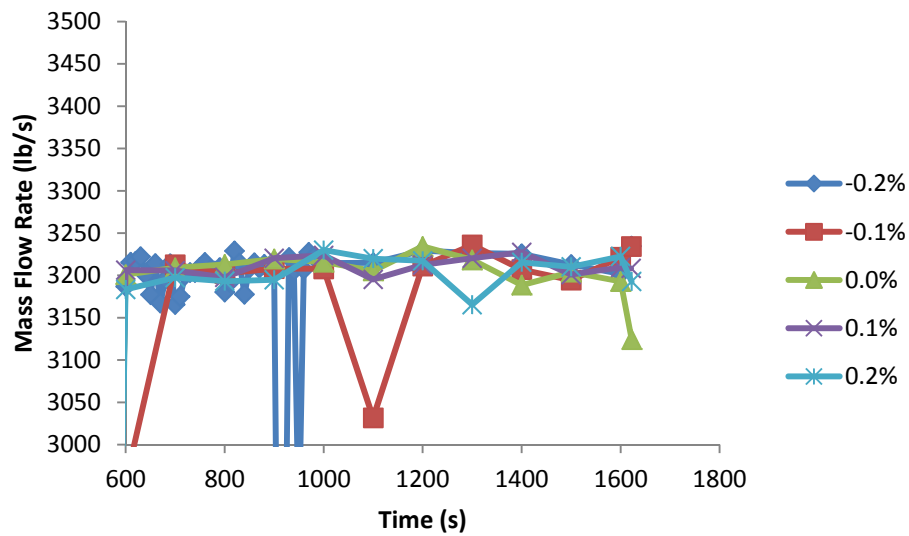
Reactor Power



Figure A24. ECCS Total Mass Flow Rate Over Time Oscillations Due to Variation of

Reactor Power
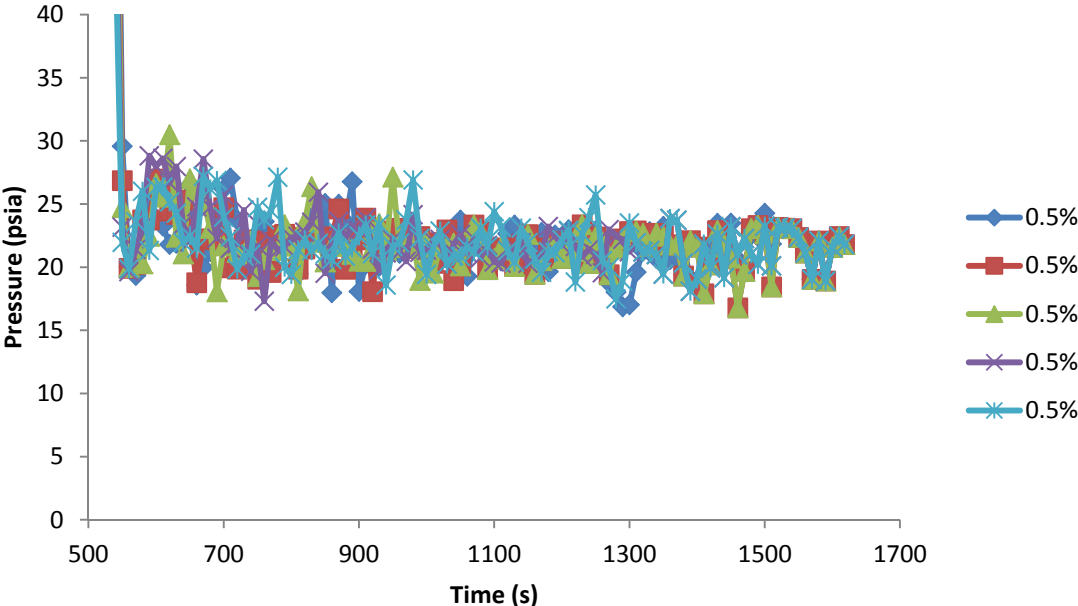
Figure A25. Primary System Pressure Over Time Oscillations Due to Variation of Two Phase Discharge
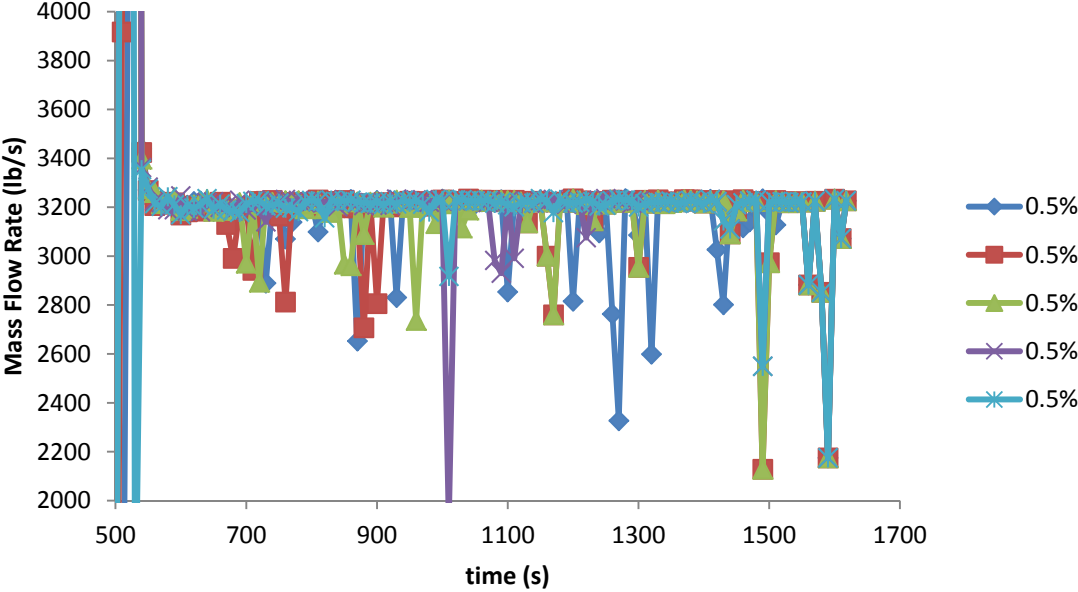


Figure A26. ECCS Total Mass Flow Rate Over Time Oscillations Due to Variation of Two Phase Discharge
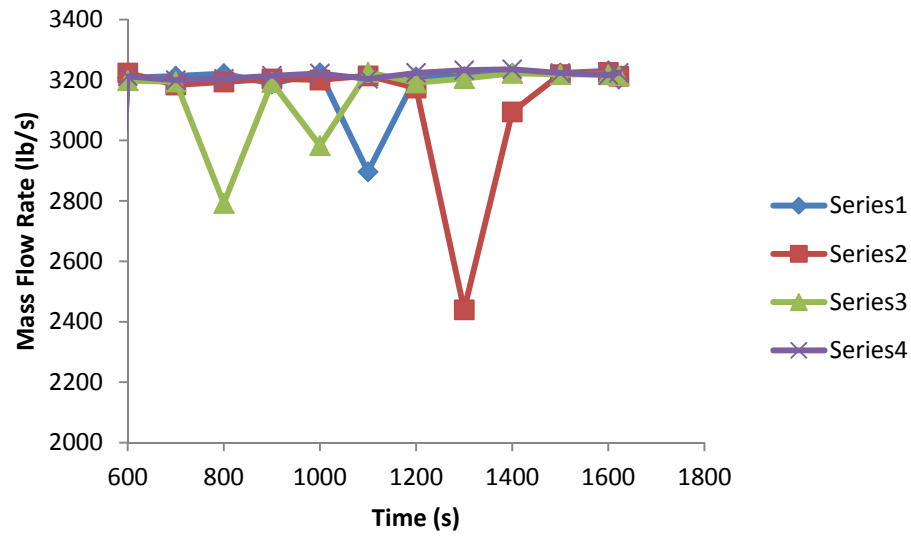
Figure A27. ECCS Total Mass Flow Rate Over Time Oscillations Due to Variation of Sub-cooled Discharge Coefficient
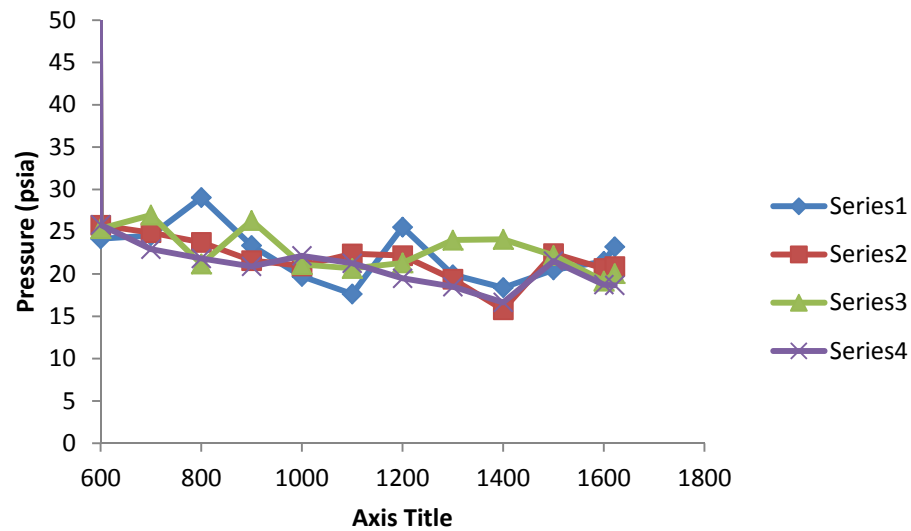


Figure A29. Primary Side Pressure Over Time Oscillations Due to Variation of Sub-cooled Discharge Coefficient
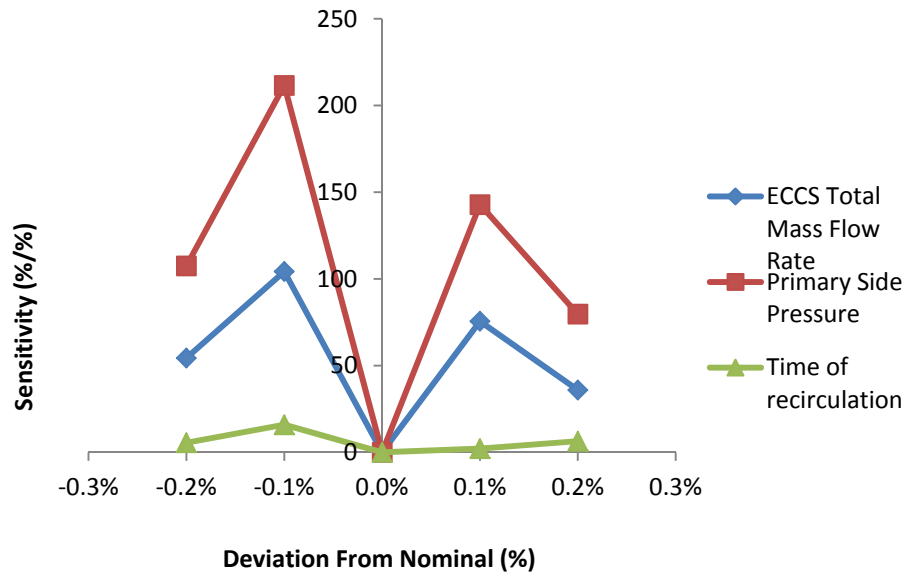
Two Inch Break
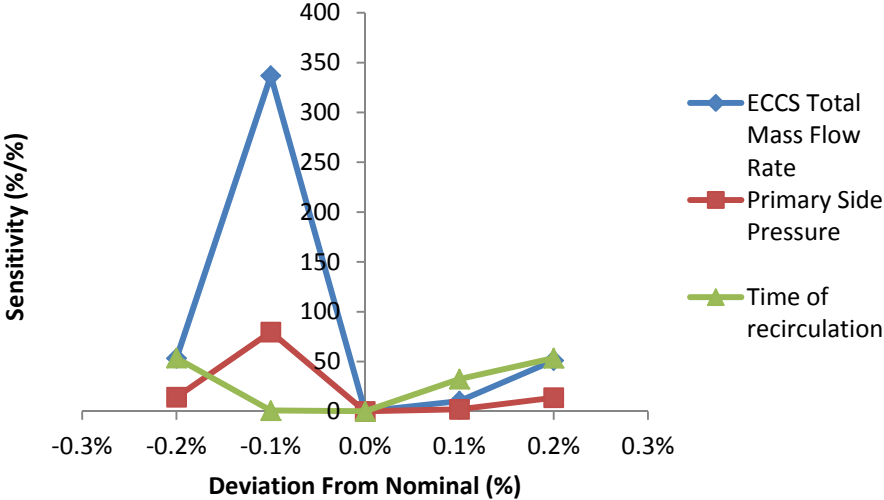


Figure A30. Sensitivity of Selected Output Parameters Due to Reactor Power Variation

for the Two Inch Break
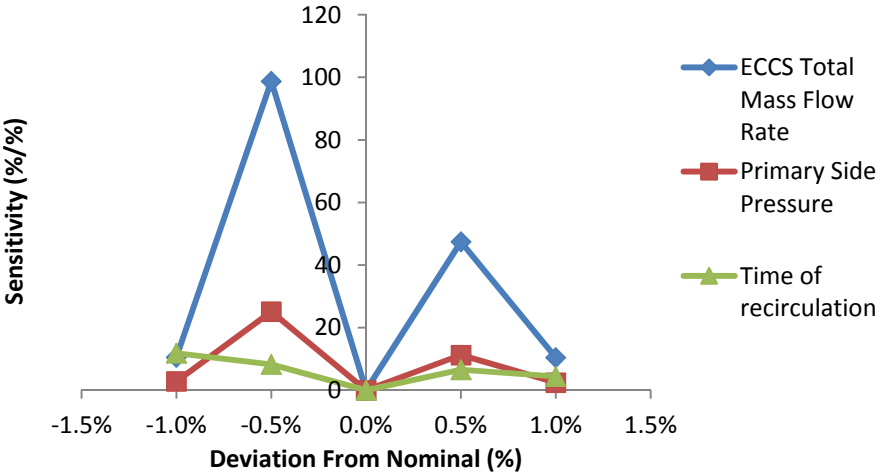


Figure A31. Sensitivity of Selected Output Parameters Due to Core Inlet Temperature

Variation for the Two Inch Break
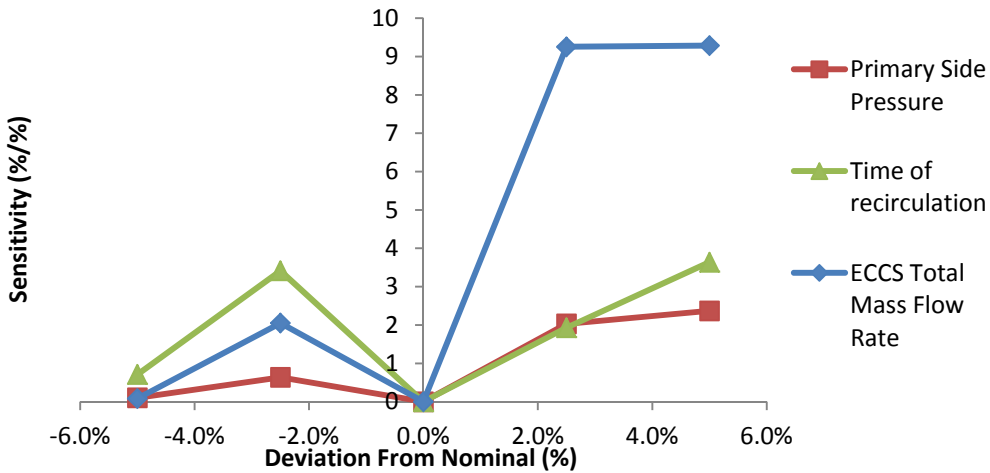
Figure A32. Sensitivity of Selected Output Parameters Due to Break Size Variation for
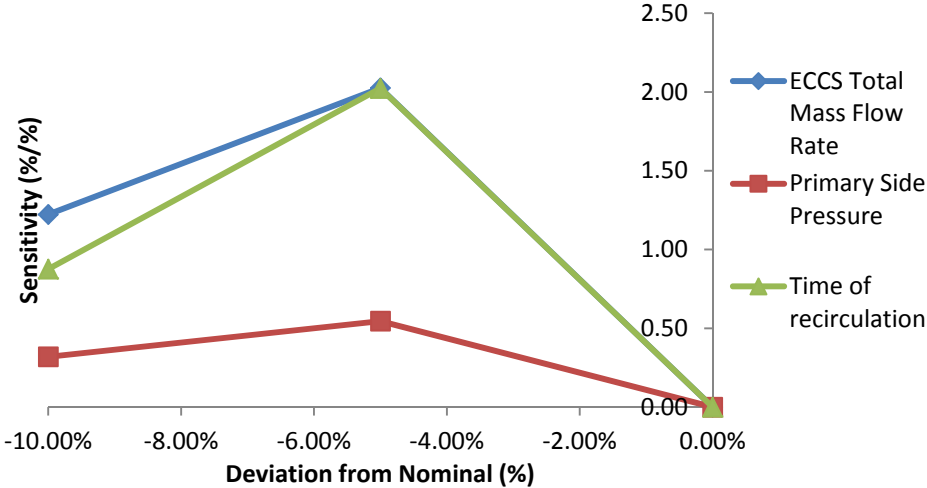
the Two Inch Break



Figure A33. Sensitivity of Selected Output Parameters Due to Sub-cooled Discharge

Variation for the Two Inch Break

*6 Inch Break*



Figure A34. Sensitivity of Selected Output Parameters Due to Reactor Power Variation
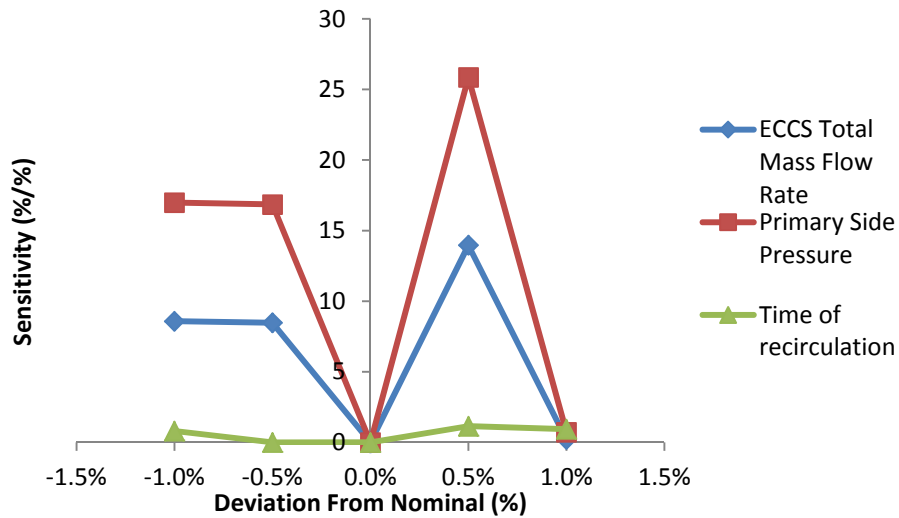
for the Six Inch Break



Figure A35. Sensitivity of Selected Output Parameters Due to Inlet Temperature

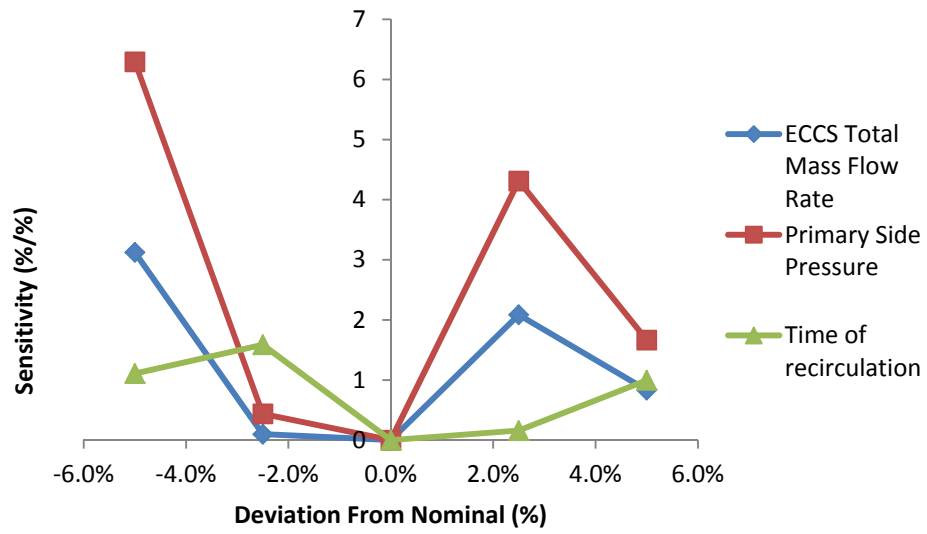Variation for the Six Inch Break

Figure A36. Sensitivity of Selected Output Parameters Due to Break Size Variation for

the Six Inch Break

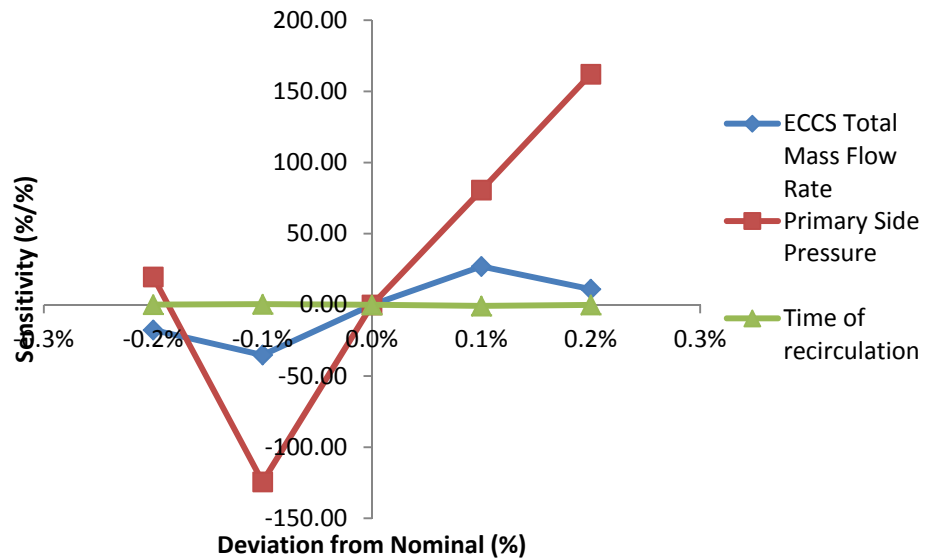*Double Ended Guillotine Break*



Figure A37. Sensitivity of Selected Output Parameters Due to Reactor Power Variation
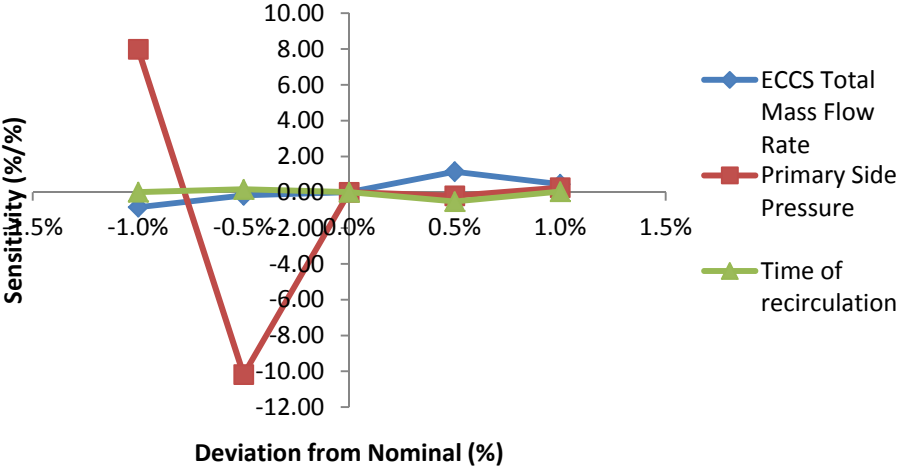
for the Guillotine Break.

Figure A38. Sensitivity of Selected Output Parameters Due to Inlet Temperature

Variation for the Guillotine Break