



ESCUELA TÉCNICA SUPERIOR DE INGENIERÍA (ICAI)
INGENIERO INDUSTRIAL

SIMULACIÓN DE UNA TURBINA DE VAPOR DE UNA CENTRAL NUCLEAR CON EL CÓDIGO TRACE

Autor: Adoración Garrido-Lestache Romero
Director: Prof. Dr. Rafael Macián Juan

Madrid
Mayo 2015

AUTORIZACIÓN PARA LA DIGITALIZACIÓN, DEPÓSITO Y DIVULGACIÓN EN ACCESO ABIERTO (RESTRINGIDO) DE DOCUMENTACIÓN

1º. Declaración de la autoría y acreditación de la misma.

El autor D. ADORACIÓN GARRIDO-LESTACHE , como ALUMNA de la UNIVERSIDAD PONTIFICIA COMILLAS (COMILLAS), **DECLARA**

que es el titular de los derechos de propiedad intelectual, objeto de la presente cesión, en relación con la obra

PROYECTO FIN DE CARRERA: SIMULACIÓN DE UNA TURBINA DE VAPOR DE UNA C. NUCLEAR CON EL CÓDIGO TRACE¹ ,

que ésta es una obra original, y que ostenta la condición de autor en el sentido que otorga la Ley de Propiedad Intelectual como titular único o cotitular de la obra.

En caso de ser cotitular, el autor (firmante) declara asimismo que cuenta con el consentimiento de los restantes titulares para hacer la presente cesión. En caso de previa cesión a terceros de derechos de explotación de la obra, el autor declara que tiene la oportuna autorización de dichos titulares de derechos a los fines de esta cesión o bien que retiene la facultad de ceder estos derechos en la forma prevista en la presente cesión y así lo acredita.

2º. Objeto y fines de la cesión.

Con el fin de dar la máxima difusión a la obra citada a través del Repositorio institucional de la Universidad y hacer posible su utilización de *forma libre y gratuita* (*con las limitaciones que más adelante se detallan*) por todos los usuarios del

¹ Especificar si es una tesis doctoral, proyecto fin de carrera, proyecto fin de Máster o cualquier otro trabajo que deba ser objeto de evaluación académica

repositorio y del portal e-ciencia, el autor **CEDE** a la Universidad Pontificia Comillas de forma gratuita y no exclusiva, por el máximo plazo legal y con ámbito universal, los derechos de digitalización, de archivo, de reproducción, de distribución, de comunicación pública, incluido el derecho de puesta a disposición electrónica, tal y como se describen en la Ley de Propiedad Intelectual. El derecho de transformación se cede a los únicos efectos de lo dispuesto en la letra (a) del apartado siguiente.

3º. Condiciones de la cesión.

Sin perjuicio de la titularidad de la obra, que sigue correspondiendo a su autor, la cesión de derechos contemplada en esta licencia, el repositorio institucional podrá:

- (a) Transformarla para adaptarla a cualquier tecnología susceptible de incorporarla a internet; realizar adaptaciones para hacer posible la utilización de la obra en formatos electrónicos, así como incorporar metadatos para realizar el registro de la obra e incorporar “marcas de agua” o cualquier otro sistema de seguridad o de protección.
- (b) Reproducirla en un soporte digital para su incorporación a una base de datos electrónica, incluyendo el derecho de reproducir y almacenar la obra en servidores, a los efectos de garantizar su seguridad, conservación y preservar el formato. .
- (c) Comunicarla y ponerla a disposición del público a través de un archivo abierto institucional, accesible de modo libre y gratuito a través de internet.²
- (d) Distribuir copias electrónicas de la obra a los usuarios en un soporte digital.³

² En el supuesto de que el autor opte por el acceso restringido, este apartado quedaría redactado en los siguientes términos:

(c) Comunicarla y ponerla a disposición del público a través de un archivo institucional, accesible de modo restringido, en los términos previstos en el Reglamento del Repositorio Institucional

³ En el supuesto de que el autor opte por el acceso restringido, este apartado quedaría eliminado.

4º. Derechos del autor.

El autor, en tanto que titular de una obra que cede con carácter no exclusivo a la Universidad por medio de su registro en el Repositorio Institucional tiene derecho a:

- a) A que la Universidad identifique claramente su nombre como el autor o propietario de los derechos del documento.
- b) Comunicar y dar publicidad a la obra en la versión que ceda y en otras posteriores a través de cualquier medio.
- c) Solicitar la retirada de la obra del repositorio por causa justificada. A tal fin deberá ponerse en contacto con el vicerrector/a de investigación (curiarte@rec.upcomillas.es).
- d) Autorizar expresamente a COMILLAS para, en su caso, realizar los trámites necesarios para la obtención del ISBN.
- d) Recibir notificación fehaciente de cualquier reclamación que puedan formular terceras personas en relación con la obra y, en particular, de reclamaciones relativas a los derechos de propiedad intelectual sobre ella.

5º. Deberes del autor.

El autor se compromete a:

- a) Garantizar que el compromiso que adquiere mediante el presente escrito no infringe ningún derecho de terceros, ya sean de propiedad industrial, intelectual o cualquier otro.
- b) Garantizar que el contenido de las obras no atenta contra los derechos al honor, a la intimidad y a la imagen de terceros.

c) Asumir toda reclamación o responsabilidad, incluyendo las indemnizaciones por daños, que pudieran ejercitarse contra la Universidad por terceros que vieran infringidos sus derechos e intereses a causa de la cesión.

d) Asumir la responsabilidad en el caso de que las instituciones fueran condenadas por infracción de derechos derivada de las obras objeto de la cesión.

6º. Fines y funcionamiento del Repositorio Institucional.

La obra se pondrá a disposición de los usuarios para que hagan de ella un uso justo y respetuoso con los derechos del autor, según lo permitido por la legislación aplicable, y con fines de estudio, investigación, o cualquier otro fin lícito. Con dicha finalidad, la Universidad asume los siguientes deberes y se reserva las siguientes facultades:

a) Deberes del repositorio Institucional:

- La Universidad informará a los usuarios del archivo sobre los usos permitidos, y no garantiza ni asume responsabilidad alguna por otras formas en que los usuarios hagan un uso posterior de las obras no conforme con la legislación vigente. El uso posterior, más allá de la copia privada, requerirá que se cite la fuente y se reconozca la autoría, que no se obtenga beneficio comercial, y que no se realicen obras derivadas.

- La Universidad no revisará el contenido de las obras, que en todo caso permanecerá bajo la responsabilidad exclusiva del autor y no estará obligada a ejercitar acciones legales en nombre del autor en el supuesto de infracciones a derechos de propiedad intelectual derivados del depósito y archivo de las obras. El autor renuncia a cualquier reclamación frente a la Universidad por las formas no ajustadas a la legislación vigente en que los usuarios hagan uso de las obras.

- La Universidad adoptará las medidas necesarias para la preservación de la obra en un futuro.

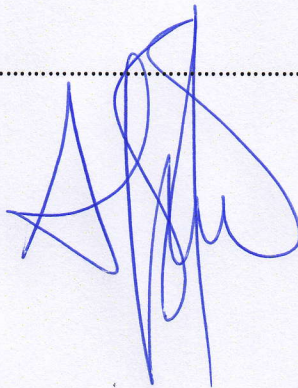
b) Derechos que se reserva el Repositorio institucional respecto de las obras en él registradas:

- retirar la obra, previa notificación al autor, en supuestos suficientemente justificados, o en caso de reclamaciones de terceros.

Madrid, a 26 de Mayo de 2015

ACEPTA

Fdo.....



Proyecto realizado por el alumno/a:

Adoración Garrido-Lestache Romero

Fdo.: 

Fecha: 15/05/2015

Autorizada la entrega del proyecto cuya información no es de carácter
confidencial

EL DIRECTOR DEL PROYECTO

Rafael Macián Juan

Fdo.: 

Fecha: 15/05/2015

Vº Bº del Coordinador de Proyectos

Fdo.:

Fecha://



ESCUELA TÉCNICA SUPERIOR DE INGENIERÍA (ICAI)
INGENIERO INDUSTRIAL

SIMULACIÓN DE UNA TURBINA DE VAPOR DE UNA CENTRAL NUCLEAR CON EL CÓDIGO TRACE

Autor: Adoración Garrido-Lestache Romero
Director: Prof. Dr. Rafael Macián Juan

Madrid
Mayo 2015

Resumen en español

SIMULACIÓN DE UNA TURBINA DE VAPOR DE UNA CENTRAL NUCLEAR CON EL CÓDIGO TRACE

Autor: Adoración Garrido-Lestache Romero

Director: Prof. Dr. Rafael Macián Juan

Entidad Colaboradora: TUM – Technische Universität München

RESUMEN DEL PROYECTO

Introducción:

Una Central Nuclear está compuesta por un primer circuito, en el que se encuentra el reactor nuclear, y por un segundo circuito, el cual es similar al de una central térmica convencional.

Desde el punto de vista de la seguridad nuclear, el primer circuito de una central es el de mayor importancia para los análisis, y por lo tanto, los modelos desarrollados de este circuito son muy sofisticados. Por otro lado, como el análisis de seguridad en el segundo circuito carece de tanta importancia, los modelos de las turbinas desarrollados para simular su comportamiento bajo condiciones normales y adversas son muy simples.

El objetivo de este proyecto es desarrollar un modelo de la turbina de vapor de una central nuclear que proporcione resultados de funcionamiento realistas bajo condiciones normales y bajo el efecto de transitorios. El código TRACE, que fue desarrollado por el gobierno de los Estados Unidos para modelar el funcionamiento de centrales nucleares, es la herramienta utilizada para el desempeño de este proyecto.

Un esquema del circuito a simular fue proporcionado al principio del trabajo y ha sido usado como base para el desarrollo del modelo. El modelo final del circuito está compuesto por: una turbina de alta y otra de baja presión, un separador, un calentador, una válvula de seguridad, y condensadores.

Metodología:

Los objetivos a conseguir fueron:

- Obtener condiciones de presión realistas a la entrada de la turbina.
- Reproducir una caída de presión realista a través de la turbina.

Para el desarrollo del modelo, la metodología seguida ha sido la siguiente:

1. Entender y comprender el código TRACE.

En primer lugar se revisó el manual del código con el objetivo de entender su funcionamiento y haciendo una primera selección de los elementos que se utilizaron para el desarrollo del modelo.

2. Estudiar que factores afectan a la caída de presión.

Teniendo en cuenta, que los dos objetivos menores que este trabajo pretende conseguir estaban basados en las presiones del sistema, fue necesario hacer un análisis de los factores afectan a estas y cómo. Los resultados obtenidos revelaron que el elemento del código que más afecta a la caída de presión dentro de los componentes del modelo es el “KFAC”. Dentro de cada componente se define un “KFAC” específico para cada borde.

3. Desarrollar los componentes del sistema y el modelo completo.

Cada elemento del sistema ha sido desarrollado independientemente. Con la ayuda del manual, se decidió qué elementos proporcionados por el código TRACE eran los más apropiados para el desarrollo de cada componente. Una vez diseñados, todos los componentes fueron conectados entre sí en un modelo más grande. Los componentes diseñados para el sistema fueron: dos turbinas (una de alta y otra de baja), un separador, un calentador, una válvula de seguridad, y un condensador.

4. Simulación del estado estacionario.

Una vez completado el modelo y perfeccionado, se ha simulado su funcionamiento bajo condiciones normales. Para conseguir resultados óptimos, la geometría y los “KFAC” de los componentes fueron modificados minuciosamente.

5. Simulación de transitorios.

Finalmente se ejecutó una simulación del funcionamiento del modelo bajo condiciones adversas (bajadas repentinas en la presión del sistema). Fueron introducidos tres transitorios distintos.

6. Recopilación de los datos de las simulaciones.

Una vez terminadas las simulaciones se recopilaron y analizaron todos los resultados.

Resultados:

Los modelos simplificados de las turbinas de alta y baja presión fueron desarrollados para probar la efectividad del modelo de las turbinas en TRACE utilizando las condiciones de contorno de diseño. Después de simular ambos modelos, los resultados fueron recopilados y comparados con los datos reales medidos en la planta. El análisis de los resultados se ha focalizado en la caída de presión a lo largo de la turbina, ya que el elemento FILL impone las condiciones a la entrada de la turbina. La comparación entre los resultados de la simulación y los datos reales de medida en la planta fue muy positiva.

Después de analizar los resultados de los modelos simplificado, se implementó el modelo completo. Los resultados obtenidos de la simulación del estado estacionario en el modelo completo no fueron tan buenos como los obtenidos de la simulación de los modelos simplificados. La diferencia que existe entre los resultados de ambos modelos es debido al efecto causado por los demás componentes del sistema (separador y calentador).

Después de obtener buenos resultados de las simulaciones de los modelos en estado estacionario, se simularon tres transitorios distintos. Los resultados obtenidos de la simulación de los transitorios han sido muy positivos. El patrón descrito por el transitorio introducido en el sistema ha sido seguido por todos los componentes.

Teniendo en cuenta los objetivos del proyecto (obtener condiciones de presión realistas en las turbinas) y que la comparación entre los datos reales y los resultados de las simulaciones es muy positiva, se ha considerado que el modelo desarrollado es válido para el estado estacionario y para la simulación de transitorios.

Resumen en inglés

SIMULATION OF A NPP STEAM TURBINE WITH THE BEST-ESTIMATE CODE TRACE

Autor: Adoración Garrido-Lestache Romero

Director: Prof. Dr. Rafael Macián Juan

Entidad Colaboradora: TUM – Technische Universität München

PROYECT SUMMARY

Introduction:

A NPP is composed of a primary side, which contains the nuclear reactor, and a secondary side, which is similar to a conventional thermal power station.

From a safety analysis point of view, the primary side of a NPP is the most important; hence, detailed models of the primary side exist in order to run simulations under different conditions. In contrast, since the safety analysis of a NPP secondary side is less important, the turbine simulation models are very simple.

The objective of this Master's Thesis is to develop a NPP secondary side model that provides realistic results under normal and transient operation conditions. The best-estimate reactor system code used to achieve this objective is TRACE, which was developed by the U.S. Nuclear Regulatory Commission for analysing steady-state behaviour, transients, Loss of Coolant Accidents (LOCA's), and other accidents in light water reactors (PWR's and BWR's).

At the beginning of this work, a basic diagram of the system to simulate was provided, which has been used as the basis for the model development. The final model includes the following main components: high-pressure turbine, separator, over-heater, low-pressure turbine, and condensers.

Methodology:

The goals to be achieved here were:

- Reach realistic conditions at the turbine inlet.
- Reproduce a realistic pressure drop through the turbine.

In order to develop the model, the methodology that has been followed was:

1. Comprehension and understanding of the best-estimate code TRACE.

The code manual was reviewed in order to understand the basic concepts necessary to work with the code. A first selection of the main components that would be used in the model was made.

2. Make an analysis of the main factors that affect the pressure drop inside the turbine components.

Taking into account that the main objective of this work is to reach a realistic pressure drop inside the turbine components, an analysis of the most significant factors that affect the pressure drop was necessary. The results obtained from the simulation showed that the element that mostly affects the pressure drop inside the components is the KFAC array. Inside every model component the KFAC array must be input.

3. Development of the system components and of the complete model.

Each element of the system has been developed separately. All the elements used for the development of the model were provided by the code TRACE. After designing each element, all of them were implemented inside a bigger model. The components designed for the system were: Low-Pressure and High-Pressure Turbine, a separator, an over-heater, a security stop valve, and a condenser.

4. Steady-State Simulation.

Once the model was completed and improved, a steady-state simulation was run. In order to reach optimal results, the geometry and the KFAC array of all the components were manually tuned.

5. Transient Simulation.

Finally a transient simulation was run (adverse conditions: smooth pressure drop). Three different transients were run.

6. Compilation of simulation results.

After running the simulations all the results were collected and analysed.

Results:

The simplified HP- and LP-Turbine models were developed in order to test the capabilities of the turbine TRACE model using boundary conditions from the turbine design data. After simulating both models, the results were collected and compared to the real turbine data. The data analysis focussed on the pressure drop inside the turbine, since the FILL component imposed the turbine inlet conditions. The simulation results and the real data show good agreement.

After analysing the simplified model data, the complete model was implemented.

The results are not as good as those of the simplified models. The slightly difference between the simulation and the real data is caused by the fact that the code has to compensate for the losses from the other components such as separator and over-heater.

After optimal results for the steady-state simulation were achieved, three different transient models were simulated. The results obtained from the transient simulations are very remarkable.

The objectives to achieve were reaching realistic conditions at both turbine inlets and reproducing a realistic pressure drop inside the turbine. A comparison between the simulation and the real data at both turbines reveals that both the inlet pressure and the pressure drop in the turbines are near to reality. This was valid for both the steady state and transient simulations.

Master's Thesis

Adoración Garrido-Lestache Romero

Simulation of a NPP steam turbine with the best-estimate system code

TRACE

Betreuer TUM: Prof. Dr. Rafael Macián-Juan
Betreuer: Dipl.-Ing. Dan-Ovidiu Melinte
Ausgegeben: 01.11.14
Abgegeben: 30.04.15

Acknowledgements

This work is part of the Research Project No. 1501441 - “Plant specific dynamic modelling of LWR turbo generator sets by adjustment of general models with measurement data” which has been financed by the German Federal Ministry for Economic Affairs and Energy (BMWi).

It would not have been possible to write this Thesis without the help and support of many people around me.

First of all, I would like to express my deepest gratitude to Dipl. Ing. Dan-Ovidiu Melinte for the guidance provided during the course of my work, his good advice, and all the time he has dedicated to assist me. Without his help this work would have not been possible. I would also like to thank Prof. Dr. Macián-Juan for his many valuable suggestions and his continuous support, and also for bringing me the opportunity to write my Master’s Thesis in the Department of Nuclear Engineering. I would also like to thank all the members of the department, for making my time here very pleasant.

I would like to take the chance to thank my family, for all their support, and for always encouraging me, but especially, for being the sustenance of my life.

Finally, I thank all my friends with whom I have shared my wonderful experience in Munich, especially to my Ludwigs-Mitbewohners and my “ICAI’s”. But, above all, I would like to thank my “PC-Raum-Team” for making these six months unforgettable.

Declaration

I hereby declare that I have made this thesis independently and without the help of others. Ideas and quotes that I have taken over directly or indirectly from other sources are marked as such. This thesis has not been submitted to any inspection authority in the same or similar form and has not been published.

I hereby agree that this thesis can be made available to the public by Lehrstuhl für Nukleartechnik.

Munich, April the 30th, 2015

Adoración Garrido-Lestache Romero.

Table of contents

Acknowledgements.....	III
Declaration.....	V
Table of contents.....	VII
List of figures	XI
List of tables.....	XV
List of acronyms	XVII
Abstract	XIX
1. Introduction	1
2. Theoretical Background.....	3
2.1 Nuclear Power Plant	3
2.1.1 Pressurized Water Reactor.....	4
2.2 TRACE.....	8
2.2.1 List of Components:.....	8
2.2.2 FRIC and K-FACTOR.....	11
2.3 System description.....	12
2.3.1 System nominal data	13
3. TRACE Model Components.....	17
3.1 Valve	17
3.2 HP-Turbine.....	19
3.3 Separator.....	24
3.4 Over-Heater	24
3.5 LP-Turbine.....	25

3.6 Condenser	29
3.7 Mass Flow Distribution	30
3.8 Complete Model	31
4. Steady-State: Simulation & Results	33
4.1 FILL	33
4.2 Results	33
4.2.1 Simplified HP-Turbine model	34
4.2.2 Simplified LP-Turbine model	36
4.2.3 Complete Model	39
4.3 K-Factors	47
5. Transient: Simulation & Results	49
5.1 Modifications	49
5.1.1 FILL	49
5.1.2 NFRC1 NAMELIST variable	49
5.2 Transient 1: Definition and results	49
FILL Table	50
HP-Turbine	50
LP-Turbine	52
5.3 Transient 2: Definition and results	54
FILL Table	54
HP-Turbine	55
LP-Turbine	57
5.4 Transient 3: Definition and results	59
FILL Table	59
HP-Turbine	60
LP-Turbine	62
6. Conclusions	65
7. Possible future improvements	67
Optimization of the KFAC arrays	67
HEATR component implementation	67

Complete the model closing the loop..... 68

Bibliography 71

List of figures

Figure 2.1: Basic scheme of a NPP (PWR)	3
Figure 2.2: Chain Reaction.....	4
Figure 2.3: Steam generator and turbine of a PWR.....	5
Figure 2.4: Rankine Cycle	6
Figure 2.5: BREAK component	8
Figure 2.6: FILL component	9
Figure 2.7: PIPE component	9
Figure 2.8: TURB component nodalization.....	10
Figure 2.9: VALVE component noding diagram.....	11
Figure 2.10: System scheme.....	13
Figure 3.1: VALVE-TRIP component.....	17
Figure 3.2: TRIP Setpoint Values	19
Figure 3.3: HP-Turbine Model.....	20
Figure 3.4: Single HP-Turbine	20
Figure 3.5: Separator	24
Figure 3.6: LP-Turbine model.....	25
Figure 3.7: Condenser.....	29
Figure 3.8: Complete Model.....	31
Figure 4.1: Modelled HP-Turbine.....	34
Figure 4.2: Graph-Simulated vs. Real HP-Turbine Pressure.....	35

Figure 4.3: Graph-Simulated vs. Real HP-Turbine Pressure Drop per stage	36
Figure 4.4: LP-Turbine simplified model.....	36
Figure 4.5: Graph-Simulated vs. Real LP-Turbine Pressure	38
Figure 4.6: Graph-Simulated vs. Real LP-Turbine Pressure Drop per stage	38
Figure 4.7: Graph-Simulated vs. Real HP-Turbine Pressure	39
Figure 4.8: Graph-Simulated vs. Real HP-Turbine Outlet Pressure	40
Figure 4.9: Graph-Simulated vs. real HP-Turbine pressure drop	40
Figure 4.10: Graph-Simulated HP-Turbine Mass Flow	41
Figure 4.11: Graph-Simulated Separator Mass Flow	42
Figure 4.12: Graph-Simulated Separator Pressure Drop.....	42
Figure 4.13: Graph-Simulated Over-Heater Gas Temperature at cell 2.....	43
Figure 4.14: Graph-Simulated vs. Real LP-Turbine Pressure	44
Figure 4.15: Graph-Simulated vs. Real LP-Turbine Pressure Drop per Stage	44
Figure 4.16: Graph-Simulated LP-Turbine Mass Flow	45
Figure 4.17: Graph-Simulated Condenser Mass Flow.....	45
Figure 4.18: TRIP Setpoint Values.....	46
Figure 4.19: Stop Valve Operation (normalized values)	46
Figure 4.20: TURB component KFAC analysis.....	47
Figure 5.1: HP-Turbine inlet mass flow	50
Figure 5.2: Real vs. Simulated HP-Turbine Inlet pressure	51
Figure 5.3: Real vs. Simulated HP-Turbine pressure	51
Figure 5.4: HP-Turbine Extraction mass flow	52
Figure 5.5: HP-Turbine Extraction pressure	52
Figure 5.6: LP-Turbine inlet mass flow.....	52
Figure 5.7: Real vs. Simulated LP-Turbine inlet pressure	53
Figure 5.8: Real vs. Simulated LP-Turbine pressure	53

Figure 5.9: Real vs. Simulated LP-Turbine outlet pressure	54
Figure 5.10: HP-Turbine inlet mass flow.....	55
Figure 5.11: Real vs. Simulated HP-Turbine inlet pressure.....	56
Figure 5.12: Real vs. Simulated HP-Turbine pressure.....	56
Figure 5.13: HP-Turbine Extraction mass flow.....	57
Figure 5.14: HP-Turbine Extraction pressure.....	57
Figure 5.15: LP-Turbine inlet mass flow	57
Figure 5.16: Real vs. Simulated LP-Turbine inlet pressure.....	58
Figure 5.17: Real vs. Simulated LP-Turbine pressure.....	58
Figure 5.18: Real vs. Simulated LP-Turbine pressure.....	59
Figure 5.19: Real vs. Simulated HP-Turbine inlet pressure.....	60
Figure 5.20: Real vs. Simulated HP-Turbine inlet pressure.....	60
Figure 5.21: Real vs. Simulated HP-Turbine pressure.....	61
Figure 5.22: HP-Turbine Extraction mass flow.....	61
Figure 5.23: HP-Turbine Extraction pressure.....	61
Figure 5.24: LP-Turbine inlet mass flow	62
Figure 5.25: Real vs. Simulated LP-Turbine inlet pressure.....	63
Figure 5.26: Real vs. Simulated LP-Turbine pressure.....	63
Figure 5.27: Real vs. Simulated LP-Turbine outlet pressure	64

List of tables

Table 2.1: HP-Turbine nominal data.....	13
Table 2.2: Separator nominal data.....	13
Table 2.3: Extraction nominal data.....	14
Table 2.4: Over-Heater nominal data.....	14
Table 2.5: LP-Turbine nominal data	14
Table 2.6: Condenser nominal data	14
Table 3.1: VALVE initial conditions.....	18
Table 3.2: VALVE description.....	18
Table 3.3: HP-Turbine geometry data	20
Table 3.4: TURB components 13-15 geometry	22
Table 3.5: TURB components 13-15 geometry 2	22
Table 3.6: TURB component 13-17 initial conditions.....	23
Table 3.7: TURB component 15-19 initial conditions.....	23
Table 3.8: BREAK 14-16-18-20 initial conditions.....	23
Table 3.9: HTSTR boundary conditions	25
Table 3.10: LP-Turbine geometry data.....	26
Table 3.11: TURB component 35-39 geometry	26
Table 3.12: TURB component 35-39 geometry 2	27
Table 3.13: TURB component 35 initial conditions.....	27
Table 3.14: TURB component 36 initial conditions.....	27
Table 3.15: TURB component 37 initial conditions.....	28
Table 3.16: TURB component 38 initial conditions.....	28

Table 3.17: TURB component 39 initial conditions	28
Table 3.18: BREAK components 30-34 initial conditions	28
Table 3.19: HTSTR boundary conditions.....	29
Table 4.1: Steady-State FILL description	33
Table 4.2: HP-Turbine simplified model boundary conditions.....	34
Table 4.3: HP-Turbine simplified model results	35
Table 4.4: LP-Turbine simplified model boundary conditions.....	36
Table 4.5: LP-Turbine simplified model results.....	37
Table 4.5: Over-Heater outlet conditions	43
Table 4.6: KFAC array	47
Table 4.7: KFAC analysis results	48
Table 5.1: Transient 1 FILL description.....	50
Table 5.2: Transient 2 FILL description.....	54
Table 5.3: Transient 3 FILL description.....	59

List of acronyms

A	Area
$A_{\text{flow_inlet}}$	Inlet Flow Area
$A_{\text{flow_outlet}}$	Outlet Flow Area
A_n	Area value used for normalization
A_{rotor}	Rotor Area
A_{total}	Total Area
B_{inlet}	Blade Inlet Height
B_{outlet}	Blade Outlet Height
BWR	Boiled Water Reactor
D_{inlet}	Inlet Mean Diameter
D_{outlet}	Outlet Mean Diameter
D_{rotor}	Rotor Diameter
eV	Electron volt
HP	High Pressure
HWR	Heavy Water Reactor
h	Enthalpy
LOCA	Loss of Coolant Accident
LP	Low Pressure
Mf	Mass Flow
M_{fn}	Mass flow value used for normalization
MPa	Mega Pascal
NFF	Friction Factor Correlation Option
NPP	Nuclear Power Plant
P	Pressure
PWR	Pressurized Water Reactor
P_n	Pressure value used for normalization

XVIII

Q_A	Added Thermal Energy
T	Temperature
T_n	Temperature value used for normalization
U	Uranium
W_T	Turbine Work
W_P	Pump Work
W_N	Net Work

Abstract

The main objective of this Master's Thesis was to simulate the operation of a NPP steam turbine under normal, as well as transient, conditions. To achieve this objective, a simulation model of a NPP secondary loop was developed, using the TRACE system code as a working tool.

The motivation for carrying out this Master's Thesis was that, since from a safety analysis point of view the turbine system has a lower priority, the models normally used for the simulation of a NPP steam-turbine operation were simplified models, which did not provide optimal results under transient conditions.

The goals to be achieved here were:

- Reach realistic conditions at the turbine inlet.
- Reproduce a realistic pressure drop through the turbine.

The model was developed and thoroughly improved until the steady-state simulation results were optimal. Also, three different transient models were set up and simulated, which provided good results.

This document includes the necessary theoretical background to understand the developed model, as well as a detailed description of the components modelling. The thesis also includes an analysis of the results obtained from the simulations.

1. Introduction

A NPP is composed of a primary side, which contains the nuclear reactor, and a secondary side, which is similar to a conventional thermal power station (see figure 2.1) [1].

From a safety analysis point of view, the primary side of a NPP is the most important; hence, detailed models of the primary side exist in order to run simulations under different conditions. In contrast, since the safety analysis of a NPP secondary side is less important, the turbine simulation models are very simple.

The objective of this Master's Thesis is to develop a NPP secondary side model that provides realistic results under normal and transient operation conditions. The best-estimate reactor system code used to achieve this objective is TRACE, which was developed by the U.S. Nuclear Regulatory Commission for analysing steady-state behaviour, transients, Loss of Coolant Accidents (LOCA's), and other accidents in light water reactors (PWR's and BWR's).

At the beginning of this work, a basic diagram of the system to simulate was provided, which has been used as the basis for the model development. The final model includes the following main components: high-pressure turbine, separator, over-heater, low-pressure turbine, and condensers. The work done has followed these steps:

1. Analysis of the geometry and k-factors effect on the pressure drop.
2. Development of the HP- and LP-Turbine simplified model.
3. Insertion of the HP- and LP-Turbine in the complete model.
4. Development of the separator, over-heater, and emergency stop valve.
5. Improvement of the model.
6. Insertion of the condenser.
7. Steady-state simulation.
8. Model set up and transient simulation.
9. Extraction of the results, comparison and conclusions.

This document includes a theoretical section, which contains a brief description of a typical NPP operation, and the main characteristics of the TRACE system code. The thesis also includes a detailed description of the model components setup, as well as an analysis of the results obtained from the simulations.

2. Theoretical Background

This chapter includes an overview of the main characteristics of a Pressurized Water Reactor, an introduction to the best-estimate system code used for the simulations, and a brief explanation of the simulated system, which corresponds to the secondary side of a PWR.

2.1 Nuclear Power Plant

A nuclear power plant operates as a conventional thermal power station, yet using a nuclear reactor as heat source. A NPP is divided in two major areas: “nuclear area” and “turbine area” [1].

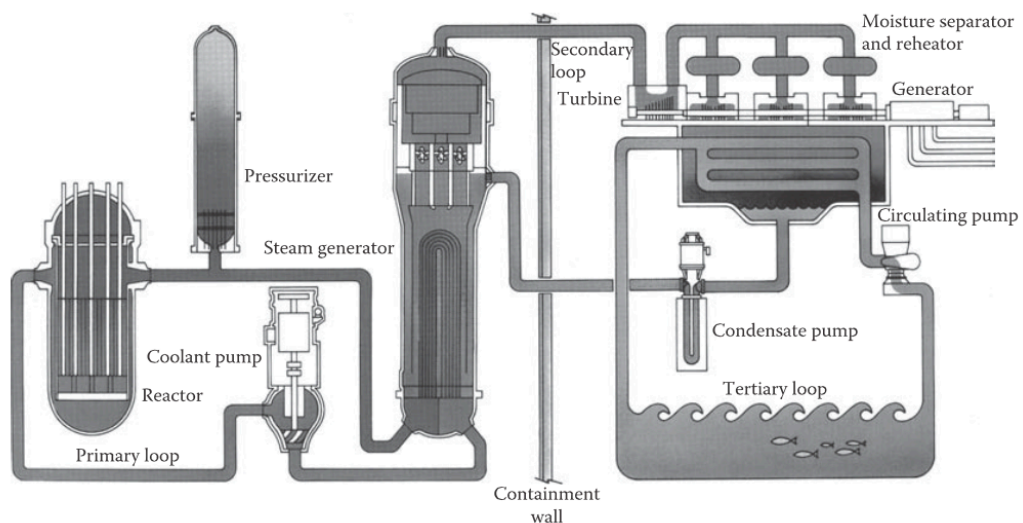


Figure 2.1: Basic scheme of a NPP (PWR)

Inside the “nuclear area”, NPP primary side, a nuclear reactor produces thermal energy from nuclear fissions: “A nuclear reactor is simply a sufficiently large mass of fissile material for a given shape in which such a fissile reaction can be sustained”. [2].

In a nuclear fission two particles collide, a neutron (projectile) and a heavy nuclei (target). The fissile nuclei split into two lighter nuclei and several neutrons, releasing approximately 200 MeV. The neutrons trigger a chain reaction, which is controlled

inside the nuclear reactor's core. The energy released by the fissions is used to heat water; hence, to produce thermal energy.

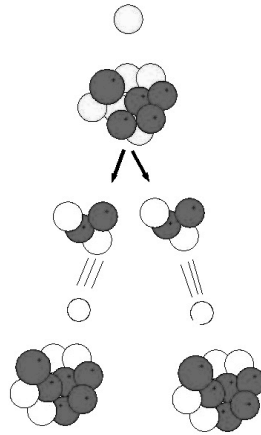


Figure 2.2: Chain Reaction

NPPs with Light Water Reactors (LWR) use uranium dioxide (UO_2) as fuel. Natural uranium is composed of two different isotopes: U^{238} (99.3%), and U^{235} (0.72%). Since the fissile nuclei are U^{235} , the enrichment of U^{235} , 2.5-5%, is necessary in order to maintain a chain reaction.

In the “turbine area”, NPP secondary side, the thermal energy is transformed in to electricity by steam turbines and generators.

Depending on the type of reactor, a NPP requires a specific configuration to transport the heat energy from the primary to the secondary side [3].

2.1.1 Pressurized Water Reactor

A PWR is a LWR that requires a NPP configuration with two different coolant circuits (see Figure 2.1). Both circuits connect the primary to the secondary side without mass exchange; hence, the radioactive material remains always inside the primary coolant loop. The main characteristic of a PWR is that the pressure inside the primary coolant circuit is maintained high, approximately 15 MPa, to avoid saturated boiling in the core [3].

Primary Side of a PWR

A containment surrounds the primary side of a PWR as a security measure since all the radioactive material is there. The elements that compound the primary side are: the pressure vessel, which contains the nuclear core, the pressurizer, which regulates the pressure of the system, and the steam generator, which operates as a connection between the primary and the secondary coolant circuits [3].

Secondary Side of a PWR

In the secondary side takes place the transformation of thermal energy into mechanical energy through the expansion of the steam inside the turbines. Then, the mechanical energy is transformed into electricity by the generator.

The main elements of the secondary side of a NPP and their operating principles are:

- Steam generator

Steam generators are vertical elements that integrate a bundle of U-tubes, through which the heated water from the reactor flows, and an upper section prepared with the proper equipment to produce high quality steam (99.75%). The preheated feed water coming from the secondary coolant loop enters the top of the steam generator, flows downward, is distributed across the bundle of U-tubes, and then flows upward and leaves the steam generator as high quality steam [4].

- Steam Turbine

The purpose of a steam turbine is to convert the internal energy of a high-temperature steam into mechanical work. Steam turbines are designed to receive a pressurized fluid and remove its momentum. [5] The fluid enters the turbine and expands through a nozzle. Part of the enthalpy of the fluid is converted into kinetic energy, producing an increase on the fluid velocity. The high-velocity fluid flows across the turbine blades, which lead the fluid to the rotor. The momentum of the fluid changes producing forces on the blades. These forces make the turbine-shaft turn generating mechanical work [6].

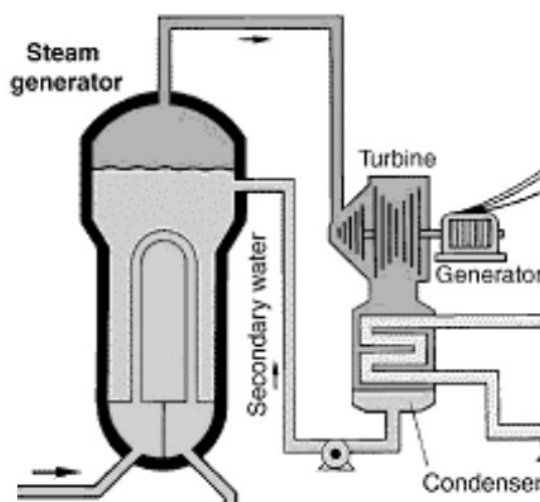


Figure 2.3: Steam generator and turbine of a PWR.

The steam generator is the connection between the primary and the secondary loop. Several U-tubes are inside the steam generator, in which the heated water of the primary coolant circuit flows. The heat is transferred through the tube walls to the

condensate of the secondary loop, which is evaporated, and then transferred to the high-pressure turbine. Inside the turbine the steam expands, decreasing its pressure and increasing its velocity. After flowing through the turbine, the quality of the steam decreases. In order to improve the water properties before entering into the low-pressure turbine, a separator and an over-heater are included. After leaving the LP-turbine, the steam is condensed and pumped back to the steam generator.

Rankine Cycle

The Rankine Cycle is a thermodynamic cycle, which takes place in the secondary loop of a PWR.

The Rankine Cycle involves a phase change of the working fluid. The explanation includes four steps that correspond to Figure 2.4:

- 1-2. Superheated or saturated vapour expands adiabatically (turbine).
- 2-3. The vapour-liquid mixture condenses to saturated liquid (condenser).
- 3-4. The liquid is pumped to higher pressure (pump).
- 4-1. The fluid is heated at constant pressure until the liquid boils and becomes saturated vapour (over-heater) [6].

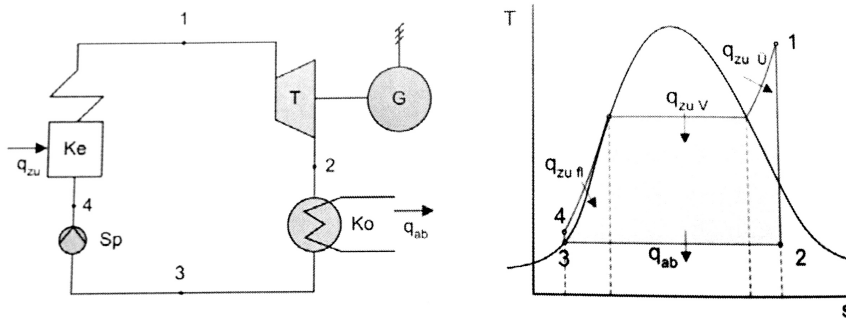


Figure 2.4: Rankine Cycle

The turbine and the pump works, W_T and W_P respectively, are:

$$W_T = m(h_1 - h_2) \quad \text{Equation 2.1}$$

$$W_P = m(h_4 - h_3) \quad \text{Equation 2.2}$$

The net work, W_N , is:

$$W_N = W_T - W_P \quad \text{Equation 2.3}$$

Inserting Eq. 2.1 and Eq. 2.2 in Eq. 2.3:

$$W_N = m((h_1 - h_2) - (h_4 - h_3)) \quad \text{Equation 2.4}$$

The addition of thermal energy, Q_A , is:

$$Q_A = m(h_4 - h_1) \quad \text{Equation 2.5}$$

The efficiency of the cycle, η , is defined as:

$$\eta = \frac{W_N}{Q_A} = \frac{(h_1 - h_2) - (h_4 - h_3)}{(h_4 - h_1)} \quad \text{Equation 2.6}$$

Neglecting the pump work:

$$\eta = \frac{W_N}{Q_A} = \frac{h_3 \approx h_4}{(h_4 - h_1)} \quad \text{Equation 2.7}$$

2.2 TRACE

Considering the objective of this thesis, which is simulating transients in NPP secondary loops, this present thesis uses TRACE.

TRACE is a best-estimate reactor system code developed by the U.S. Nuclear Regulatory Commission for analysing steady-state behaviour, transients, Loss of Coolant Accidents (LOCA's), and other accidents in light water reactors (PWR's and BWR's). TRACE uses finite-volume numerical methods to solve the two-phase flow and heat transfer equations.

TRACE takes a component-based approach to model a reactor system. Each TRACE-component can be divided in different cells over which the fluid, conduction and kinetics equations are averaged. The hydraulic components defined in TRACE are: pipes (PIPE), plenums (PLENUM), pressurizers (PRIZR), BWR fuel channels (CHAN), pumps (PUMP), jet pumps (JETP), separators (SEPD), tees (TEE), turbines (TURB), feed-water heaters (HEATR), valves (VALVE), and vessels (VESSEL). To model fuel elements or heated walls, heat structures (HTSTR) are included in TRACE. To simulate steady state and transients, FILL and BREAK components are used to apply the desired coolant-flow and boundary conditions.

The simulation time is a function of the total number of cells, the maximum step size, and the rate of change of the evaluated neutronic and thermal-hydraulic phenomena.

TRACE is only applicable within its assessment range. TRACE has been qualified to analyse the Economic Simplified Boiling Water Reactor (ESBWR) design and conventional PWR and BWR break Loss of Coolant Accidents (LOCAs) [7].

2.2.1 List of Components:

This list describes all the components used to accomplish the simulation.

- **BREAK**

The **BREAK** component is the most often used TRACE component to specify boundary conditions. The **BREAK** component imposes a pressure boundary condition one cell away from its adjacent component (see figure 2.5). The **BREAK** component does not necessary represent a physical entity in the simulated model; it represents the known hydrodynamic conditions of an actual physical location of the modelled system.

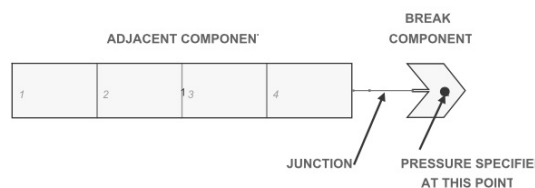


Figure 2.5: *BREAK* component

- **FILL**

The **FILL** component is used to impose boundary conditions at any 1D hydraulic component junction. The **FILL** component does not necessarily represent a physical entity in the simulated model. A **FILL** component imposes a coolant velocity or a mass flow boundary condition at the junction with its adjacent component [8].

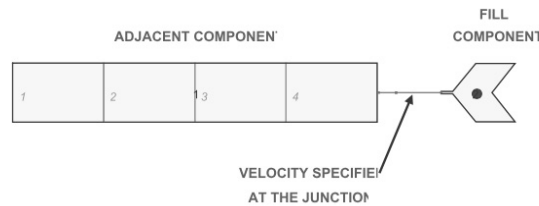


Figure 2.6: *FILL* component

- **HTSTR**

The **HTSTR** component evaluates the dynamics of conduction, convection, and gap-gas radiation heat transfer in a fuel-rod or structure hardware element. The pipe walls modelled by all other components are also evaluated by the **HTSTR** component [8].

- **PIPE**

The **PIPE** component models coolant flow in a 1D tube, channel, duct, or pipe. It can be used with a **BREAK**- and/or **FILL**-component boundary conditions to model 1D flow in a pipe, or it can be used as a connecting pipe between other components. It has the capability to model coolant flow-area changes, wall heat sources, and heat transfer between the wall inner and outer surfaces. Figure 2.7 shows a typical noding diagram for a **PIPE** component. The numbers within the **PIPE** indicate cell numbers, and those above the **PIPE** indicate cell-interface numbers. The geometry is specified by providing a volume and length for each cell, and a flow area and hydraulic diameter at each cell face. The junction-interface variables, **JUN1** and **JUN2**, provide reference numbers for connecting the **PIPE** to other component junctions [8].

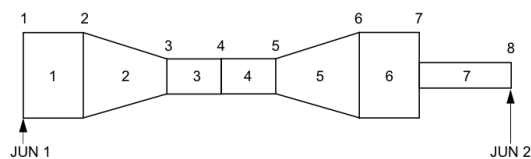


Figure 2.7: *PIPE* component

- **SEPD**

The SEPD component is usually employed to model the steam separators and moisture dryers [8].

- **TURB**

The TURB component is a special TEE component case (the TEE component consists of two pipe components connected by a single junction), which includes additional models to simulate the operation of a steam turbine. The TURB component simulates energy removal from the flow due to the conversion of fluid energy to mechanical energy, the efficiency of the turbine, and the pressure losses. In addition, the TURB component includes the capability to simulate liquid drains or steam taps, and the dynamics of multi- stages and turbine rotor assembly[8].

The TURB component must be simulated with 2 cells in the primary arm of the TEE component (i.e., NCELL1 = 2) and with one cell in the side arm (i.e., NCELL2 = 1). The side arm connects to cell 2 of the primary arm (i.e., JCELL = 2) (see Figure 2.8). The flow through the turbine is not treated in detail based on first principals, but is simulated by adjusting the momentum and energy flow at cell edge 2 consistent with a lumped momentum and energy balance for the turbine component[8].

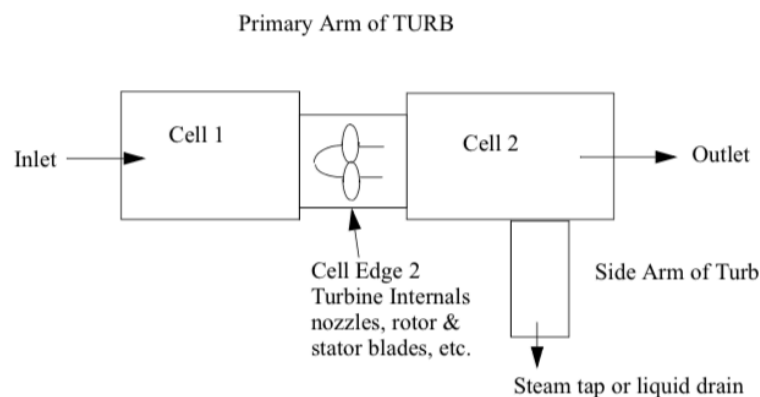


Figure 2.8: TURB component nodalization

- **VALVE**

The VALVE component is used to model various types of valves associated with light-water reactors. The valve action is modelled by a component action that adjusts the flow area and hydraulic diameter at a cell interface of a 1D hydraulic component as shown in Figure 2.9 [8].

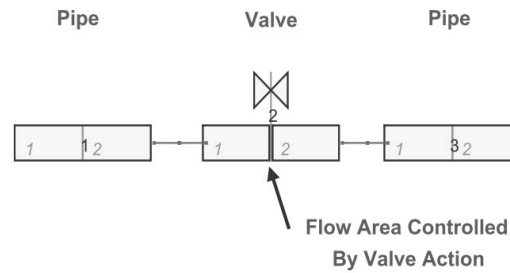


Figure 2.9: VALVE component noding diagram

2.2.2 FRIC and K-FACTOR

K-Factors are form-loss coefficients, and all the TRACE components included in the model require setting K-Factors as input.

To model pressure losses in 1D components TRACE system code includes the friction-factor correlation option flag (NFF). Several options are available.

- NFF = 1 applies a homogeneous-flow friction factor for wall and structure drag.
- NFF = -1 is the same but adds an internal form-loss computation for abrupt changes in flow area between mesh cells.
- NFF = -100 applies the form-loss computation only.

It is recommended that $NFF = 1$ or -1 be applied at mesh-cell interfaces everywhere except at an interface where flow choking is anticipated. $NFF = 0$ is recommended for this case. The reason for setting $NFF = 0$ at the flow-choking interface is to avoid becoming friction limited as the onset of flow choking is approached. It is recommended that the user account for gradual flow-area change, flow turning, and orifice form losses by specifying FRIC or KFAC additive form-loss coefficients as well [8].

TRACE models all flow area changes as smooth area changes and evaluates only the Bernoulli-equation reversible pressure loss or gain associated with such an area change. Therefore, additive loss coefficients must be input for all irreversible form losses in the modelled system. For example, additive loss coefficients must be input to model the irreversible form loss at a cell-edge interface for a change in cell-average fluid flow area, a change in flow direction, or a flow orifice. It may be necessary to add additional loss coefficients to obtain the correct pressure drops across the component [8].

Inside the NAMELIST included in the model some variables can be defined. IKFAC is the option that defines whether additive loss coefficients (FRIC) or K-factors need to be input in the component data:

- IKFAC=0: additive loss coefficients will be input.
- IKFAC=1: K-factors will be input.

When this option is selected (IKFAC=1), K-factors (dimensioned NCELLS+1) replace the FRIC array input cards in all components, and vice versa.

For TURB components a FRIC of 10^{21} at a cell edge results in essentially no liquid flow across that cell edge, while a FRIC of -10^{21} at a cell edge results in essentially no vapour flow across that cell edge. A FRIC of 10^{21} results in TRACE using a wall drag friction coefficient of 10^{21} for the liquid phase and a wall drag friction coefficient of 0.0 for the vapour phase and zero interfacial shear at that cell edge. A FRIC of -10^{21} results in TRACE using a Wall drag friction coefficient of 0.0 for the liquid phase and a wall drag coefficient of 10^{21} for the vapour phase [9].

To input K factors that depend on flow direction, TRACE system code includes the NFRC1 option. This option can be used to require input of forward and reverse loss coefficients for all 1D components. When NFRC1=2, the dimension of the FRIC array is doubled to $2 \times (\text{NCELLS} + 1)$. The input data must be defined to provide two consecutive arrays of loss coefficients, each dimensioned $\text{NCELLS} + 1$. Each array is in LOAD format and must be terminated with an "E." The first array provides loss coefficients that are used with positive velocities, and the second array provides loss coefficients that are used with negative velocities [9].

2.3 System description

In the secondary side of a NPP the thermal energy extracted from the nuclear reactor is transformed into mechanical energy.

The secondary system to simulate is composed of a steam generator, one double-flux HP-turbine and three double-flux LP-turbines. A separator, an extraction, and an over-heater are placed between the HP-Turbines and the LP-Turbines. Three condensers are located after the LP-Turbines, and a set of over-heaters and pumps close the circuit.

The steam leaves the steam generator and enters the HP-Turbine. Inside the turbine the vapour quality decreases. After leaving the turbine, the fluid flows through the separator, which extracts most of the liquid, the extraction, which reduces the mass flow, and the over-heater, which provides the fluid the necessary temperature to become over-heated vapour. Then, the high-quality vapour is distributed to the LP-Turbines. Finally, the steam is condensed, heated, and pumped back to the steam generator.

Since this system corresponds to a real NPP, all the data are confidential. Consequently, the data included in this document, both nominal data and results, have been normalized. Normalization consists of dividing each type of parameter (pressure, mass flow...) by a set value: Pn for pressure, Tn for temperature, An for the flow area, and Mfn for mass flow.

- Pn is the pressure at the inlet of the system valve.
- Tn is the saturated temperature at Pn.

- M_{in} is the mass flow at the inlet of the HP- Turbine.
- A_n is the flow area at the inlet of the HP-Turbine.

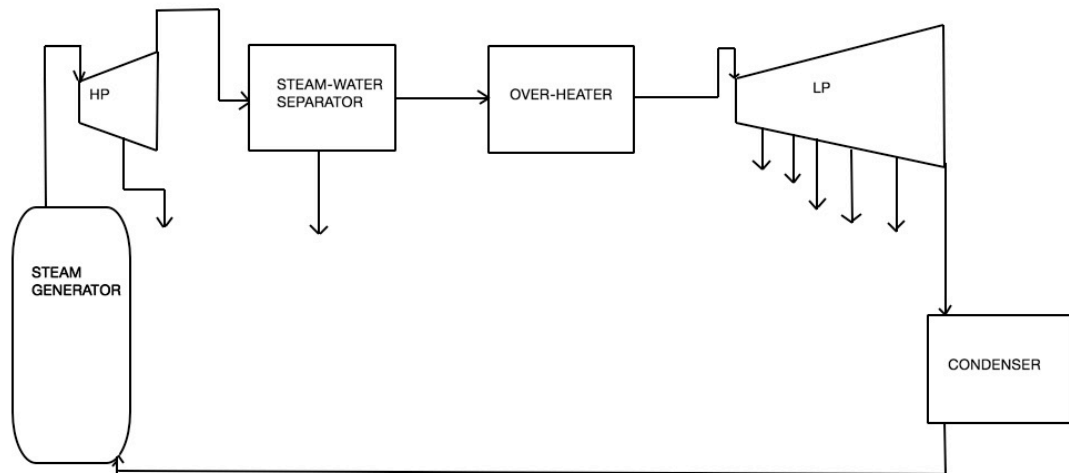


Figure 2.10: System scheme

2.3.1 System nominal data

HP-Turbine

Each HP-Turbine has a steam extraction between the inlet and the outlet.

The data included in the table correspond to a single HP-Turbine.

HP-TURBINE	PRESSURE	MASS FLOW	FLOW AREA	VAP. QUALITY
INLET	1,000	1,000	1,000*	1
EXTRACTION	0,358	0,084	-	1
OUTLET	0,191	0,915	2,925*	0,88

Table 2.1: HP-Turbine nominal data

* The HP-Turbine flow area calculation is included in section 3.2

Separator, Extraction, and Over-Heater

The separator and over-heater geometrical data were not provided.

SEPARATOR	PRESSURE	MASS FLOW	VAPOUR QUALITY
INLET	0,1914	1,8299	0,8800
EXTRACTION	0,1867	0,2209	0,0000
OUTLET	0,1775	1,6090	1,0000

Table 2.2: Separator nominal data

EXTRACTION	PRESSURE	MASS FLOW	VAPOUR QUALITY
INLET	0,0177	1,6090	1,0000
EXTRACTION	0,0177	0,1420	1,0000
OUTLET	0,0177	1,4670	1,0000

Table 2.3: Extraction nominal data

OVER-HEATER	PRESSURE	VAPOUR QUALITY
INLET	0,0177	1,0000
OUTLET	0,1806	1,0000

Table 2.4: Over-Heater nominal data

LP-Turbine

Each single LP-Turbine has five steam extractions. In the data provided, two of the extractions are considered as one.

The data included in the table correspond to a single LP-Turbine.

LP-TURBINE	PRESSURE	MASS FLOW	FLOW AREA	VAP. QUALITY
INLET	0,1806	0,2445	1,0289*	1
EXTRACTION 1	0,0606	0,0097	-	1
EXTRACTION 2	0,0276	0,0144	-	0,98
EXTRACTION 3	0,0100	0,0154	-	0,74
EXTRACTION 4	0,0023	0,0133	-	0,38
OUTLET	0,0009	0,1918	24,758*	0,89

Table 2.5: LP-Turbine nominal data

* The LP-Turbine flow area calculation is included in section 3.5

Condenser

The geometrical data of the condenser were not provided.

CONDENSER	PRESSURE	VAPOUR QUALITY
INLET	0,00093	0,8900
OUTLET	0,00093	0,0000

Table 2.6: Condenser nominal data

3. TRACE Model Components

This chapter includes a detailed description of the components designing. All the data included have been normalized.

3.1 Valve

The model includes a stop valve, which is set to close when the pressure on its cell 1 reaches a 108% of the set value. To simulate the valve, a TRIP, a Signal Variable, and a VALVE component were necessary.

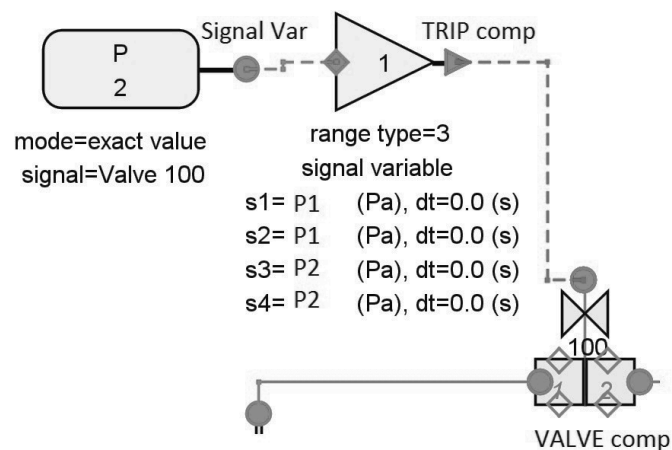


Figure 3.1: VALVE-TRIP component

VALVE component

TRACE system code includes different types of valves. The model uses VALVE component [3]: constant flow area until trip IVTR is set ON, then flow-area fraction vs. independent-variable-form table is evaluated [9].

The valve geometry is defined as a normal pipe. The flow area and the junction flow area are the same as the inlet flow area of the HP-Turbine, which is defined in section 3.2. The length and the volume were estimated in order to be compatible with the adjacent components.

The VALVE component characteristic inputs are:

Pressure	1
Temperature	1
Vapour quality	1

Table 3.1: VALVE initial conditions

- Valve Interface
The valve interface is the cell interface number where the VALVE flow area is adjusted. In this model the valve interface is at edge 2.
- Flow Area Adjustment type: [0] Flow area fraction per second.
- Opening conditions

Maximum valve rate	0,5 [1/s]
Off adjustment rate	0,5 [1/s]
Minimum position	0 [-]
Maximum position	1 [-]
Initial flow area fraction	1 [-]
Valve stem position	1 [-]

Table 3.2: VALVE description

- Adjustment tables

First Adjustment Table

No unit	Area Fraction
0	1

Second Valve Table

Time [s]	No unit
0	1
5	0

The first adjustment table corresponds to the initial conditions, and the second valve table is activated as the TRIP is set ON.

Signal Variable

A signal variable is a specialized control block, which takes its input from some parameter at any location in the computational mesh, and sends that value to its own output [8].

In this model the signal variable type is [21] Pressure, its behaviour mode is “Exact Value”, and the signal source is the first cell pressure of the VALVE component.

TRIP component

A trip is an ON/OFF switch that can be used to decide when to evaluate a component hardware action.

The trip-signal definition and setpoint values are associated with the input specification of a trip. In this model the trip-signal type is a Signal Variable and the input source is the pressure signal defined above. Setpoint data are values that define the exact state of the trip (ONforward, ONreverse, or OFF), based upon where the actual trip-signal lies, as compared to the setpoint point(s) [8]. For the modelled TRIP, the setpoint values are $P1=1,08P_n$ and $P2=1,1P_n$ (see figure 3.2). The TRIP initial set status is [1] ON forward.

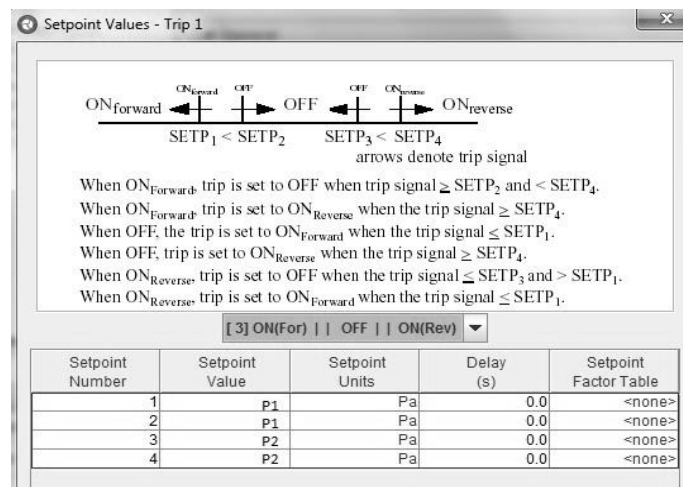


Figure 3.2: TRIP Setpoint Values

3.2 HP-Turbine

The real system includes double-flux turbines. Since the TRACE system code does not model double-flux turbines, each turbine has been simulated as two different elements.

The simulation of a single HP-turbine needs two TURB components, one to simulate the pressure drop between the inlet and the extraction, and another one to simulate the pressure drop between the extraction and the outlet. Since the model needs two turbines to simulate a double-flux turbine, the total number of TURB components that includes the model is four.

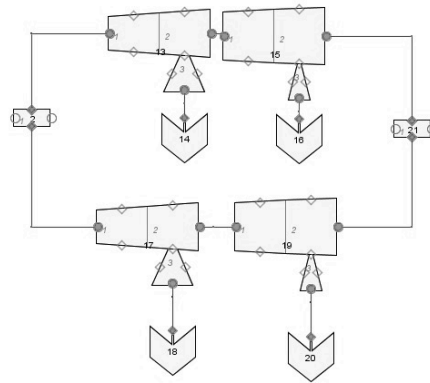


Figure 3.3: HP-Turbine Model

The code needs many inputs to define the turbine operation: geometry, physical constants, and initial and boundary conditions.

GEOMETRY

TRACE requires different values to define the geometry of the TURB component. Since the turbine geometry data provided are limited many assumptions were made.

Each TURB component is composed of three cells and their respective edges (see figure 3.4), which have to be geometrically defined.

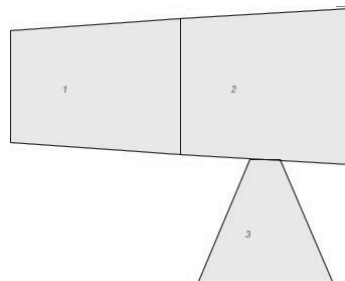


Figure 3.4: Single HP-Turbine

The only real data of the turbine geometry are:

HP-Turbine	Blade Height (B)	Mean Diameter (D)
Inlet	1,000	1,000
Outlet	2,619	1,117

Table 3.3: HP-Turbine geometry data

The HP-Turbine and the LP-Turbine geometrical data provided of the have been divided by the blade height and the middle diameter at the HP-Turbine inlet (normalization).

The HP-turbine inlet and outlet flow area calculation is possible with the data provided. These flow areas correspond to the first edge flow area of TURB components 13 and 17, and to the last edge flow area of TURB components 15 and 19 (see figure 3.3).

The calculation of the flow areas is the following:

- Inlet flow area:

$$D_{inlet} = 2 \cdot \left(\frac{B_{inlet}}{2} \right) + D_{rotor} \quad \text{Equation 3.1}$$

$$A_{rotor} = \left(\frac{D_{rotor}}{2} \right)^2 \cdot \pi \quad \text{Equation 3.2}$$

$$A_{total} = \left(\frac{D_{rotor} + 2 \cdot B_{inlet}}{2} \right)^2 \cdot \pi \quad \text{Equation 3.3}$$

$$A_{flow_inlet} = A_{total} - A_{rotor} \quad \text{Equation 3.4}$$

The rotor diameter is constant along the turbine.

- Outlet flow area:

$$A_{total} = \left(\frac{D_{rotor} + 2 \cdot B_{outlet}}{2} \right)^2 \cdot \pi \quad \text{Equation 3.5}$$

$$A_{flow_outlet} = A_{total} - A_{rotor} \quad \text{Equation 3.6}$$

The geometry of a single turbine, which is defined by two TURB components, requires the input of five flow areas, one per junction. A linear distribution from the inlet to the outlet flow area is the solution. The following table includes three edge flow areas for TURB components 13 and 17, inlet (edge 1), junction 1 (edge 2), and junction 2 (edge 3), and three flow areas for TURB components 15 and 19, junction 2 (edge 1), junction 3 (edge 2), and outlet (edge 3). Junction 2 flow area is the same for both TURB components, TURB 13-17 edge 3 and TURB 15-19 edge 1.

HP-Turbine	Flow Area
Inlet	1,000
Junction 1	1,481
Junction 2	1,962
Junction 3	2,444
Outlet	2,925

Table 3.4: TURB components 13-15 geometry

The remaining geometry data are:

- Length: set in order to optimize pressure losses.
- Volume averaged flow area: average between the edges flow area.

HP-Turbine	Vol. Avg. Flow Area
TURB 13/17 Cell 1	1,2405
TURB 13/17 Cell 2	1,7215
TURB 15/19 Cell 1	2,203
TURB 15/19 Cell 2	2,6845

Table 3.5: TURB components 13-15 geometry 2

- Cell volume: calculated by TRACE.
- Hydraulic diameter: calculated by TRACE.

Each TURB component includes a side tube, which models a water extraction. The only inputs that define the side tube are its geometry and the K-Factors. Since the mass of water that flows through the tube cannot be defined, both the geometry and the K-factors have been set to reach the required mass flow at the extraction. Since the side tube measure has no relevance for this analysis it is not included in this document.

CONSTANTS

- Turbine efficiency
The turbine efficiency was assumed 0.8 [-]. This value was assumed based on typical turbine parameters.

INITIAL CONDITIONS

The TURB component includes a table, in which the initial conditions must be set for each cell.

TURB component 13-17

Cell Number	Pressure	Liquid Temperature	Vapour Temperature	Gas Volume Fraction
1	1	1	1	0,9953
2	0,358	0,8905	0,8905	0,907
3	0,358	0,8905	0,8905	0,907

Table 3.6: TURB component 13-17 initial conditions

TURB component 15-19

Cell Number	Pressure	Liquid Temperature	Vapour Temperature	Gas Volume Fraction
1	0,358	0,8905	0,8905	0,907
2	0,191	0,8353	0,8353	0,8424
3	0,191	0,8353	0,8353	0,8424

Table 3.7: TURB component 15-19 initial conditions

BOUNDARY CONDITIONS

The BREAK components located at the outlet of the side tubes set the boundary conditions. To define a BREAK component the inputs are:

- Volume
The BREAK component volume is the same as the volume of its adjacent cell.
- Length
The BREAK length is defined to provide the same flow area as the one of the adjacent cell edge.

$$BREAK_{length} = \frac{BREAK_{volume}}{Adj. CELL_{flow\ area}} \quad \text{Equation 3.7}$$

The BREAK component adjacent cell corresponds to the TURB component side tube.

- Initial Gas Volume Fraction
The initial gas volume fraction is 0 for the four BREAK components.
- Pressure and temperature

	BREAK 14-18	BREAK 16-20
Pressure	0,359	0,191
Temperature	0,8905	0,8353

Table 3.8: BREAK 14-16-18-20 initial conditions

3.3 Separator

The system separator has been simulated with a SEPD component. A SEPD component is composed of a main tube and a side tube. The separator has been designed with two cells on the main tube and one on the side tube, and its orientation is vertical. Inside the second cell, the fluid is separated in steam and liquid. The steam flows out of the separator, and the liquid water flows downwards through the side-tube.

The flow area of the separator main tube is the same as the HP-Turbine inlet flow area, and the side-tube flow area was estimated in order to extract the desired liquid mass flow. The initial conditions are the same as the HP-Turbine's outlet conditions. The BREAK component, which is located after the side tube, is set to 0.187 [P/Pn] and 0.833 [T/Tn].

The separator type used in the model is: [0] Ideal Separator.

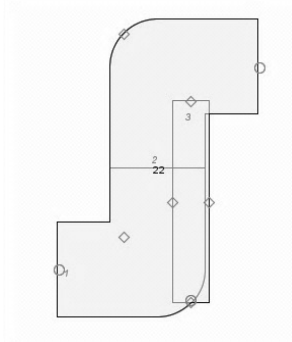


Figure 3.5: Separator

3.4 Over-Heater

To simulate an over-heater the model uses a PIPE in combination with a HTSTR.

PIPE

The pipe is composed of two cells, which have the same flow area as the HP-turbine inlet. The initial conditions are the same as at the LP-Turbine inlet. The PIPE component includes the option of editing a HTSTR within it.

HTSTR

The HTSTR component, which is edited inside the PIPE component, is cylindrical, and its axial plane is in X direction.

To define a HTSTR the code includes a table, in which the heat structure boundary condition data and axial nodalization are set.

Inner Surface Boundary Conditions	Axial Cell	Outer Surface Boundary Conditions
[2] Pipe: 25 Cell: 1	1	[1] HTC:- SFT:-
[2] Pipe: 25 Cell: 2	2	[5] Surf Temperature

Table 3.9: HTSTR boundary conditions

Boundary conditions:

- [1] Constant HTC SFT: constant heat transfer coefficient, and constant sink temperature.
- [2] Hydro component: heat structure is coupled to the hydro component.
- [5] Constant Surface Temp: constant fixed surface temperature.

The inner surface boundary conditions are type [2]; therefore, the heat structure inner surface is coupled to PIPE 25, which is the PIPE that models the over-heater.

The outer surface boundary condition of HTSTR axial cell 1 is type [1] in order to heat the steam and evaporate the liquid. At axial cell 2 the outer boundary condition is type [5] since the system requires a specific temperature at the LP-Turbine inlet.

3.5 LP-Turbine

The real system includes three double-flux LP-turbines. Each turbine has been simulated as two different elements; therefore, the LP-Turbine model includes six turbines. Each single LP-Turbine has five extractions, but just four of them are simulated since there are no data from one extraction. For the simulation of an extraction the code requires a TURB component. Figure 3.6 shows the complete model of the LP-Turbine.

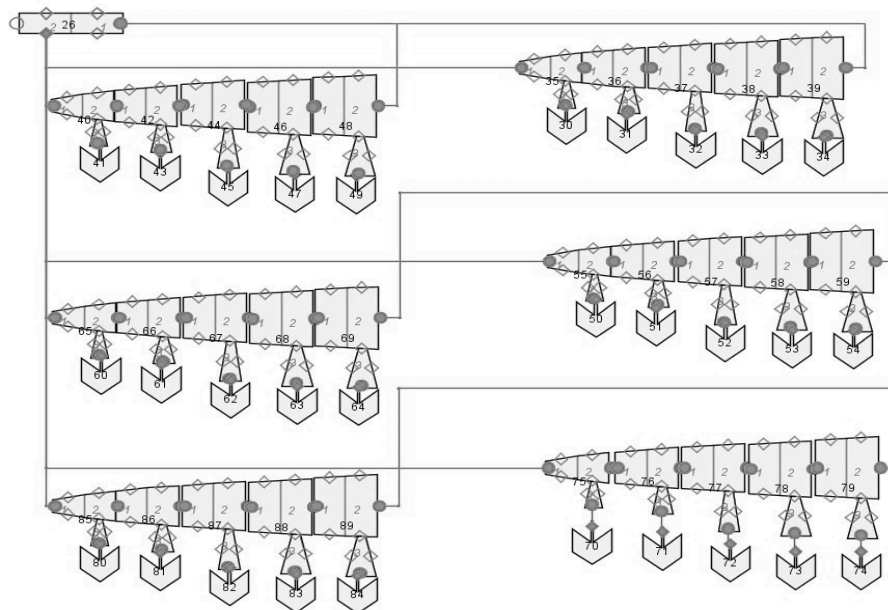


Figure 3.6: LP-Turbine model

The code needs many inputs to define the turbine operation: geometry, physical constants, and initial and boundary conditions.

GEOMETRY

The only real data of the LP-Turbine geometry are:

LP-Turbine	Blade Height (B)	Middle Diameter (D)
Inlet	0,6031	1,7
Outlet	10,624	2,431

Table 3.10: LP-Turbine geometry data

Following the same steps for the calculation of the LP-Turbine inlet and outlet flow areas as those explained in section 3.2 (HP-Turbine), and applying a linear distribution to obtain a flow area for each edge, the results are:

LP-Turbine	Flow Area
Inlet	1,029
Junction 1	3,505
Junction 2	5,980
Junction 3	8,456
Junction 4	10,932
Junction 5	13,408
Junction 6	15,883
Junction 7	18,359
Junction 8	20,835
Junction 9	23,311
Outlet	24,758

Table 3.11: TURB component 35-39 geometry

The remaining geometry data are:

- Length: set in order to optimize pressure losses.
- Volume averaged flow area: average between the edges flow area.

LP-Turbine	Avg. Vol. Flow Area
TURB 35 Cell 1	2,2667
TURB 35 Cell 2	4,7422
TURB 36 Cell 1	7,2180
TURB 36 Cell 2	9,6940
TURB 37 Cell 1	12,1698
TURB 37 Cell 2	14,6455
TURB 38 Cell 1	17,1213
TURB 38 Cell 2	19,5970
TURB 39 Cell 1	22,0728
TURB 39 Cell 2	24,0341

Table 3.12: TURB component 35-39 geometry 2

- Cell volume: calculated by TRACE.
- Hydraulic diameter: calculated by TRACE.

Each TURB component includes a side tube, which models a water extraction. The only inputs that define the side tube are geometry and K-Factors. Since the mass of water that flows through the tube cannot be defined, both geometry and K-factors have been set to reach the required mass flow at the extraction. Since the side tube measure has no relevance for this analysis it is not included in this document.

TURB component 35

Cell Number	Pressure	Liquid Temperature	Vapour Temperature	Gas Volume Fraction
1	0,1806	0,83	0,83	1
2	0,0607	0,751	0,751	1
3	0,0607	0,751	0,751	1

Table 3.13: TURB component 35 initial conditions

TURB component 36

Cell Number	Pressure	Liquid Temperature	Vapour Temperature	Gas Volume Fraction
1	0,0607	0,751	0,751	1
2	0,276	0,704	0,704	0,98
3	0,276	0,704	0,704	0,98

Table 3.14: TURB component 36 initial conditions

TURB component 37

Cell Number	Pressure	Liquid Temperature	Vapour Temperature	Gas Volume Fraction
1	0,276	0,704	0,704	0,98
2	0,276*	0,704*	0,704*	0,74
3	0,01	0,652	0,652	0,74

Table 3.15: TURB component 37 initial conditions

TURB component 38

Cell Number	Pressure	Liquid Temperature	Vapour Temperature	Gas Volume Fraction
1	0,03*	0,709*	0,709*	0,74
2	0,03*	0,709*	0,709*	0,38
3	2,32E-03	0,709	0,709	0,38

Table 3.16: TURB component 38 initial conditions

TURB component 39

Cell Number	Pressure	Liquid Temperature	Vapour Temperature	Gas Volume Fraction
1	0,03*	0,709*	0,709*	0,38
2	0,03*	0,709*	0,709*	0,89
3	9,26E-04	0,559	0,559	0,89

Table 3.17: TURB component 39 initial conditions

For initialization purposes all the data marked with a “*” are higher than the real initial conditions.

BOUNDARY CONDITIONS

Pressure and temperature:

	BREAK 30	BREAK 31	BREAK 32	BREAK 33	BREAK 34
Pressure	0,0607	0,276	0,01	2,32E-03	9,26E-04
Temperature	0,751	0,704	0,652	0,591	0,559

Table 3.18: BREAK components 30-34 initial conditions

The remaining BREAK component design features are explained on section 3.2.

3.6 Condenser

The real system includes three condensers, one per double-flux LP-Turbine. Each condenser was defined as a long PIPE component in combination with a HTSTR.

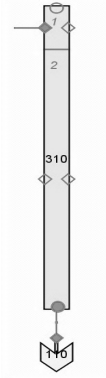


Figure 3.7: Condenser

PIPE

The PIPE component dimensions were designed in order to condense the whole mass flow. The PIPE was defined with two cells, the first one much shorter than the second one (see figure 3.7). The PIPE initial conditions are the same as at the LP-Turbine outlet (see table 3.17, cell 3).

HTSTR

The HTSTR is defined within the PIPE component. The HTSTR geometry is cylindrical, and its axial plane is in X direction.

To define a HTSTR the code includes a table, in which the heat structure boundary conditions and axial nodalization are set.

Inner Surface Boundary Conditions	Axial Cell	Outer Surface Boundary Conditions
[2] Pipe: 310 Cell: 1	1	[5] Surf Temperature
[2] Pipe: 310 Cell: 2	2	[0] Flux

Table 3.19: HTSTR boundary conditions

Boundary conditions:

- [0] Constant Heat Flux
- [2] Hydro component: heat structure is coupled to the hydro component.
- [5] Constant Surface Temp: constant fixed surface temperature.

The inner surface boundary conditions are type [2]; therefore, the heat structure inner surface is coupled to PIPE 310, which is one of the PIPEs that model the condensers.

The outer surface boundary condition inside cell 1 is type [5]. The fluid is cooled inside the first cell due to the constant surface temperature imposed by the HTSTR. In cell 2 the outer surface boundary condition is type [0]. A constant heat flux is extracted and the steam is condensed.

3.7 Mass Flow Distribution

Since the model includes components that work in parallel, at some points the mass flow needs to be divided. The solution for the mass flow division is to use simple PIPE components, which include cross flow connections. The hydraulic components connected to the cross flow connections receive exactly the same amount of mass flow.

The modelled system requires a mass flow division before the HP- and LP-Turbine (see figure 3.8). The mass flow enters a PIPE component with cross flow connections, and divides into two before the HP-Turbine and into six before the LP-Turbine.

Cross flow connections were also used to model a water extraction in between the separator and the over-heater, and to gather two different flows, at the outlet of the turbines (see Figure3.8).

3.8 Complete Model

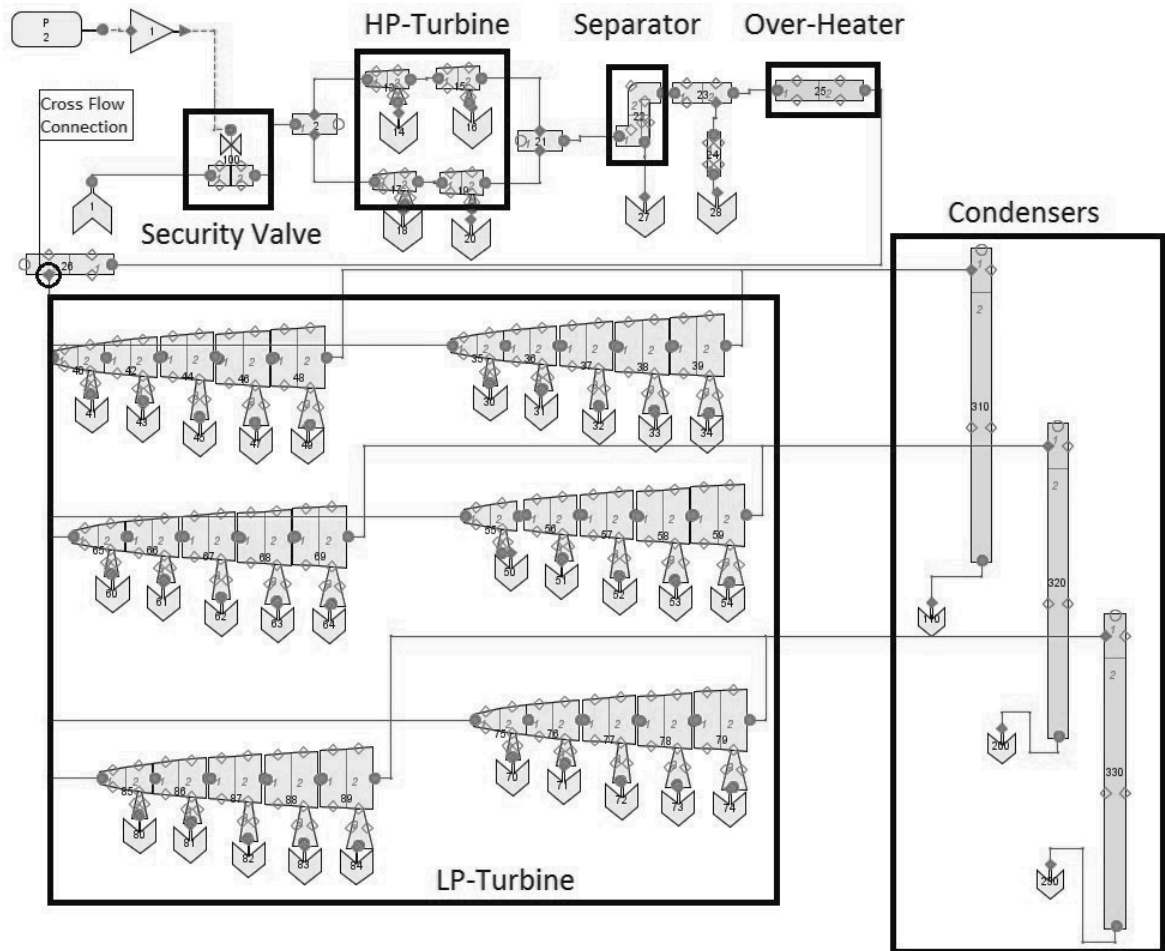


Figure 3.8: Complete Model

4. Steady-State: Simulation & Results

To run a steady-state simulation the option [0] Steady-State must be set. The model used for the steady-state simulation is the one defined in chapter 3.

This chapter includes the FILL component set up, the steady-state simulation results, and an argumentation for the K-factors used in the model.

4.1 FILL

The FILL component included in the model simulates the conditions before the fluid enters the valve and the HP-Turbine. These conditions are not the same as at the steam generator outlet since there are form losses and a steam extraction located between the HP-turbine and the steam generator.

TRACE includes different configurations of the FILL component. For the steady-state simulation this model uses FILL [2] (constant mass flow).

The main input data used to set the FILL component are:

FILL type	[2]
Pressure	1
Temperature	1
Mass Flow	2
Vapour quality	1

Table 4.1: Steady-State FILL description

4.2 Results

After running the TRACE model the relevant data provided by the program were analysed. This section includes a comparison between the extracted data from the real NPP and the data provided by the simulation. All the data included in this section have been normalized.

Since the model includes many components, which affect the boundary conditions at the inlet and outlet of the turbines, both the HP- and LP-Turbines have been developed in two separate models. In this way it is possible to set proper boundary conditions, and obtain simulation results, which are very close to the expected ones. The simplified model results will be compared with the data obtained from the complete model

simulation, in order to explain how the components included in the complete system affect the HP- and LP-turbines operation.

4.2.1 Simplified HP-Turbine model

The HP-Turbine simplified model (see figure 4.1) includes two TURB components, three BREAK components, and one FILL component.

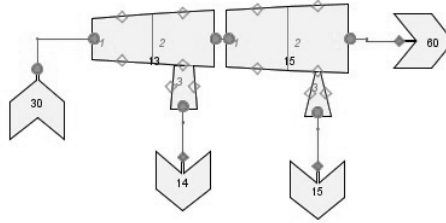


Figure 4.1: Modelled HP-Turbine

Both TURB components have the same initial conditions and geometry as the components included in the complete model (see section 3.2).

The FILL and the BREAK components set the boundary conditions at the HP-Turbine inlet, extraction, and outlet.

	Pressure	Saturation Temperature	Mass Flow	Vapour Quality
FILL 60	1	1	1	1
BREAK 14	0,358	0,891	-	0
BREAK 15	0,191	0,827	-	0
BREAK 60	0,191	0,827	-	0

Table 4.2: HP-Turbine simplified model boundary conditions

The TRACE system code does not include the option of defining a fixed inlet mass flow in the BREAK component.

The simulation of the HP-Turbine simplified model reaches steady state after 10.022 simulated seconds, and the CPU time is 0.359 [s].

The HP-Turbine real data provided are pressure measurements at some specific points of the turbine: inlet, outlet, and fluid extraction.

	Inlet Pressure	Extraction Pressure	Outlet Pressure
REAL	0,9620	0,3867	0,1846
SIMULATED	0,9725	0,3721	0,1935
ERROR [%]	1,092	-3,775	4,844

Table 4.3: HP-Turbine simplified model results

Table 4.3 shows the error calculated from the comparison between the simulation results and the real pressure data. At the inlet of the HP-Turbine, the calculated error is very small. At the extraction and at the outlet the error is bigger.

Figure 4.2 and 4.3 correspond to graphs, which show a comparison between the real data (blue) and the simulation results (red).

Figure 4.2 shows the pressure inside the HP-Turbine stages per unit time. Each line represents the pressure inside the TURB component cells:

- Sim. Inlet Pressure: TURB component 13, cell 1.
- Sim. Extraction Pressure: TURB component 13, cell 2.
- Sim. Outlet Pressure: TURB component 15, cell 2.

The graphical comparison between the real and the simulated data reveals that the simulated pressure inside the turbine is realistic.

Figure 4.3 shows the HP-Turbine pressure drop per stage, which follows the same pattern as the real pressure drop.

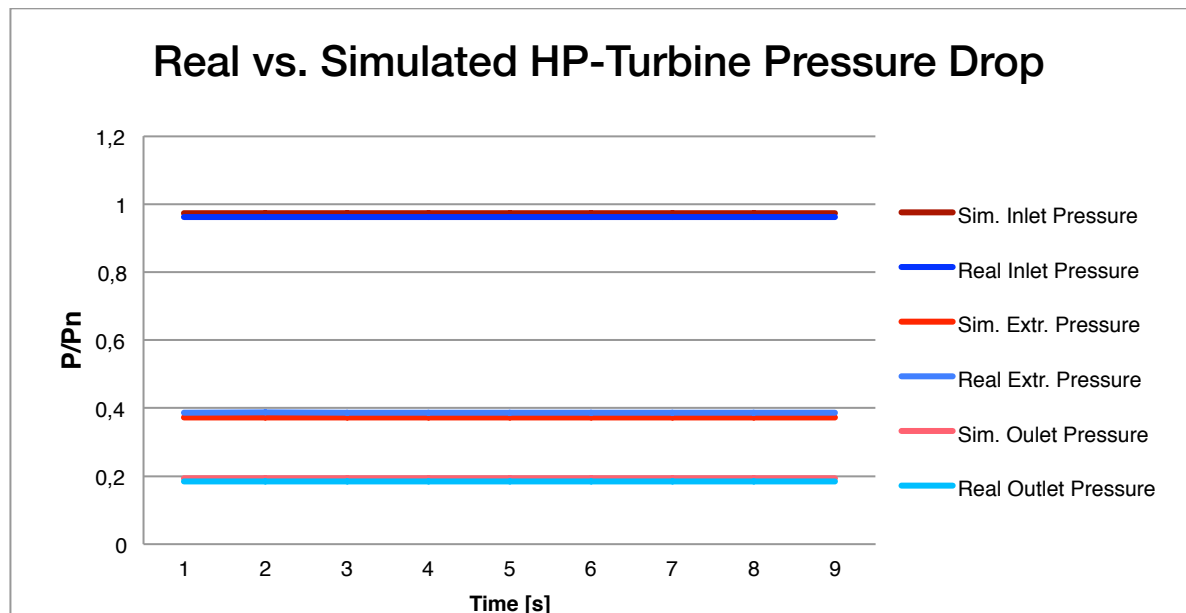


Figure 4.2: Graph-Simulated vs. Real HP-Turbine Pressure Drop

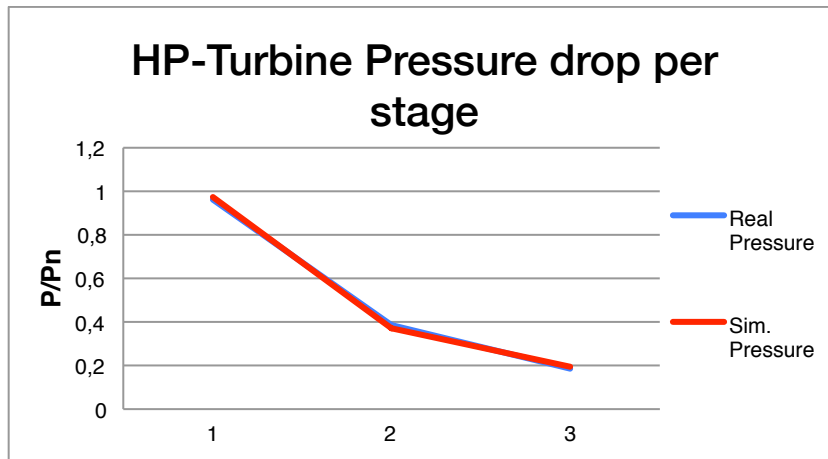


Figure 4.3: Graph-Simulated vs. Real HP-Turbine Pressure Drop per stage

4.2.2 Simplified LP-Turbine model

The LP-Turbine simplified model (see figure 4.4) includes five TURB components, six BREAK components, and one FILL component.

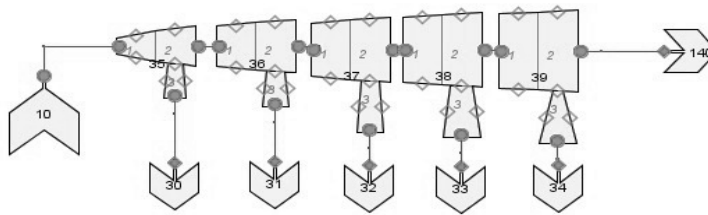


Figure 4.4: LP-Turbine simplified model

All the TURB components have the same initial conditions and geometry as the components included in the complete model (see section 3.6).

The FILL and the BREAK components set the boundary conditions at the HP-Turbine inlet, extractions, and outlet (see table 4.4).

	Pressure	Saturation Temperature	Mass Flow	Vapour Quality
FILL 10	0,181	0,831	0,238	1
BREAK 30	0,061	0,751	-	0
BREAK 31	0,0276	0,704	-	0
BREAK 32	0,01	0,652	-	0
BREAK 33	2,31E-03	0,591	-	0
BREAK 34	9,26E-04	0,559	-	0
BREAK 140	9,26E-04	0,559	-	0

Table 4.4: LP-Turbine simplified model boundary conditions

The simulation of the HP-Turbine simplified model reaches steady state after 61.84 simulated seconds, and the CPU time is 0.514 [s].

	Inlet Pressure	Extraction 1 Pressure	Extraction 2 Pressure	Outlet Pressure
REAL	0,17835	0,06271	0,02721	0,00129
SIMULATED	0,16559	0,06122	0,02770	0,00123
ERROR [%]	-7,15415	-2,36664	1,77813	-4,52251

Table 4.5: LP-Turbine simplified model results

Table 4.5 shows the error calculated from the comparison between the simulation results and the real pressure data. At the inlet of the LP-Turbine, the calculated error is significant, but at the extractions and at the outlet the error is smaller.

Figure 4.5 and 4.6 correspond to graphs, which show a comparison between the real data (blue) and the simulation results (red).

Figure 4.5 shows the pressure inside the LP-Turbine stages per unit time. Each line represents the pressure drop inside a TURB component cell:

- Sim. Inlet Pressure: TURB component 35, cell 1.
- Sim. Extraction 1 Pressure: TURB component 35, cell 2.
- Sim. Extraction 2 Pressure: TURB component 36, cell 2.
- Sim. Outlet Pressure: TURB component 39, cell 2.

The remaining extractions are not taken into consideration since no real data of them were provided.

Except for the simulated inlet pressure, which is slightly lower than the real one (see table 4.5), the simulated pressure at the extractions and the outlet is very similar to the real one.

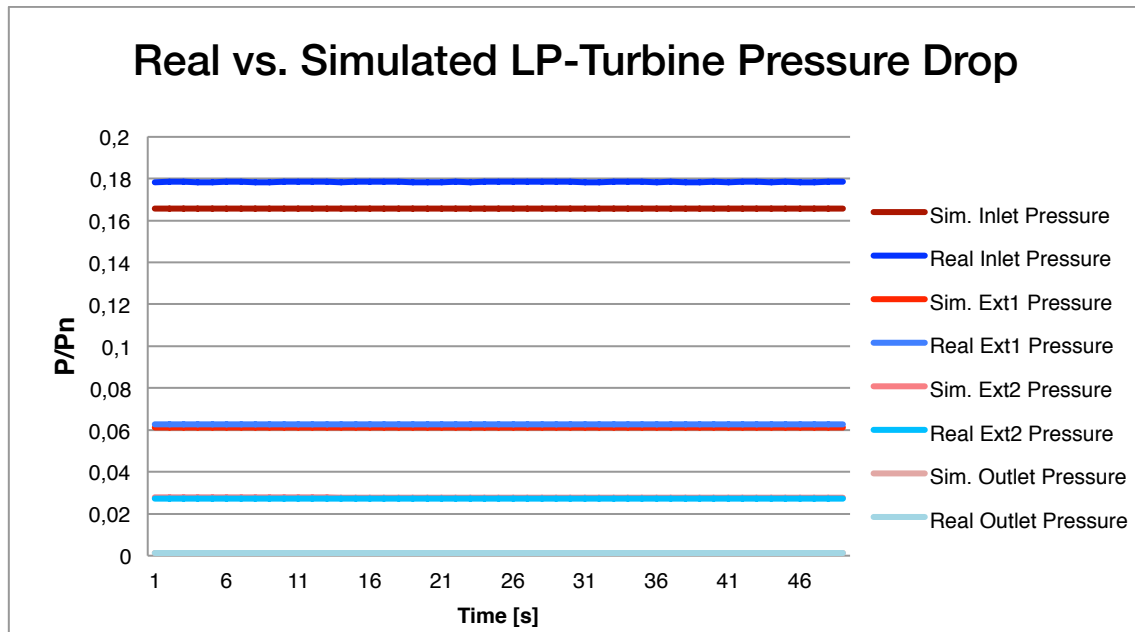


Figure 4.5: Graph-Simulated vs. Real LP-Turbine Pressure Drop

Figure 4.6 shows the LP-Turbine pressure drop per stage. This graph includes the simulated pressure at all the stages, and the real pressure at four stages since no real data were provided of extractions 3 and 4. As the graph shows, the simulated pressure drop follows the same pattern as the real one.

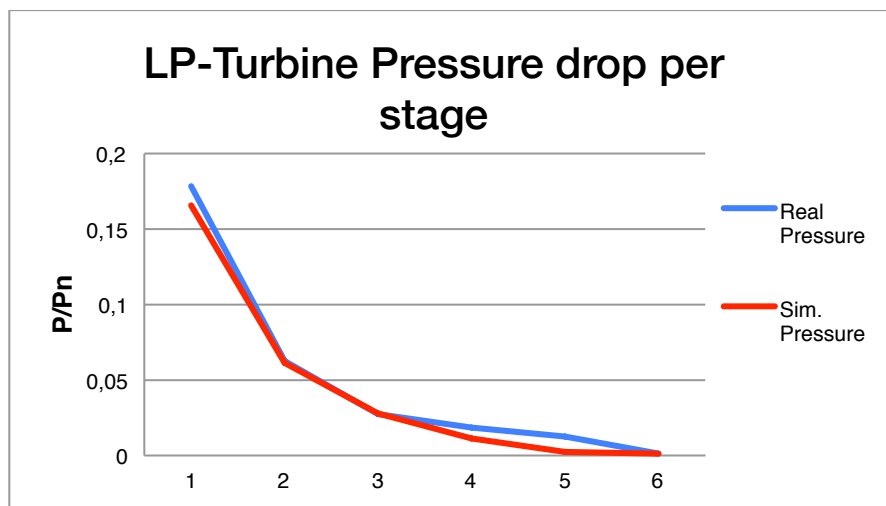


Figure 4.6: Graph-Simulated vs. Real LP-Turbine Pressure Drop per stage

4.2.3 Complete Model

This section includes a more detailed analysis of all the components included in the complete model (see figure 3.8). The main data of each component were extracted from the simulation results

HP-TURBINE

In the complete model, the HP-Turbine corresponds to two turbines (see Figure 3.3). Since the TURB components are placed in parallel (13-17 and 15-19 respectively) just one single turbine results (TURB components 13 and 15) are taken into consideration for the data analysis.

Figure 4.7 shows the real (blue) and simulated (red) pressure inside the HP-Turbine stages per unit time. Each line represents the pressure inside the TURB component cells:

- Sim. Inlet Pressure: TURB component 13, cell 1.
- Sim. Extraction Pressure: TURB component 13, cell 2.
- Sim. Outlet Pressure: TURB component 15, cell 2.

The difference between the simulated and the real outlet pressure is big (see figure 4.8) due to the effect that the separator and the over-heater cause on the system.

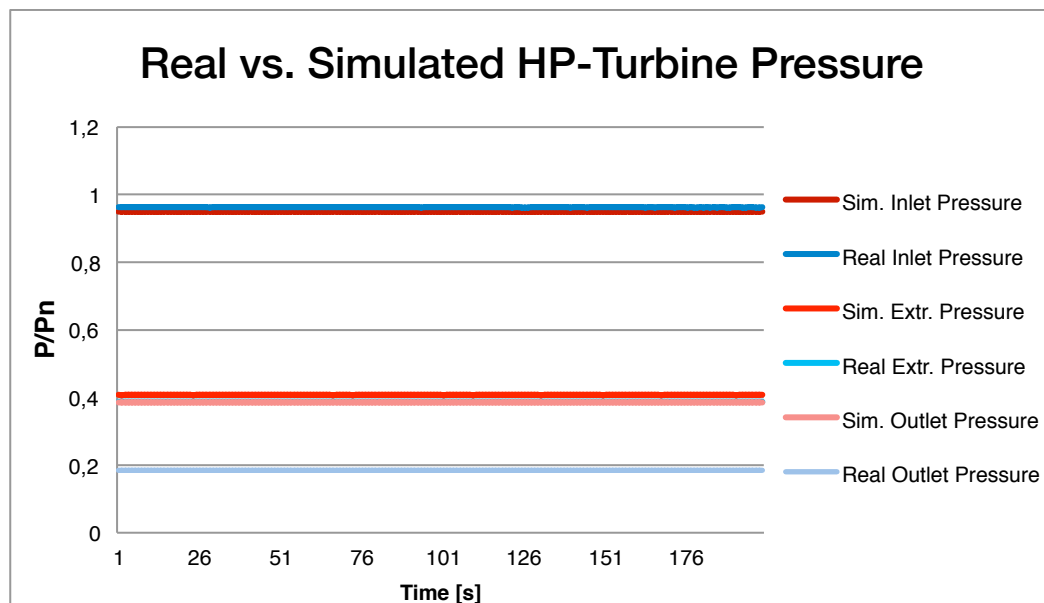


Figure 4.7: Graph-Simulated vs. Real HP-Turbine Pressure

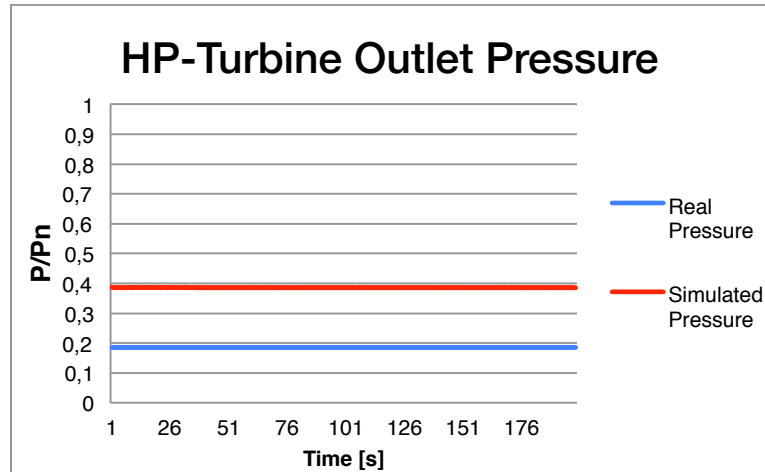


Figure 4.8: Graph-Simulated vs. Real HP-Turbine Outlet Pressure

Figure 4.9 shows the HP-Turbine pressure drop per stage. This graph illustrates that there is hardly any pressure drop between the second and the third stage of the simulated model, while between the real stages the pressure drop is significant.

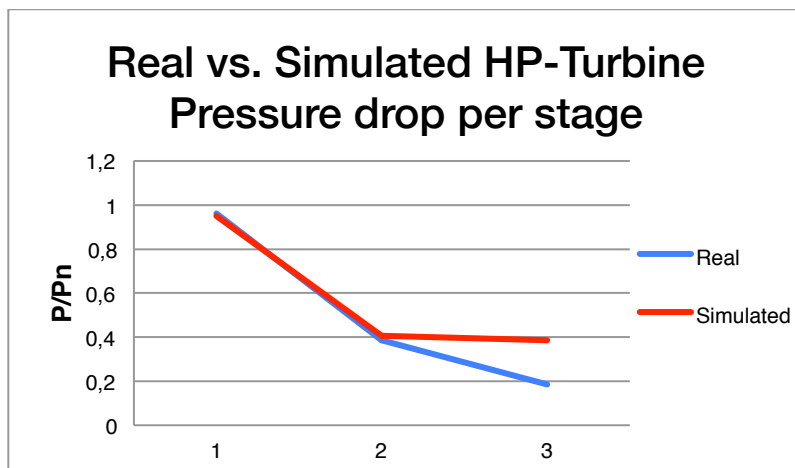


Figure 4.9: Graph-Simulated vs. real HP-Turbine pressure drop

Figure 4.10 shows the progress of the mass flow through the HP-Turbine per unit time. The water extracted in the simulation is lower than the nominal one: 0,0447 [Mf/Mfn] vs. 0,0866 [Mf/Mfn]. The water is only extracted in TURB component 13 because the TURB component 15 side-tube was blocked by the input of a special FRIC array (see section 2.2.2 FRIC and K-Factors). In TURB components 15 and 19 a FRIC value of 10^{21} and -10^{21} was set at edges 4 and 5 respectively.

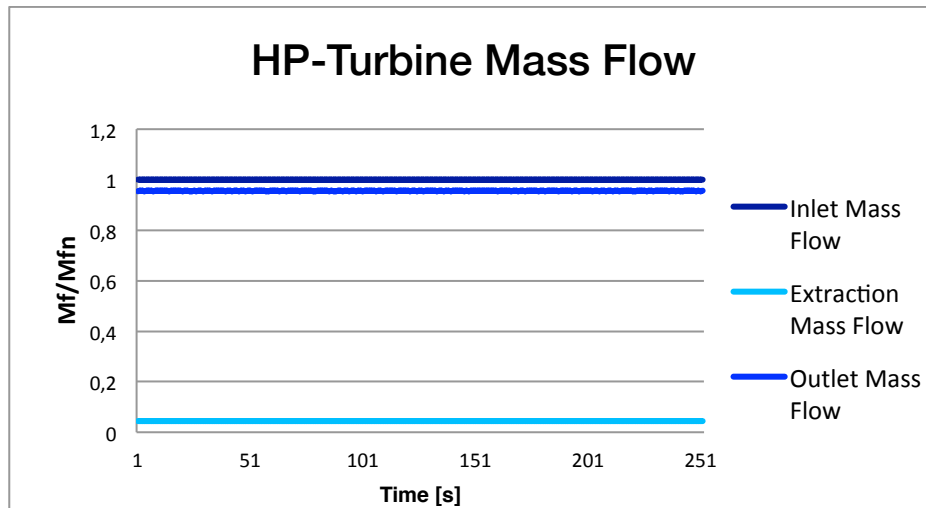


Figure 4.10: Graph-Simulated HP-Turbine Mass Flow

SEPARATOR, OVER-HEATER

The purpose of the separator is to extract the liquid water from the mass flow. Figure 4.11 shows the difference between the inlet mass flow (dark colours) and the outlet mass flow (light colours).

- Inlet total mass flow: total mass flow at SEPD edge 2.
- Inlet vapour mass flow: vapour mass flow at SEPD edge 2.
- Inlet liquid mass flow: liquid mass flow at SEPD edge 2.
- Outlet total mass flow: total mass flow at SEPD edge 3.
- Outlet vapour mass flow: vapour mass flow at SEPD edge 3.
- Outlet liquid mass flow: liquid mass flow at SEPD edge 3.

The extracted liquid water is the difference between the two green lines, which also corresponds to the difference between the blue lines. As the graph shows the liquid mass flow at the separator outlet is near to zero.

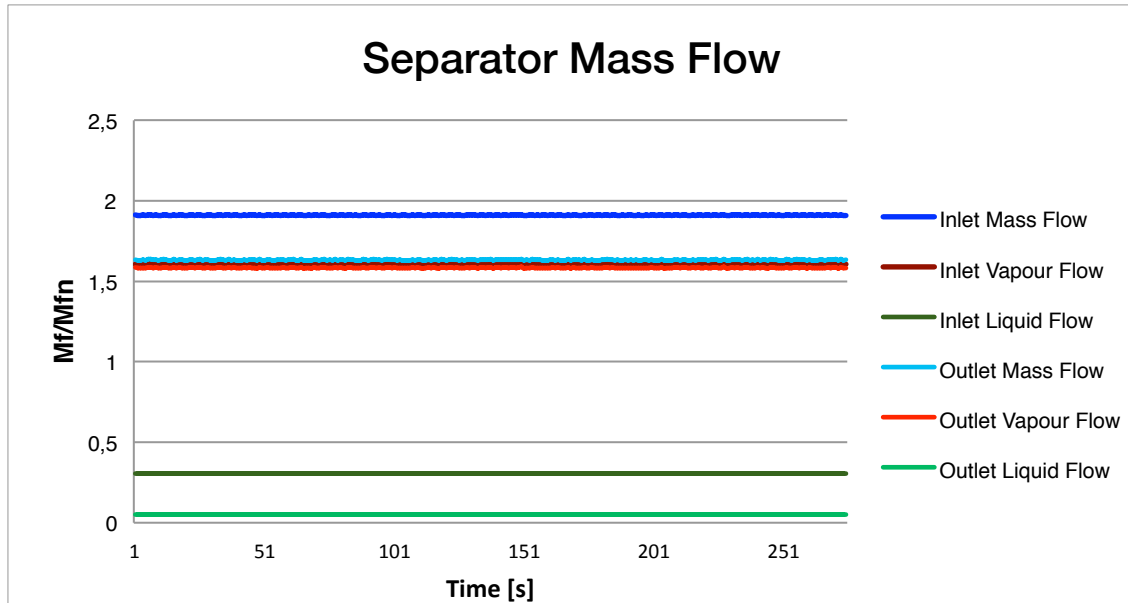


Figure 4.11: Graph-Simulated Separator Mass Flow

Figure 4.12 shows the pressure drop inside the separator per unit time, which is bigger than the expected one.

- Inlet pressure: pressure at SEPD cell 2.
- Outlet pressure: pressure at SEPD cell3.

A considerable pressure drop takes place inside the separator since the TRACE system code requires the definition of a KFAC in between cells 2 and 3 to run.

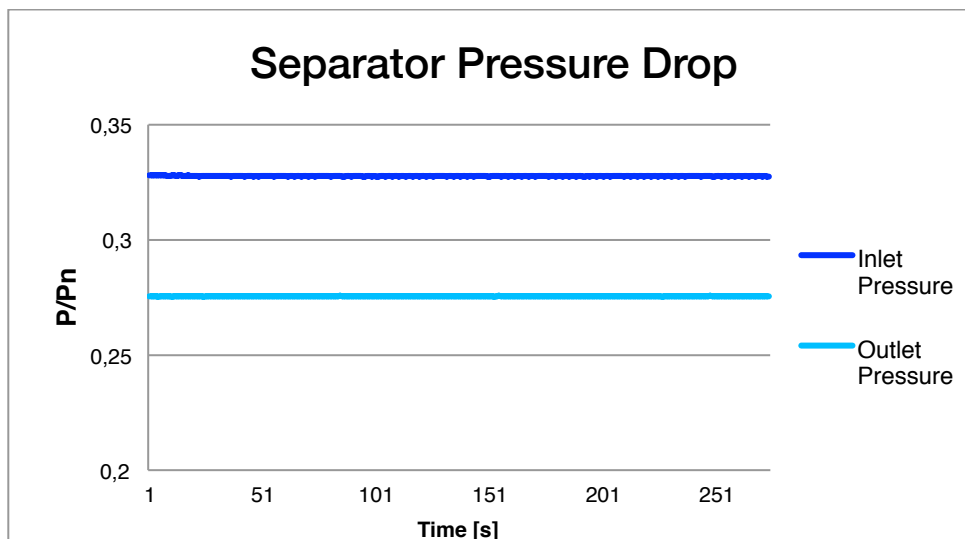


Figure 4.12: Graph-Simulated Separator Pressure Drop

The purpose of the over-heater is to make the fluid reach a certain temperature and vapour quality before it enters the LP-Turbine. Figure 4.13 shows the gas temperature inside PIPE 25 (over-heater), cell 2. Table 4.6 shows the difference between the nominal data and the simulation results, which is very small.

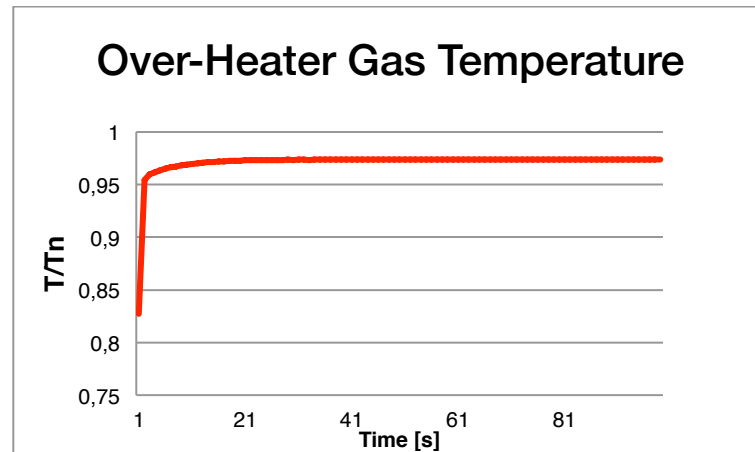


Figure 4.13: Graph-Simulated Over-Heater Gas Temperature at cell 2

	Nominal	Simulated
Temperature	0,9673	0,973625933
Vapour Quality	1	1

Table 4.6: Over-Heater outlet conditions

LP-TURBINE

In the complete model, the LP-Turbine corresponds to six turbines (see Figure 3.6). Since the TURB components of each turbine are placed in parallel, just the results of one turbine (TURB components 35-39) are taken into consideration.

Figure 4.14 shows the real (blue) and simulated (red) pressure inside the LP-Turbine stages per unit time. Each line represents the pressure drop inside a TURB component cell:

- Sim. Inlet Pressure: TURB component 35, cell 1.
- Sim. Extraction 1 Pressure: TURB component 35, cell 2.
- Sim. Extraction 2 Pressure: TURB component 36, cell 2.
- Sim. Outlet Pressure: TURB component 39, cell 2.

The comparison between the real and the simulated pressures reveals that due to the error accumulated in the first part of the complete system (HP-Turbine, separator, and over-heater) the pressure inside the simulated LP-Turbine is higher than the real one. In

spite of reaching a higher pressure, figure 4.15 shows that the pressure drop in the simulated turbine follows the same pattern as the real one.

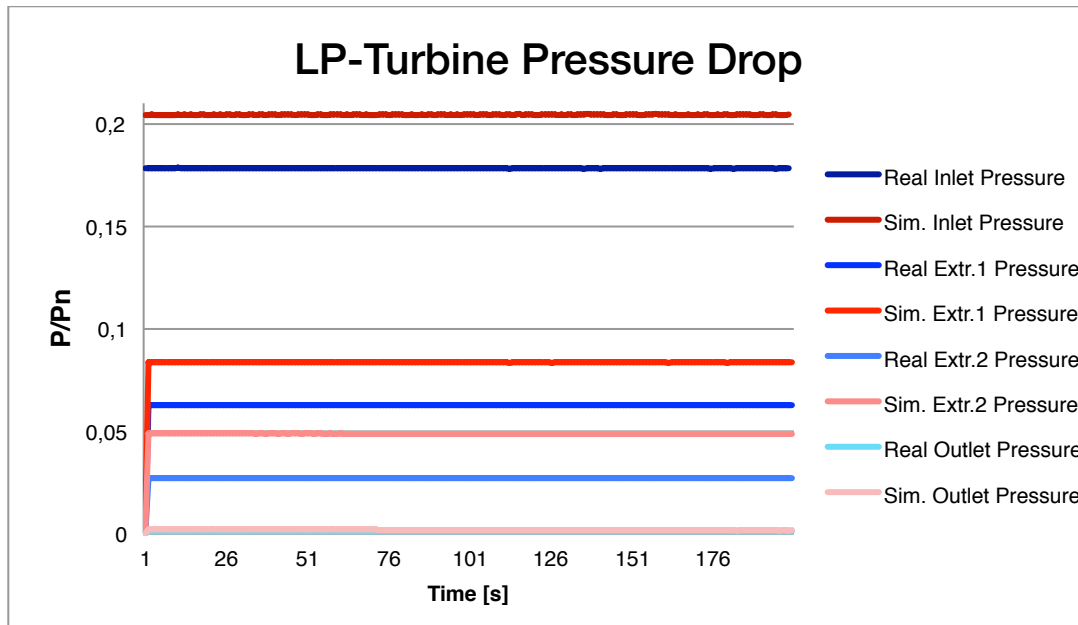


Figure 4.14: Graph-Simulated vs. Real LP-Turbine Pressure

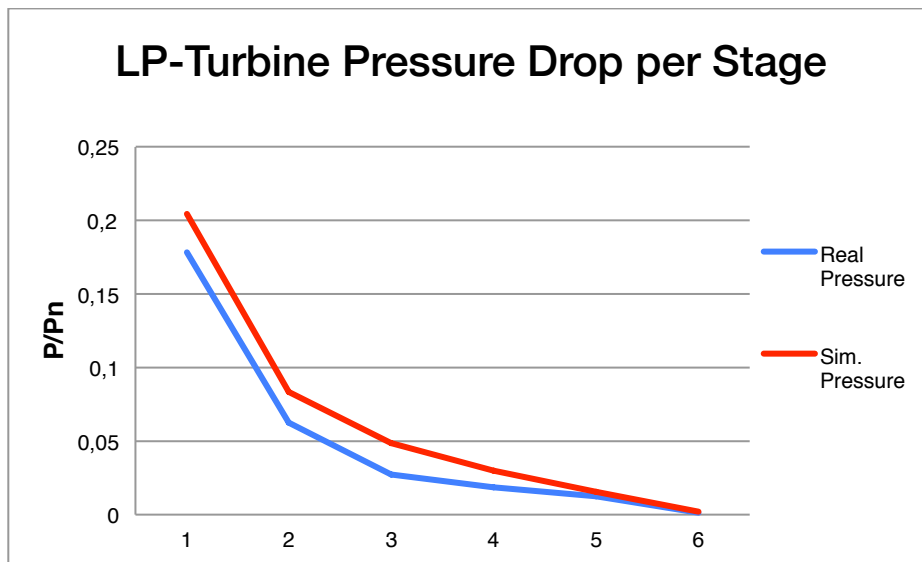


Figure 4.15: Graph-Simulated vs. Real LP-Turbine Pressure Drop per Stage

Figure 4.16 shows the progress of the mass flow through the LP-Turbine per unit time:

- Inlet Mass Flow: total mass flow at TURB 35, edge 1.
- Sum of Extractions: sum of the TURB 35, 36, 37 and 38 extractions.

- Outlet Mass Flow: total mass flow at TURB 39, edge 3.

The water is extracted in TURB components 35, 36, 37 and 38 because the TURB component 39 side-tube was blocked by the input of an especial FRIC array (see section 2.2.2 FRIC and K-Factors). In TURB component 39 a FRIC value of 10^{21} and -10^{21} was set at edges 4 and 5 respectively.

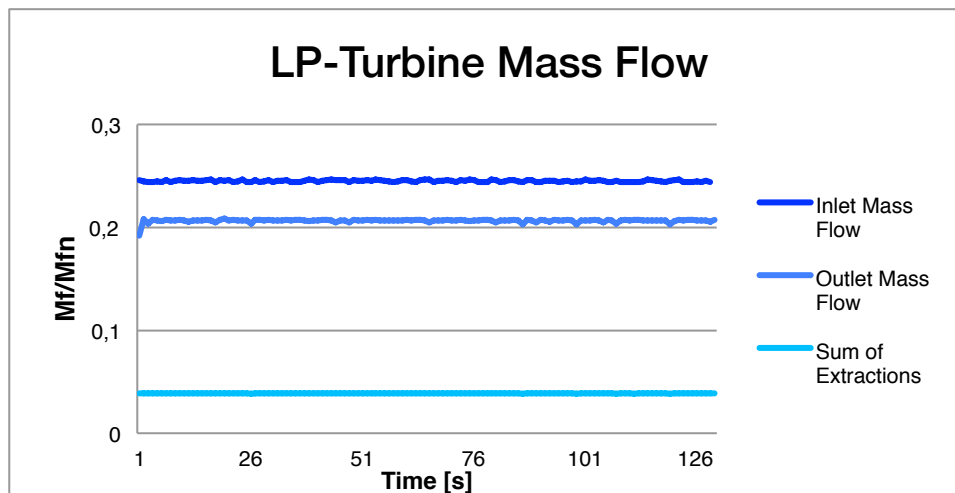


Figure 4.16: Graph-Simulated LP-Turbine Mass Flow

CONDENSER

TRACE system code does not include a specific component to simulate condensers; therefore, a simple PIPE component in combination with a HTSTR was used to simulate a condenser. The aim of this element is to condense the whole vapour mass flow. Figure 4.17 shows how the vapour mass flow is reduced to zero.

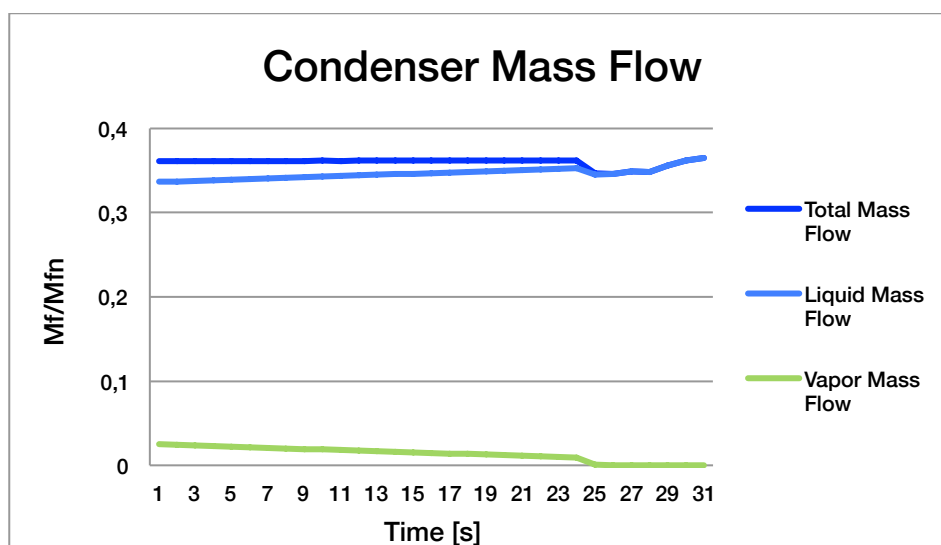


Figure 4.17: Graph-Simulated Condenser Mass Flow

VALVE

In order to simulate the operation of the stop valve a specific model was necessary, in which the pressure at valve cell 2 increases up to 107,5%, TRIP set value (see section 3.1). At this value (red marker in figure 4.19) the TRIP component changes its status from [1]ONforward to [0]OFF since the pressure value at valve cell 2 is in between Setpoints 2 and 4. As the pressure is still increasing, the TRIP component changes its status from [0]OFF to [-1]ONreverse. When the TRIP starts changing its status, the valve starts closing (see figure 4.19). After the valve starts closing, the pressure at valve cell 2 decreases and the TRIP component changes its status from [-1]ONreverse to [0]OFF. As the valve flow area is decreasing, the pressure at valve cell 1 increases rapidly, which makes the simulation crash.

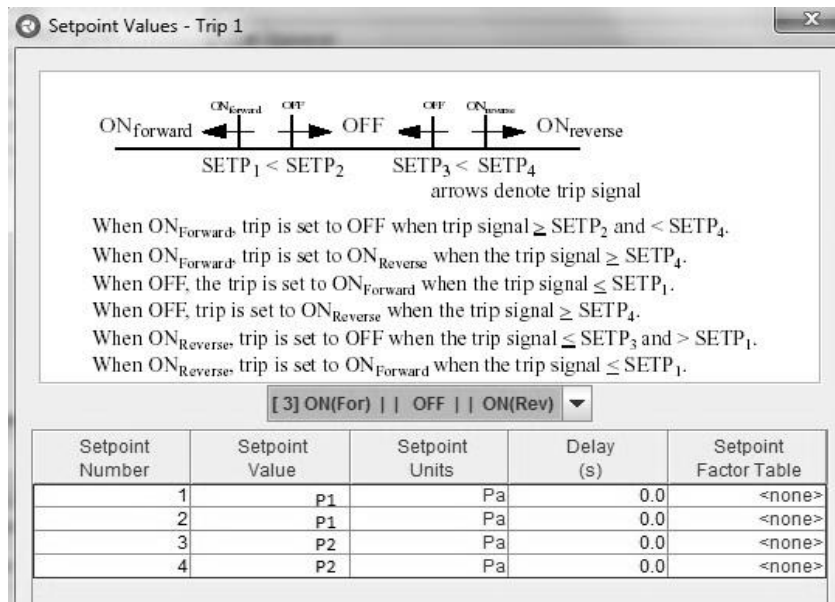


Figure 4.18: TRIP Setpoint Values

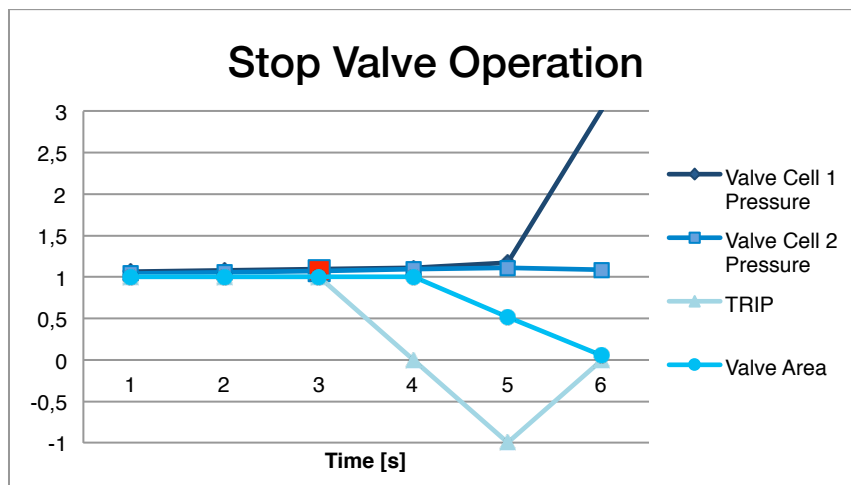


Figure 4.19: Stop Valve Operation (normalized values)

4.3 K-Factors

K-Factors are form-loss coefficients that are required as input by all the TRACE components included in the model.

To model pressure losses in 1D components, TRACE system code includes the friction-factor correlation option flag (NFF). NFF=1 is the correlation option used in the components.

An analysis of the results from simulations with a simple-turbine-model was made (see Figure 4.20) in order to see how the KFACs and the geometry affect the pressure drop inside the components. Three different simulations were run and compared:

- Simulation 1: The first simulation corresponded to a turbine-model with a simple geometry and with no KFAC array defined.
- Simulation 2: The second simulation corresponded to a turbine-model with an increasing flow area from the inlet to the outlet, and with no KFAC array defined.
- Simulation 3: The third simulation corresponded to a turbine-model with a simple geometry and with a KFAC array defined.

Edge	KFAC
1	0
2	200
3	0
4	10
5	10

Table 4.7: KFAC array

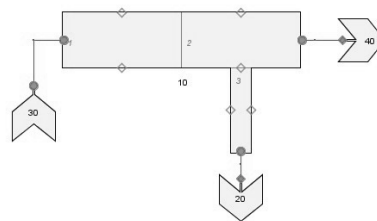


Figure 4.20: TURB component KFAC study

The required conditions in the turbine are:

- Inlet pressure (pressure in cell1) as close as possible to the FILL pressure.
- Outlet pressure (pressure in cell 2) as close as possible to the BREAK pressure.

Results:

	FILL Pressure	Inlet Pressure	Outlet Pressure	BREAK pressure
Simulation 1	1	0,3889	0,3777	0,3776
Simulation 2	1	0,3922	0,38	0,3776
Simulation 3	1	0,895	0,3852	0,3776

Table 4.8: KFAC analysis results

Table 4.8 shows the results of the three simulations, from which is concluded that the KFAC array strongly affects the pressure drop inside the TURB component.

Taking into consideration the conclusion obtained from the analysis, an analytic method was sought to calculate the TURB component KFAC array for the simplified models. Due to the lack of geometrical data many assumptions for the calculation of the KFACs were made. Since the simulation results of the simplified models (see figures 4.1 and 4.4) after inserting the KFACs were not the expected ones, the KFAC array calculated was rejected. Finally, all the KFACs were manually tuned until the pressure drop inside the turbines was as close as possible to the real data.

As the explanation above clarifies (section “4.2 Results”), both the HP- and LP-Turbines were developed separately and then inserted in the same model. The results obtained from the simulation of the complete model were not the expected ones. In order to reach the desired pressure drop inside the HP- and LP-Turbines, the KFAC arrays of the TURBs and of the other components were manually tuned again.

To block the TURB component side-tubes a specific FRIC array is necessary. To insert a FRIC array, the IKFAC NAMELIST option needs to be set to 0. After that, all the KFACs inserted in the model components are automatically transformed into FRIC coefficients. The specific FRIC array that blocks the TURB component extraction only works during a simulation when IKFAC is set to 1.

After revising the results from the simulations and comparing them to the real data, the conclusion is that the KFACs inserted in the model are optimal since the results are very close to the expected ones.

5. Transient: Simulation & Results

To run a transient simulation the option [1] Transient must be set. The model used to run transients is the one explained in chapter three. Three different transients were run; hence, three different model setups were necessary.

This chapter includes the model modifications and the transient results, which correspond to the HP- and LP-Turbine pressure.

5.1 Modifications

5.1.1 FILL

The transient was defined in the FILL component. The FILL type used in the transient simulation was: [5] mass flow vs. independent-variable from the table.

The transient was defined by inserting a specific mass flow distribution in the FILL table. The inserted mass flow corresponds to the real transient mass flow, and the independent variable was the time.

5.1.2 NFRC1 NAMELIST variable

In the three simulated transients the pressure drops up to very low values. In order to avoid water flowing from the BREAK components into the system, the NFRC1 NAMELIST variable was set to 2 (see Chapter 2- FRIC and KFAC). When NFRC1 is set to 2, the dimension of the FRIC array is doubled to $2 \times (\text{NCELLS} + 1)$, the first array provides loss coefficients that are used with positive velocities, and the second array provides loss coefficients that are used with negative velocities.

In order to block the water flowing from BREAK components (negative velocities) to the system, the second FRIC array of the all the TURB side-tubes (edges 4 and 5) was set to 10^{21} and -10^{21} . For the SEPD and the remaining PIPEs, the second FRIC array was set to 10^{20} and -10^{20} on the edges connected to the BREAKs.

5.2 Transient 1: Definition and results

Transient 1 simulates a smooth pressure drop in the system. The system power decreases from 100% to 60% (approximately) in 800 seconds. To produce the required pressure drop, the FILL component table imposes a smooth mass flow change, which follows the same pattern as the transient.

FILL Table

To run the first transient, the table input in the FILL component is Table 5.1. The first part of the transient simulation lasts 2200 seconds, in which the mass flow is constant, since it is the time necessary to reach the steady state conditions. After that time, the mass flow decreases linearly from 2 to 1.194.

Time [s]	Mass Flow
0	2
2200	2
3001	1,194
3200	1,178

Table 5.1: Transient 1 FILL description

HP-Turbine

Figure 5.1 shows the HP-Turbine inlet mass flow, which corresponds exactly to half of the FILL mass flow.

A comparison between figures 5.1 and 5.2 reveals that both the pressure and the mass flow follow the transient pattern, which proves that there is a close relationship between pressure and mass flow. It is important to note that, as the data are normalized, both graphs start at 1 and end at 0.6.

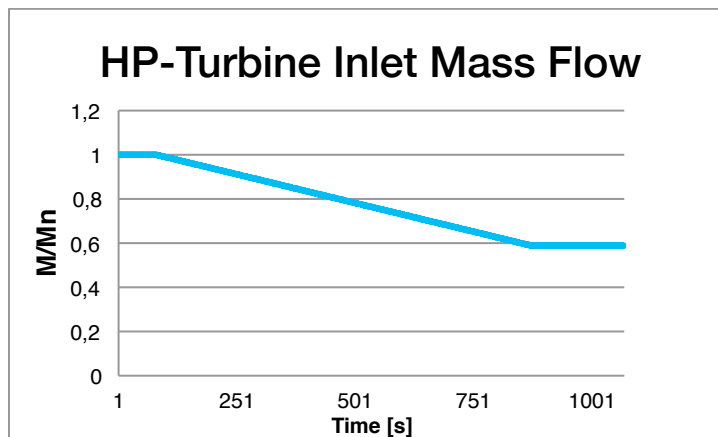


Figure 5.1: HP-Turbine inlet mass flow

Figure 5.2 shows the pressure at the first stage of the HP-Turbine. The pressure obtained at the inlet of the HP-Turbine is very close to the real one. Taking into consideration that the only input that describes the transient is the FILL table, this result is considered realistic.

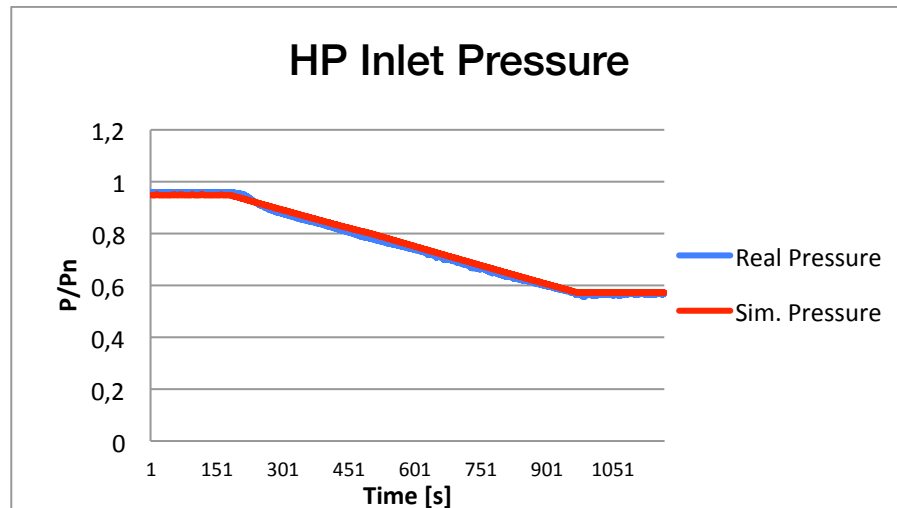


Figure 5.2: Real vs. Simulated HP-Turbine Inlet pressure

Figure 5.3 shows the pressure at the extraction and outlet. As it was expected from the observation of the Steady-State simulation results, the pressure at the outlet of the simulated HP-Turbine is higher than the real one. Despite of the higher pressure, the graph shows that both, the pressure at the extraction and at the outlet, follow the same pattern as the real pressure.

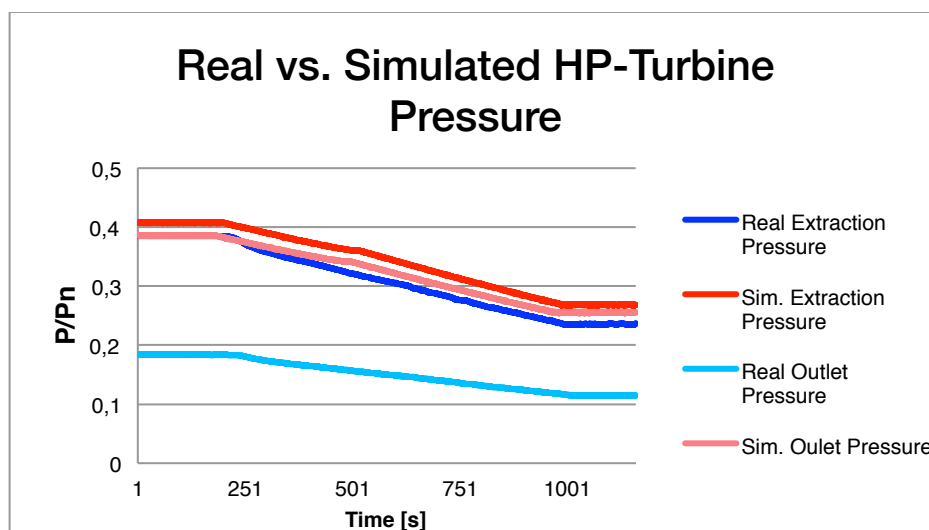


Figure 5.3: Real vs. Simulated HP-Turbine pressure

Figure 5.4 shows the mass flow at the HP-Turbine extraction, which decreases at the same rate as the HP-Turbine inlet mass flow does. The extracted mass flow becomes 0 when the extraction pressure (see figure 5.5) reaches the same value as the one defined in the break that is connected to the turbine, 0.359 [P/Pn] (see table 3.8). Since the pressure at the turbine extraction is still decreasing, the mass flow inside the BREAK component tends to flow towards the turbine. The reverse flow FRIC array inserted (see

section 5.1.2) blocks the flow from the break to the turbine. Due to the blocking, the extracted mass flow remains constant (0) until the end of the simulation.

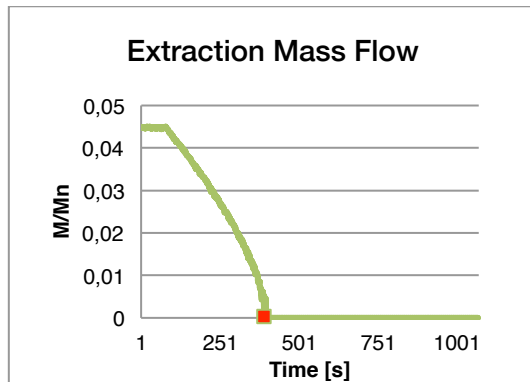


Figure 5.4: HP-Turbine Extraction mass flow

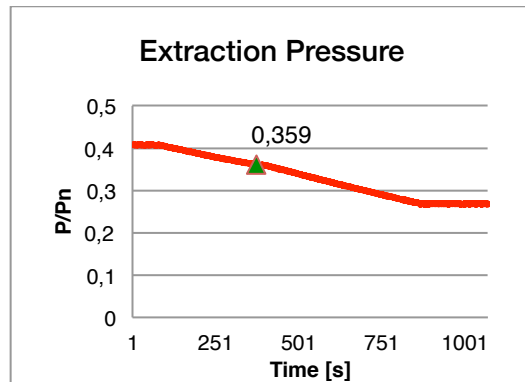


Figure 5.5: HP-Turbine Extraction pressure

LP-Turbine

Figure 5.6 shows the mass flow at the LP-Turbine inlet, which follows the transient pattern distinctly. Compared to the HP-Turbine inlet mass flow (see figure 5.1), the LP-Turbine inlet mass flow is slightly more fluctuating due to the disturbances produced by the other components.

A comparison between figures 5.6 and 5.7 reveals that both the pressure and the mass flow follow the transient pattern, which proves that a close relationship between pressure and mass flow exists. The results of the LP-Turbine simulation are remarkable since they reveal that the transient pattern is followed through the whole system, which ratifies the validity of the design model.

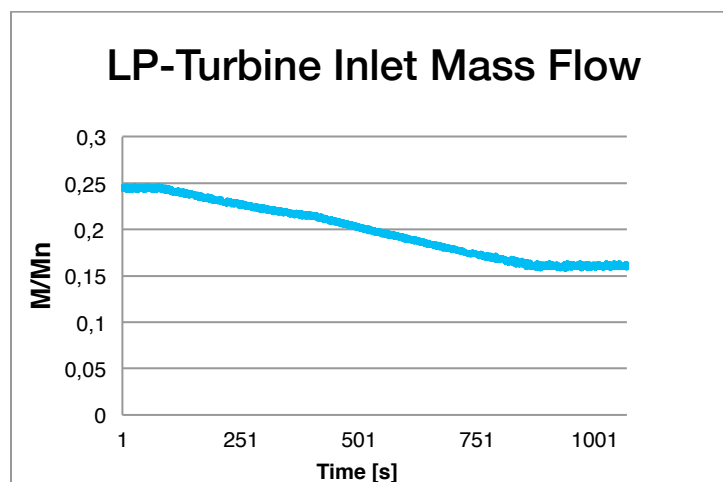


Figure 5.6: LP-Turbine inlet mass flow

Figure 5.7 shows the pressure at the inlet of the LP-Turbine. The simulated pressure is higher than the real pressure, as it is in the steady state simulation. The simulated pressure follows the same pattern as the real one, which was the main objective.

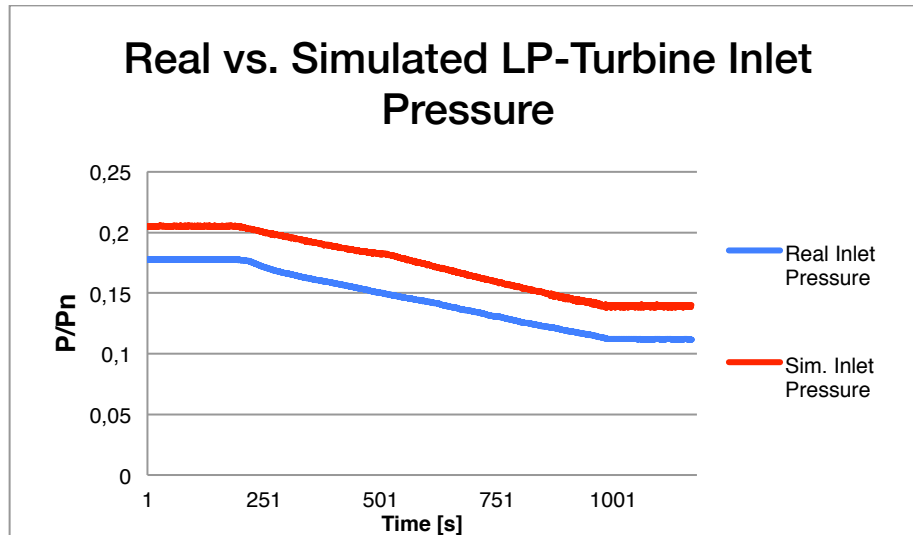


Figure 5.7: Real vs. Simulated LP-Turbine inlet pressure

Figure 5.8 shows that the difference between the real and the simulated pressure remains constant at the extractions, but again the transient pattern is followed by the pressure.

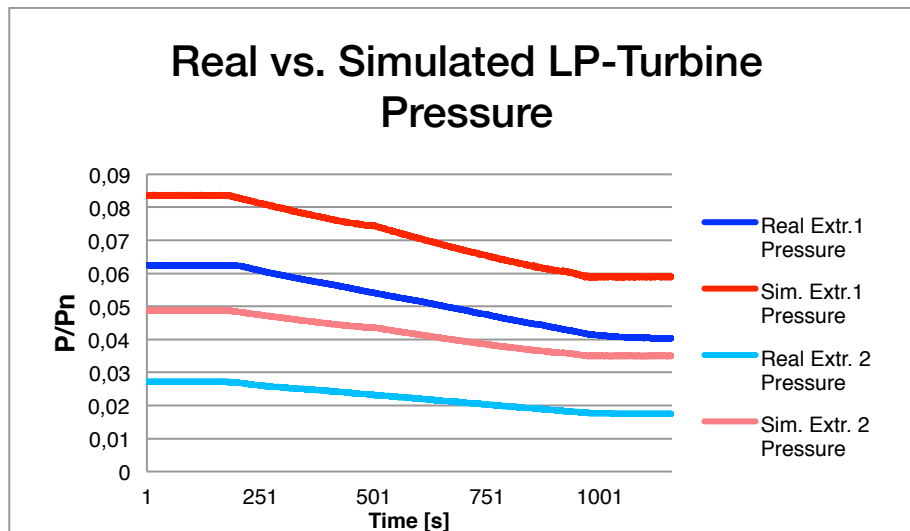


Figure 5.8: Real vs. Simulated LP-Turbine pressure

As figure 5.9 shows, the real pressure does not follow the transient pattern as clear as the other pressure measurements do, which, in addition to the effect caused by the

condenser pressure, affect the LP-Turbine outlet pressure, which differs slightly from reality.

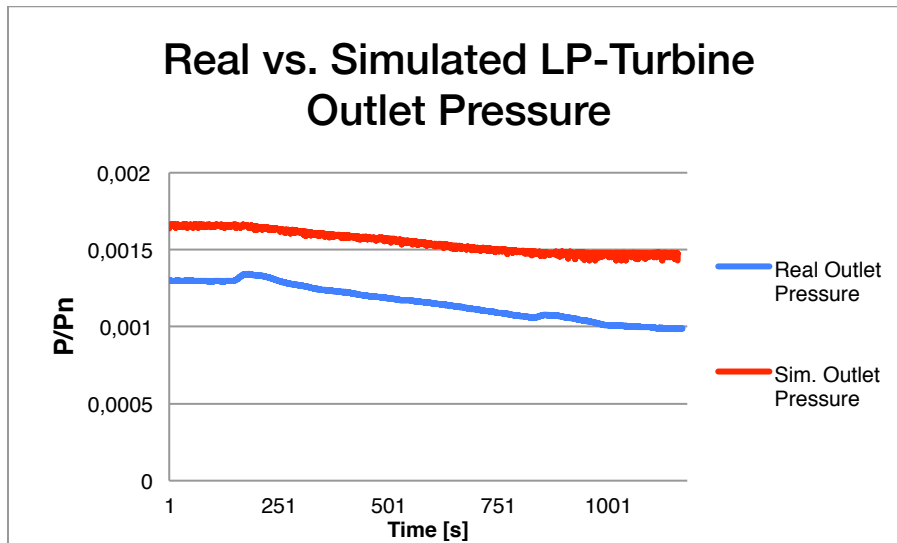


Figure 5.9: Real vs. Simulated LP-Turbine outlet pressure

5.3 Transient 2: Definition and results

Transient 2 simulates a smooth pressure increase in the system. The system power increases from 60% to 100% (approximately) in 860 seconds. To produce the required pressure drop, the FILL component table imposes a smooth mass flow change, which follows the same pattern as the transient.

FILL Table

To run the second transient, the table input in the FILL component is Table 5.2. The first part of the transient simulation lasts 2210 seconds, in which the mass flow is constant, since is the time necessary to reach the steady state conditions. After that time, the mass flow increases linearly from 1.184 to 2.01.

Time [s]	Mass Flow
0	1,184
2210	1,184
3070	2,010
3200	2,010

Table 5.2: Transient 2 FILL description

HP-Turbine

Figure 5.10 shows the HP-Turbine inlet mass flow, which corresponds exactly to half of the FILL mass flow.

A comparison between figure 5.10 and figure 5.11 reveals that both the pressure and the mass flow follow the transient pattern, which proves that there is a close relationship between pressure and mass flow. It is important to note that, as the data are normalized, both graphs start at 0.6 and end at 1.

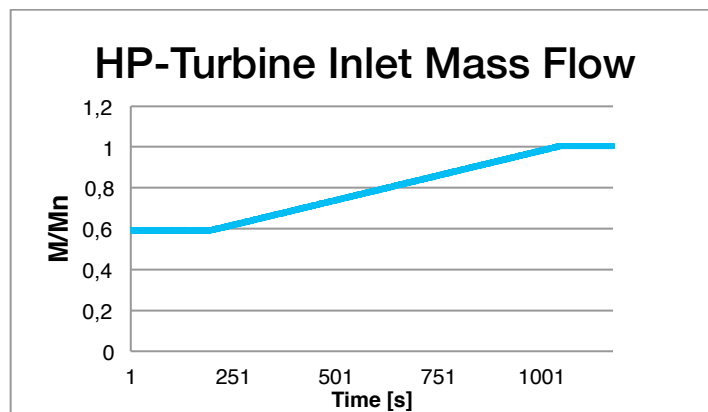


Figure 5.10: HP-Turbine inlet mass flow

Figure 5.11 shows the real (blue) and simulated (red) HP-Turbine inlet pressure. The comparison between both pressures reveals that the turbine inlet conditions are realistic. These results are remarkable since the transient starts at a lower pressure value and increases until the nominal one in a realistic way. It also should be noted that the only input to describe the transient pattern is the mass flow distribution described in the FILL table.

Figure 5.12 shows the simulated HP-Turbine outlet pressure, which is higher than the real one due to the effect caused by the other system components.

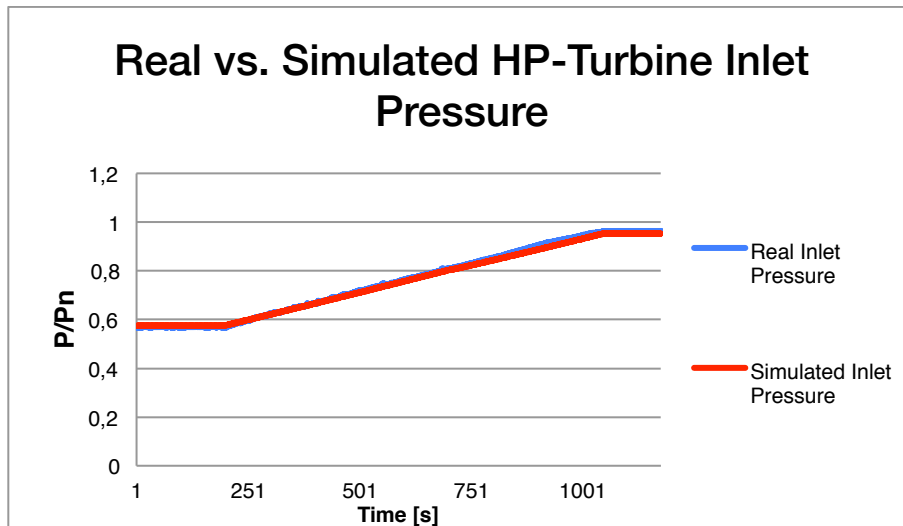


Figure 5.11: Real vs. Simulated HP-Turbine inlet pressure

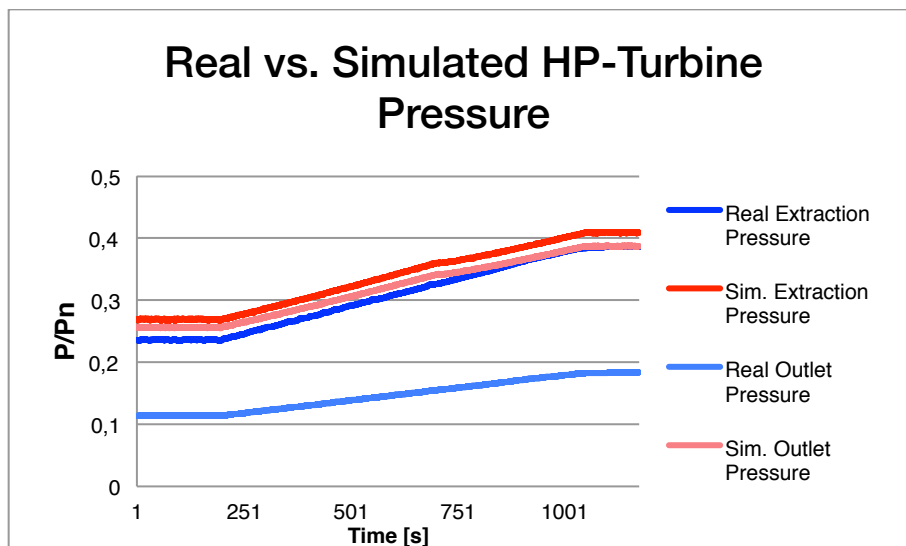


Figure 5.12: Real vs. Simulated HP-Turbine pressure

Figure 5.13 shows the mass flow at the HP-Turbine extraction, which is zero at the beginning of the simulation due to the reverse flow FRIC array inserted to block negative flow. The fluid tends to flow from the break towards the turbine since the pressure at the turbine extraction is lower than the pressure at the BREAK component. The extraction mass flow remains constant until the pressure at the extraction (see figure 5.14) reaches the same value as the one defined in the break, 0.359 [P/Pn] (see table 3.8). After that, the fluid starts flowing towards the break following the transient pattern.

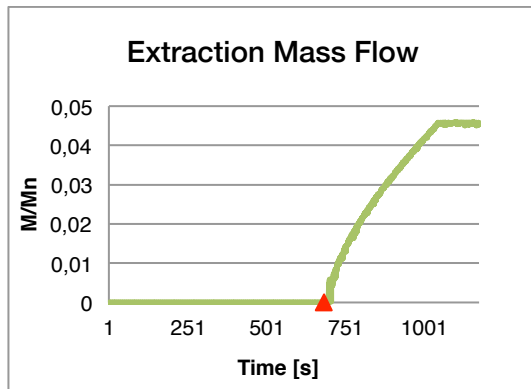


Figure 5.13: HP-Turbine Extraction mass flow

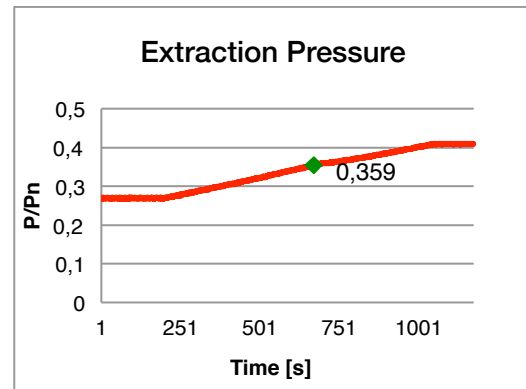


Figure 5.14: HP-Turbine Extraction pressure

LP-Turbine

Figure 5.15 shows the mass flow at the LP-Turbine inlet, which follows the transient pattern distinctly. Compared to the HP-Turbine inlet mass flow (see figure 5.10), the LP-Turbine inlet mass flow is slightly more fluctuating due to the disturbances produced by the other components.

A comparison between figures 5.15 and 5.16 reveals that both the pressure and the mass flow follow the transient pattern, which proves that a close relationship between pressure and mass flow exists. The results of the LP-Turbine simulation are remarkable since they reveal that the transient pattern is followed through the whole system.

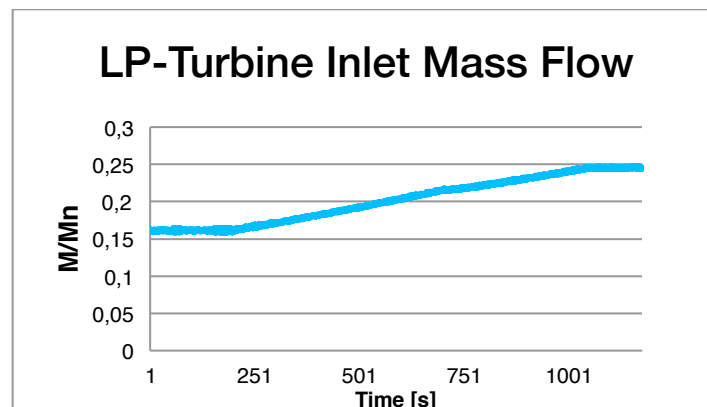


Figure 5.15: LP-Turbine inlet mass flow

Figure 5.16 shows a comparison between the real and the simulated LP-Turbine inlet pressure. The simulated pressure follows the transient pattern as the real one, which was the main objective of the simulation. Due to the error accumulated in the first part of the complete system, the simulated pressure is higher than the expected one.

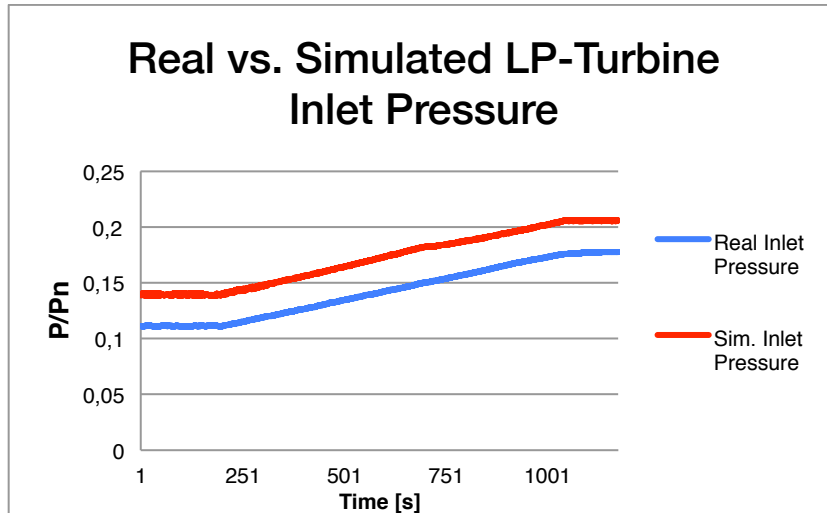


Figure 5.16: Real vs. Simulated LP-Turbine inlet pressure

Figure 5.17 shows a comparison between how the simulated pressure and the real pressure evolve inside the LP-Turbine, and figure 5.18 illustrates that the LP-Turbine outlet pressure does not follow the transient pattern as clearly as the inlet pressure.

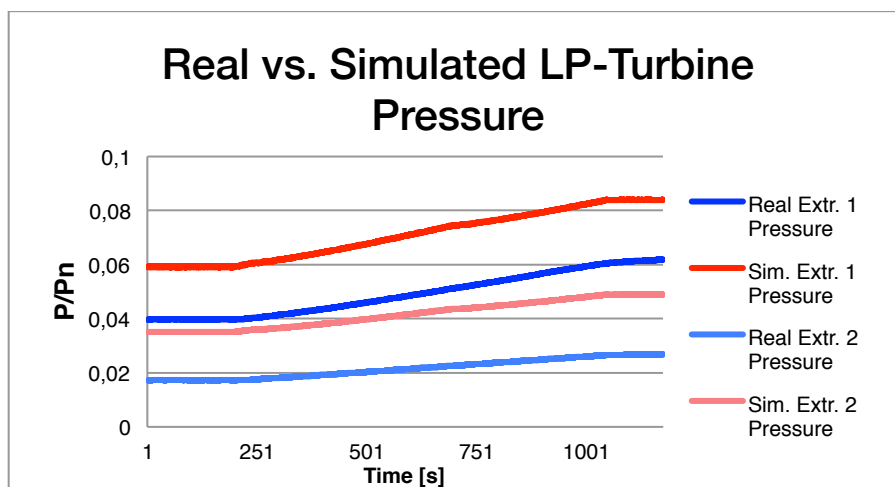


Figure 5.17: Real vs. Simulated LP-Turbine pressure

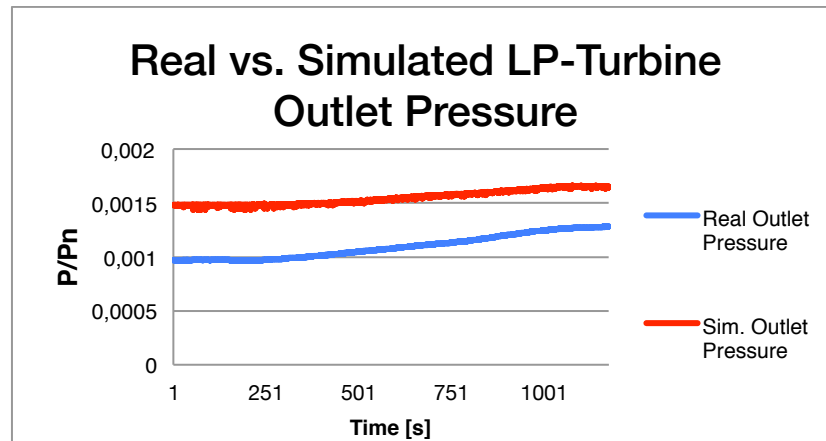


Figure 5.18: Real vs. Simulated LP-Turbine pressure

5.4 Transient 3: Definition and results

Transient 3 simulates a pressure drop and a pressure increase. First of all, the pressure decreases from 90% to 60%, remains constant, at 60%, and finally it increases up to 100%. This transient pattern is difficult to follow, since the pressure is changing in both directions. To produce this pressure drop, the FILL component table imposes a mass flow change, which follows the same pattern as the pressure.

FILL Table

To run the third transient, the table input in the FILL component is Table 5.3. The first part of the transient simulation lasts 1880 seconds, in which the mass flow is constant, since is the time necessary to reach the steady state conditions. After that time, the mass flow decreases linearly from 1.793 to 1.202 in 820s. After that, the mass flow increases up to 1.208 in 70 seconds, and remains constant for 1620s. Finally the mass flow increases from 1.208 to 2.010 in 1170s, and remains constant for the last 430s.

Time [s]	Mass Flow
0	1,793
1880	1,793
2700	1,202
2780	1,208
4400	1,208
5570	2,010
6000	2,010

Table 5.3: Transient 3 FILL description

HP-Turbine

Figure 5.19 shows the HP-Turbine inlet mass flow, which corresponds exactly to half of the FILL mass flow.

Once again, the comparison between the simulated inlet mass flow and inlet pressure (see figures 5.19 and 5.20) reveals the close relation that exists between them. As the graphs show, both the inlet pressure and the inlet mass flow follow clearly the transient pattern.

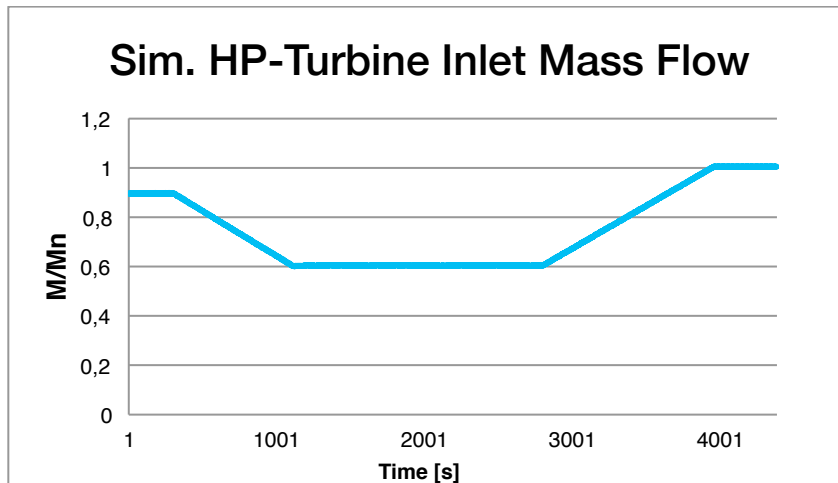


Figure 5.19: Real vs. Simulated HP-Turbine inlet pressure

Figure 5.20 shows a comparison between the simulated (red) and the real (blue) HP-Turbine inlet pressure. As the graph illustrates, the simulated pressure is exactly the same as the real one, even though the transient pattern is complex. Considering the complexity of the pattern, and the real and simulated pressure analogy, this result confirms the validity of the designed system.

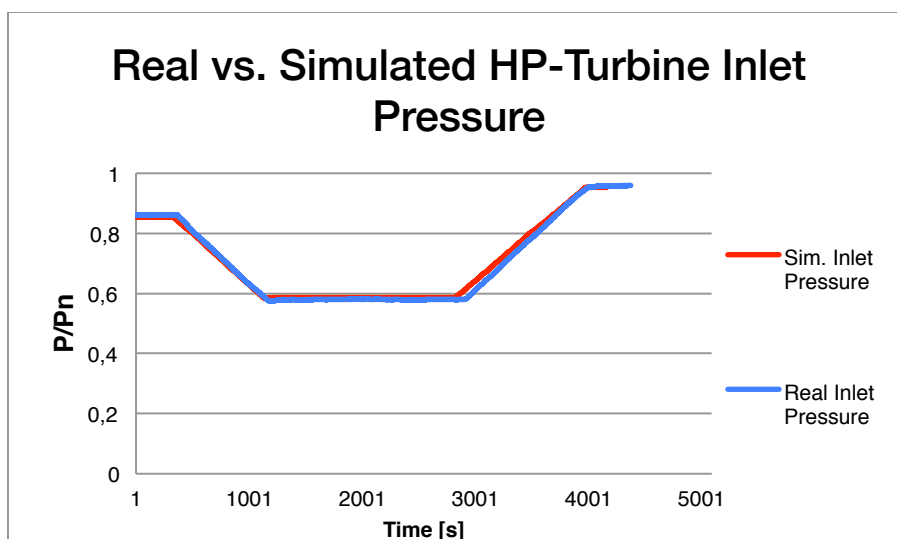


Figure 5.20: Real vs. Simulated HP-Turbine inlet pressure

Figure 5.21 shows the pressure at the HP-Turbine extraction and outlet. As it was explained with the results of transients 1 and 2, the outlet pressure is higher than the desired one due to the effect caused by the other components.

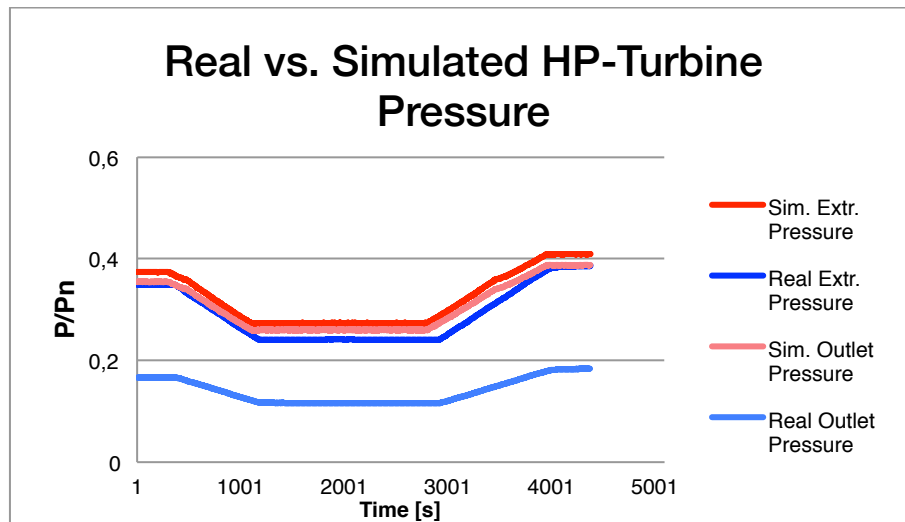


Figure 5.21: Real vs. Simulated HP-Turbine pressure

Figures 5.22 and 5.23 show the progress of the mass flow and the pressure at the HP-Turbine extraction over the transient simulation. A comparison between both graphs demonstrates that when the pressure at the extraction is higher than the break pressure, 0.359 [P/Pn], the extracted mass flow is positive. On the other hand, when the pressure at the extraction is lower than the break pressure, the extracted mass flow is zero. The results obtained from the comparison between figures 5.22 and 5.23, verify the proper function of the reverse flow FRIC array inserted in the model (see section 5.1.2).

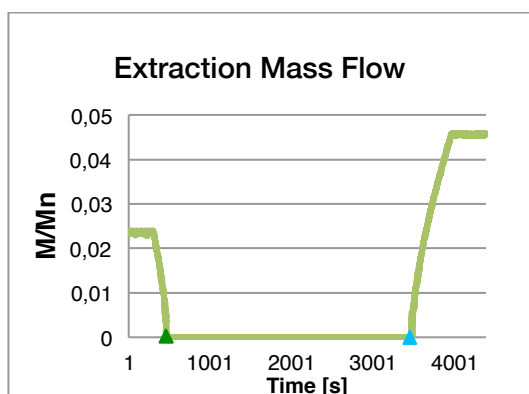


Figure 5.22: HP-Turbine Extraction mass flow

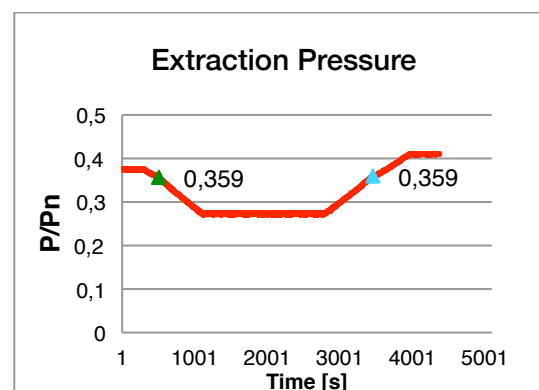


Figure 5.23: HP-Turbine Extraction pressure

LP-Turbine

Figure 5.24 shows the mass flow at the LP-Turbine inlet, which follows the transient pattern clearly. The LP-Turbine inlet mass flow fluctuates more than the HP-Turbine inlet mass flow (see figure 5.19) due to the disturbances produced by the vapour flow through the system components.

A comparison between figures 5.23 and 5.24 shows that, both the pressure and the mass flow follow the transient pattern. The results of the LP-Turbine simulation are remarkable since they reveal that the transient pattern is followed through the whole system.

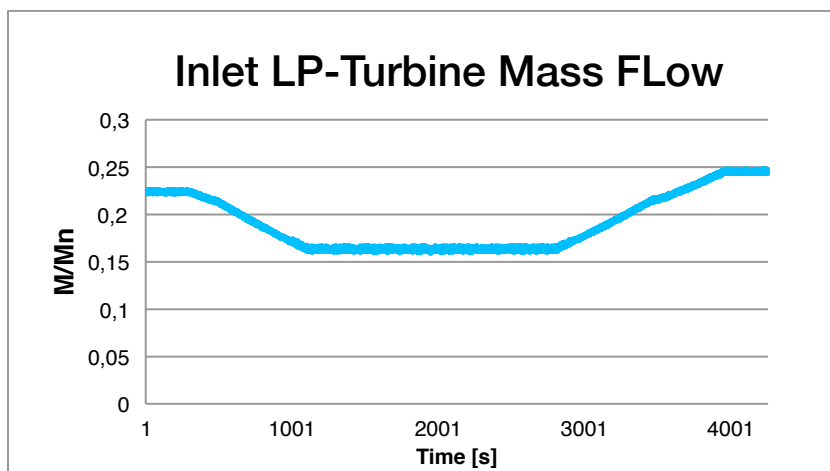


Figure 5.24: LP-Turbine inlet mass flow

Figure 5.25 shows a comparison between the real (blue) and the simulated (red) LP-Turbine inlet pressure. As the graph illustrates both the real and the simulated pressure follow the transient pattern, which also verifies, as the results of the HP-Turbine do, the validity of the designed system.

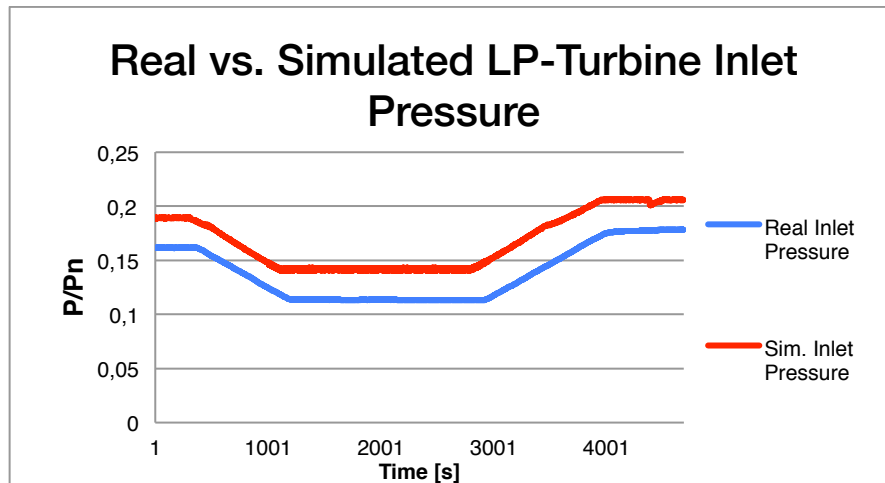


Figure 5.25: Real vs. Simulated LP-Turbine inlet pressure

Figures 5.26 and 5.27 show the pressure at the two first extractions and at the outlet of the LP-Turbine respectively. Even though the simulated pressure is higher than the real one, both graphs illustrate that the transient pattern is followed in all the pressure stages.

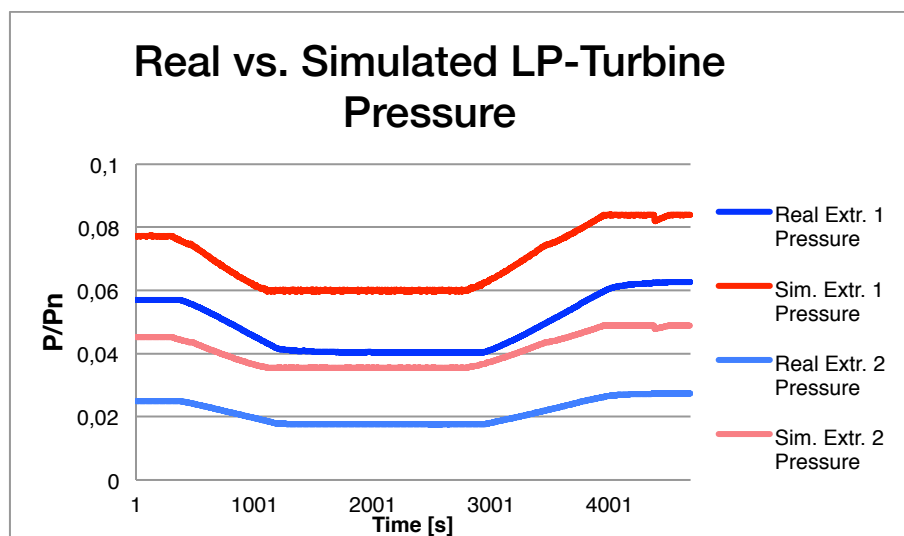


Figure 5.26: Real vs. Simulated LP-Turbine pressure

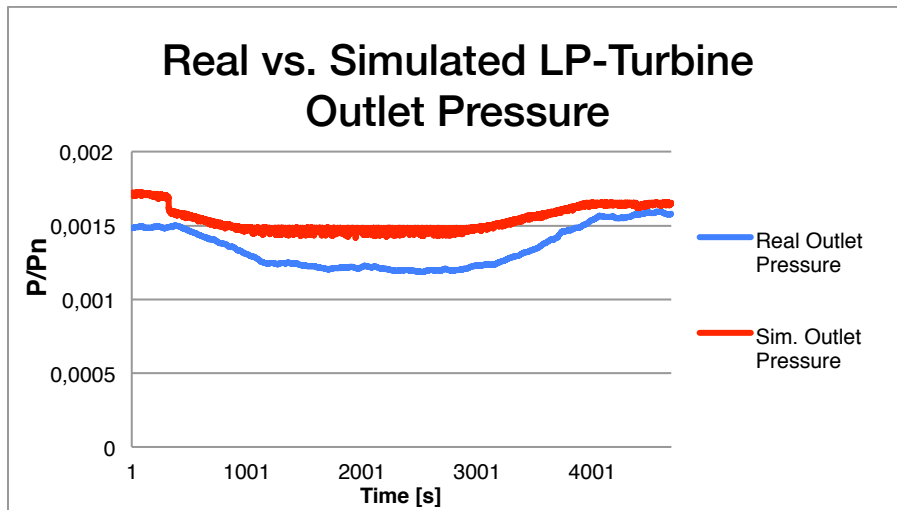


Figure 5.27: Real vs. Simulated LP-Turbine outlet pressure

6. Conclusions

The main objective of this Master's Thesis was to simulate the operation of a NPP steam turbine in order to obtain realistic results. Using the TRACE system code as a working tool, part of a NPP secondary loop model was developed. The two main characteristics to be achieved by the designed model were:

- Reach realistic conditions at the turbine inlet.
- Reproduce a realistic pressure drop throughout the model.

The developed model has been used to run steady-state and transient simulations, from which relatively good results were obtained. The criterion used to determine whether the results are acceptable or not is their comparison to real data.

As it is explained in this document, after developing and improving the model, the steady-state model was run. The analysis of the steady-state results was divided in three different sections:

- Simplified HP-Turbine
- Simplified LP-Turbine
- Complete model

The simplified HP- and LP-Turbine models were developed in order to test the capabilities of the turbine TRACE model using boundary conditions from the turbine design data. After simulating both models, the results were collected and compared to the real turbine data. The data analysis focussed on the pressure drop inside the turbine, since the FILL component imposed the turbine inlet conditions. The simulation results and the real data show good agreement.

After analysing the simplified model data, the complete model was implemented.

Again, the results obtained from the simulation were compared to the turbine real data. As the graphs included in section 4.2.3 illustrates, the results are not as good as those of the simplified models. The slightly difference between the simulation and the real data is caused by the fact that the code has to compensate for the losses from the other components such as separator and over-heater. For instance, graph 4.11 illustrates that inside the separator a significant pressure drop takes place due to the extraction of liquid water, which affects the HP-Turbine outlet pressure.

Figures 4.7 and 4.13 illustrate a comparison between the simulation and real pressure data at the inlet of the HP- and the LP-Turbine respectively. Also, figures 4.9 and 4.13 show a comparison between the simulated and the real pressure drop inside both

turbines. Considering the similarity between the simulation and the real data, the complete model data analysis ratifies the validity of the designed system.

Section 4.2.3 also includes the results of the separator, the over-heater, and the condenser, since part of the work covered by this Master's Thesis was the improvement of the complete model operation in order to reach optimal conditions at the HP- and LP-Turbines inlet.

As previously mentioned, after the optimal results for the steady-state simulation were achieved, three different transient models were simulated. The results obtained from the transient simulations are very remarkable.

The objectives to achieve were reaching realistic conditions at both turbine inlets and reproducing a realistic pressure drop inside the turbine. A comparison between the simulation and the real data at both turbines reveals that both the inlet pressure and the pressure drop in the turbines are near to reality. This was valid for both the steady state and transient simulations.

7. Possible future improvements

The model developed here can be further improved regarding the following aspects:

- Optimization of KFAC array
- Implementation of the HEATR component
- Complete the model by closing the loop

Optimization of the KFAC arrays

The biggest problem encountered during the model design was the definition of the KFAC array of each component. The first attempt to define the KFAC array was an analytical calculation. Due to the lack of real data and as a result of all the approaches tried, the calculated KFAC array did not provide the expected results. Based on the previous calculations, the k-factors of each component were manually tuned until the obtained results were considered close enough to the real data.

But through more modifications of the KFAC array aspects such as pressure drop or turbine water extractions can be further optimized.

HEATR component implementation

The HEATR component provides you with the capability of modelling typical feed-water heaters found in nuclear steam supply systems. A HEATR can be used to model the Shell (condensing) side of a closed-type feed-water heater, and is meant to be coupled to a PIPE and HTSTR to include modelling of the feed-water flow and heat transfer within the heaters. A HEATR (or a series thereof) component can also be employed to model the main steam condensers, again in combination with a PIPE and HTSTR, which in this case would model the circulating cooling water [8].

During the development of the complete model, the HEATR component was chosen for the designing of the over-heater and the condensers. Due to the limited timeframe of the project the decision was to design simplified models for both components in order to reach the desired conditions at the inlet and outlet of the LP-Turbine.

With enough time and the proper data, a realistic model of an over-heater or a condenser can be developed using a HEATR component coupled to a PIPE and a HTSTR.

Complete the model by closing the loop

As it is clarified in Chapter 2: Theoretical Background, the steam turbine is a component of the NPP secondary loop. In order to simulate the proper operation of the steam turbine the circuit should be completed.

The connection between the condenser and the HP-Turbine is composed of many heat exchangers, which collect the steam from the turbine extractions to heat the condensate, and the Steam-generator, in which the heated condensate is transformed into steam.

The development of such a complex model could be another way of further development of the model.

Bibliography

- [1] Nuclear Engineering HandBook, Section I, Chapter 2 PWR, R. Schreiber.
- [2] Macián, (2014), TU Munich, Course: Fundamentals of Nuclear Engineering, Lecture 1.
- [3] Macián, (2013), TU Munich, Course: Introduction to Nuclear Energy, Lecture 8.
- [4] Mechanical Engineers' Handbook: Energy and Power, Volume 4, Third Edition.
Edited by Myer Kutz. Copyright © 2006 by John Wiley & Sons, Inc.
- [5] Mecánica de Fluidos, Sexta edición, Frank M. White.
- [6] Macián, (2013), TU Munich, Course: Fundamentals of Thermal-hydraulics in Nuclear Systems, Lecture 4.
- [7] TRACE V5.0 Theory Manual: Field Equations, Solution Methods, and Physical Models. Division of Safety Analysis Office of Nuclear Regulatory Research.
- [8] TRACE V5.0 USER'S MANUAL, Volume 2: Modelling guidelines. Division of Safety Analysis Office of Nuclear Regulatory Research.
- [9] TRACE V5.0 USER'S MANUAL, Volume 1: Input Specification. Division of Safety Analysis Office of Nuclear Regulatory Research.

Adoración
Garrido-Lestache
Romero

**SIMULACIÓN DE UNA TURBINA DE VAPOR DE UNA CENTRAL NUCLEAR CON EL
CÓDIGO TRACE**

