Direction de l'Energie Nucléaire
Département d'Etudes des Réacteurs
Service de Simulation THermohydraulique
Laboratoire de Développement des Applications pour les Systèmes

BORDEREAU D'ENVOI

Date : 31 mars 2006 Page :

N/Réf : CEA/DEN/GRE/DER/SSTH/LDAS
DO 77 13/03/06

06PPQN000077

diffusé le : 13/03/06

Désignation :
SSTH/LDAS/EM/2005-034
CATHARE 2 V2.5_1 : USER GUIDELINES

Rédacteur : *G. LAVIALLE*

Observation :

B. FAYDIDE
Chef du Service de Simulation en THermohydraulique

Destinataires :

EDF/SEPTEN :	M. HENRIOT (2 ex) Mmes FERRIER*, MALATRAIT*, MM. HEMMERICH*, PENTORI*, GOY*
EDF/R&D :	Mme SOUYRI*, MM. ERHARD*, MARCHAND*
AREVA NP :	M. BARBIER (2 ex) MM. GANDRILLE*, SAUVAGE*
TECHNICATOME :	MM. MARIE, ROUX
IRSN/DESTQ (Cad) :	M. REOCREUX*
IRSN/DPAM (Cad) :	Mme CAMOUS (2 ex) MM. BARRE*, MARCHAND*, Mme GUILLARD*
IRSN/DSDRE/DPER (Far) :	M. de FRANCO*
IRSN/DSR (Far) :	M. DUBOIS (2ex) M. DUFEIL*, Mme MESSER*
DEN/DM2S/SFME (Sac)	M. CARUGE*
DEN/DM2S/SFME/LTMF (Sac) :	M. STUDER*
DEN/DM2S/SFME/LETR (Sac)	M. ROYER*, Mme ANIEL*
DEN/DM2S/SFME/LGLS (Sac)	M. DADA*
CS-SI :	Mme EYMARD*

DEN/DSNI :	M. PARADIS*
DEN/DSOE:	M. WILLERMOZ*
DEN/DER/DIR :	M. PORRACCHIA*
DEN/DER/SESI (Cad) :	Mme CHENAUD, M. PIGNATEL, MM. ROUAULT*, AUJOLLET*, DUMAZ*, MALO*, Mme TOSELLO*
DEN/DER/SPRC :	M. GROUILLER*
DEN/DER/SPEX :	Mme MERGUI*
DEN/DER/SRES :	M. CABRILLAT*
DEN/DER/SSTH :	MM. FAYDIDE*, TENCHINE*
DEN/DER/SSTH/LMDL :	MM. DUCROS*, BESTION*
DEN/DER/SSTH/LDAS :	Mmes FARVACQUE, M. LAVIALLE Mmes de CRECY, DOR, GEFFRAYE, MESSIE, RAMEAU, KADRI MM. ANTONI, BAZIN, PARENT, BENTIVOGLIO, PIALLA, LYVET, GERMAIN P., GERMAIN T., RUBY MOREAU, MIEUSSET, MOUTIN, MARLIET, PELLISSIER, DUMONT, VAN DEN HOVE
DEN/DER/SSTH/LIEX	M. HERVIEU*, Mme NOEL
DEN/DTN/SMTM/LMTR (Cad) :	M. ALBAREDE*
DEN/DTN/SE2T :	M. GARNIER*
DEN/DTN/SE2T/LTGD :	M. CHATAING*

**Résumé uniquement envoyé par courriel*
(le texte complet peut être envoyé sur demande par PDF)

Commissariat à l'Energie Atomique
DEN/DER/SSTH/LDAS
17, rue des Martyrs – 38054GRENOBLE Cédex 9
Tel : 04 38 78 50 43 - Secrétariat : 04 38 78 35 93 - Fax : 04 38 78 94 53 – E-mail : martine.farvacque@cea.fr

DIRECTION DE L'ENERGIE NUCLEAIRE

DEPARTEMENT D'ETUDES DES REACTEURS

SERVICE DE SIMULATION EN THERMOHYDRAULIQUE

LABORATOIRE DE DEVELOPPEMENT DES APPLICATIONS POUR LES SYSTEMES

C A T H A R E

CATHARE2 V2.5_1 : USER GUIDELINES

G. LAVIALLE

DER/SSTH/LDAS/EM/2005-034 *FEVRIER 2006*

CEA/DEN/GRE/DER/SSTH/LDAS
DO 68 21/02/06
06PPQN000068
diffusé le : 21/02/06

DIRECTION DE L'ÉNERGIE NUCLÉAIRE
DIRECTION DU CENTRE DE CADARACHE
DÉPARTEMENT D'ÉTUDES DES RÉACTEURS
SERVICE DE SIMULATION EN THERMOHYDRAULIQUE
LABORATOIRE DE DEVELOPPEMENT DES APPLICATIONS POUR LES SYSTEMES

Note Technique

TITRE: CATHARE 2 V2.5_1 : User guidelines

AUTEUR: G. LAVIALLE

RÉSUMÉ: Ce document est le guide utilisateur de CATHARE V2.5_1. Les conseils et réponses à diverses questions donnés dans ce guide sont le fruit de l'expérience des utilisateurs du code et de l'équipe CATHARE. Les principales caractéristiques de chaque module ou sous modules sont expliquées afin de garder à l'esprit les limites de chaque potentialité.

ABSTRACT This document is the CATHARE V2.5_1 user guide. The answers given in this user guide result from the acquired experience of both code users and CATHARE team. The main features and basic assumptions of each module and sub-module are explained to help the user keep in mind the limitations of each module.

MOTS CLÉS: CATHARE, GUIDELINES, THERMAL HYDRAULICS

Nom du Fichier Informatique: SSTH_LDAS_EM_2005-034.doc

	Rédacteur	Vérificateur		Approbateur
Fonction				Le Chef du LDAS
visa				
NOM	G. LAVIALLE	M. PARENT		M. FARVACQUE
Date	21/02/2006	21/02/2006		20/02/2006

Propriété Industrielle	Cadre de réalisation	Classification					Qualité
		DR	CC	CD	SD	Ss	
En l'absence d'accord ou de contrat, la diffusion des informations contenues dans ce document auprès d'un organisme tiers extérieur au CEA est soumise à l'accord du Chef de Département.	DOB.....DSNI Domaine.....Sûreté Projet.....THACC Contrat.....					X	AFAQ VERSION 2000

Indice	OBJET DES RÉVISIONS	DATE	NATURE et CHRONO UNITÉ
0	Émission initiale pour la version CATHARE 2 V2.5	29/06/05	SSTH/LDAS/EM/2004-067
0	Mise à jour pour la version CATHARE 2 V2.5_1 §2: MISE A JOUR QUALIFICATION FEBA ET SEFLEX §8 : WHAT ARE THE FLUIDS AVAILABLE IN THE CATHARE CODE? : NEW CHAPTER §17 : THERMAL RESISTANCE §19: JETPUMP MODEL FOR TEE SUB-MODULE §34 : HOW TO MODEL CORE NEUTRONICS? §36: HOW TO MODEL A GCR CORE : NEW CHAPTER. §38: HOW TO MODEL THE FUEL BALLOONING : NEW CHAPTER §39 HOW TO MODEL WALLS §46: VALVE HYSTERESIS §58: HOW TO MODEL HEAT LOSSES §64: HOW TO MODEL SPECIFIC COMPONENTS : NEW CHAPTER §65: RÉFÉRENCES §72: COMMON USER §73: HOW TO USE THE PERSONAL FILES : NEW CHAPTER §84: HOW TO CHOOSE AND TO USE BOUNDARY CONDITIONS §85: HOW TO CALCULATE A GAS SINGLE PHASE FLOW? : NEW CHAPTER §86: HOW TO CALCULATE USING NON-CONDENSABLE GASES §91 : HOW TO PROGRAM SIGNALS, TRIPS AND ACTIONS? §95: DIRECTIVES FUELDTMX ET DTMASS3D §96: DIRECTIVES FUELDTMX ET DTMASS3D §105: HOW TO SIMULATE FACILITY MEASUREMENTS *La Société CS-SI a participé à la mise à jour de ce document dans le cadre des contrats AB04B008 et AB04B006.*	21/02/06	SSTH/LDAS/EM/2005-034

INTRODUCTION

This document refers to the CATHARE 2 V2.5_1 version.

The CATHARE 2 user guide is divided into three sections with each section corresponding to a basic question that the user might ask himself.

- WHAT IS...?
- HOW TO...?
    - HOW TO MODEL...?
    - HOW TO USE...?
- WHY, WHEN...?

The answers given in this user guide result from the acquired experience of code users. The main features of each module and the basic assumptions are explained to help the user keep in mind the limitations of each module.

For each question, a reference to the relevant documents is made. The list of CATHARE documentation is given in the *HOW TO USE the CATHARE DOCUMENTATION?* section.

The user guide does not replace the USER MANUAL, nor the DICTIONARY of OPERATORS and DIRECTIVES. So, each time the answer to a question is given in one of these documents, no further explanation is given in the user guide.

In some cases, it was difficult to choose between WHAT, HOW, WHY, and WHEN. For some questions, reference is made to other relevant parts of the user guide, or the same information is repeated in several articles.

An INDEX OF WORDS in alphabetical order is provided to help you select the article(s) where you can find the information you are looking for.

CONTENTS

KEYWORD INDEX

22.	CHOKED FLOW	***HOW TO MODEL a BREAK?*** ***HOW TO CHOOSE a NODALISATION FOR AN AXIAL ELEMENT?***
23.	CIRCUIT	***WHAT is a CIRCUIT?***
24.	COMMON	***HOW TO USE the COMMON USER?***
25.	COMPONENT	***HOW TO MODEL SPECIFIC COMPONENTS***
26.	COMPRESSOR	***HOW TO MODEL a COMPRESSOR?*** ***HOW TO MODEL a TURBO-MACHINE?***
27.	CONVERGENCE	***WHAT are the AVAILABLE OPTIONS?*** ***HOW TO CONTROL TIME STEP?*** ***HOW TO EXPLAIN TIME STEP DECREASE?*** ***HOW TO REDUCE the COST of a CALCULATION*** ***HOW TO PERFORM SENSITIVITY STUDIES?***
28.	CONTAINMENT	***WHY-WHEN TO USE a VOLUME?*** ***HOW TO MODEL CONTAINMENT?*** ***HOW TO MODEL a SINK?*** ***HOW TO MODEL a SOURCE?***
29.	CORE	***HOW TO MODEL CORE THERMAL-HYDRAULICS?*** ***HOW TO MODEL CORE NEUTRONICS?***
30.	CRITICAL FLOW	***HOW TO MODEL a BREAK?*** ***HOW TO CHOOSE a NODALISATION FOR AN AXIAL ELEMENT?***
31.	DEAD END (BLIND)	***HOW TO CHOOSE and to use a BOUNDARY CONDITION?*** ***WHAT is a BOUNDARY CONDITION?*** ***HOW TO USE the BCMOD command?***
32.	DEAD ZONE	***HOW TO initialise a DEADZONE?***
33.	DEGRADATION (PUMP)	***WHAT is a ONE-NODE PUMP?*** ***HOW TO USE the ONE-NODE PUMP SUB-MODULE?***
34.	DIAPHRAGM	***HOW TO MODEL a DIAPHRAGM?***
35.	DOCUMENTATION	***HOW TO USE the CATHARE DOCUMENTATION?***
36.	DOWNCOMER	***HOW TO MODEL a DOWNCOMER?***
37.	DRYER	***HOW TO MODEL a SEPARATOR and a DRYER?***
38.	ECCS	***HOW TO MODEL a SOURCE?*** ***HOW TO MODEL an ACCUMULATOR?***
39.	ELEMENT	***WHAT is a MODULE?*** ***WHAT is a SUB-MODULE?***
40.	ENERGY BALANCE	***HOW TO CHECK the ENERGY and MASS BALANCE?***
41.	ERROR	***WHAT are the USUAL SYNTAX ERRORS?*** ***HOW to EXPLAIN a TIME STEP DECREASE?*** ***HOW TO READ ERROR MESSAGES?***
42.	EXCHANGER	***HOW TO MODEL a HEAT EXCHANGER?*** ***WHAT ARE THERMAL STRUCTURES?***
43.	FAILURE (VALVE)	***HOW TO MODEL a VALVE?*** ***HOW TO MODEL VALVE FAILURES?***
44.	FILES	***WHAT are the FILES USED BY CATHARE?*** ***HOW TO USE a PERSONNAL FILE?***
45.	FILL	***HOW TO MODEL a SOURCE?***
46.	FLOODING LIMIT	***HOW TO CONTROL CCFL and FLOODING LIMIT*** ***HOW TO USE THE CCFL SUB-MODULE?***
47.	FLOW LIMITER	***HOW TO MODEL a VALVE?***

		HOW TO MODEL a DIAPHRAGM? ***HOW TO MODEL an ABRUPT AREA CHANGE?***
48.	FORM LOSS COEFFICIENT	***WHAT is a SINGULAR PRESSURE DROP?*** ***HOW TO MODEL a SINGULAR PRESSURE DROP?***
49.	FUEL	***WHAT ARE THERMAL STRUCTURES?*** ***HOW TO MODEL FUEL RODS?*** ***HOW TO INITIALISE COLD/HOT FUEL DATA?*** ***WHY-WHEN TO USE THE STAND ALONE FUEL?*** ***HOW TO MODEL the FUEL BALLOONING***
50.	FUEL BALLOONING	***HOW TO MODEL the FUEL BALLOONING***
51.	GADJET	***WHAT is a SUB-MODULE?***
52.	GAS COOLED REACTOR	***HOW TO MODEL a TURBO-MACHINE?*** ***HOW TO MODEL a TURBINE?*** ***HOW TO MODEL a COMPRESSOR?*** ***HOW TO MODEL a GCR core?*** ***WHAT are the FLUID AVAILABLE in the CATHARE CODE?***
53.	GUIDE TUBE	***HOW TO MODEL a GUIDE TUBE?***
54.	GUILLOTINE RUPTURE	***WHAT is a BOUNDARY CONDITION?*** ***HOW TO CHOOSE and TO USE a BOUNDARY CONDITION?*** ***HOW TO MODEL a BREAK?***
55.	HEAD	***HOW TO MODEL a STEAM GENERATOR CHANNEL HEAD?*** ***WHAT is a ONE-NODE PUMP?*** ***WHAT is a SINGULAR PRESSURE DROP?*** ***HOW TO MODEL a SINGULAR PRESSURE DROP?***
56.	HEAT EXCHANGER	***WHAT is a ONE-NODE STEAM GENERATOR?*** ***HOW TO MODEL a STEAM GENERATOR?*** ***HOW TO MODEL a HEAT EXCHANGER?***
57.	HEAT LOSS	***HOW TO MODEL HEAT LOSSES?***
58.	HEAT SINK	***HOW TO MODEL a HEAT SINK?***
59.	HEAT SOURCE	***HOW TO MODEL a HEAT SOURCE?***
60.	HEATER	***HOW TO MODEL a PRESSURISER?***
61.	HPSI	***HOW TO MODEL a SOURCE?***
62.	HYDRAULIC	***HOW TO CHOOSE the HYDRAULICS of a 3D ELEMENT?*** ***HOW TO CHOOSE the HYDRAULICS of an AXIAL ELEMENT?*** ***HOW to model CORE THERMAL-HYDRAULICS?***
63.	HYDROGEN	***HOW TO CALCULATE USING NON-CONDENSABLE GASES?*** ***HOW TO CALCULATE a GAZ SINGLE PHASE FLOW?***
64.	IMPLICIT CODE	***WHAT is an IMPLICIT CODE?***
65.	INITIAL STATE	***HOW TO REACH the INITIAL STATE or STEADY STATE?***
66.	INPUT DECK	***WHAT are the CATHARE COMPUTATIONAL VARIABLES?*** ***HOW TO RESTORE CCV?*** ***HOW TO REACH the INITIAL STATE or STEADY STATE?*** ***HOW TO PROGRAM SIGNALS, TRIPS and ACTIONS?*** ***HOW TO PROGRAM within CATHARE (UTILX)?*** ***HOW TO USE the COMMON USER?*** ***HOW TO PERFORM a RESTART?*** ***HOW to SAVE the RESULTS?*** ***HOW TO USE a PERSONNAL FILE?***
67.	INTERMEDIATE LEG	***HOW TO MODEL an INTERMEDIATE LEG?*** ***HOW TO MODEL a BEND?***
68.	INTERPOLATION	***HOW TO PROGRAM SIGNALS, TRIPS and ACTIONS?***

69.	JUNCTION	***WHAT is a JUNCTION?***
70.	LBLOCA (Large Break Loss Of Coolant Accident)	***HOW TO MODEL CORE THERMAL-HYDRAULICS?*** ***HOW TO CONTROL TIME STEP?*** ***WHAT is the REFLOODING SUB-MODULE?*** ***HOW TO USE the BOTTOM-UP REFLOODING SUB-MODULE?*** ***HOW TO USE the TOP-DOWN REFLOODING SUB-MODULE?*** ***WHY-WHEN TO USE a REFLOODING SUB-MODULE?*** ***HOW TO INITIALISE COLD/HOT FUEL DATA?*** ***WHY-WHEN TO USE THE STAND ALONE FUEL?***
71.	LISTING	***HOW TO READ a LISTING?***
72.	LOWER PLENUM	***HOW TO MODEL a LOWER PLENUM?*** ***WHY-WHEN TO USE a VOLUME?***
73.	LPSI	***HOW TO MODEL a SOURCE?***
74.	MANWAY	***HOW TO MODEL a MANWAY?*** ***HOW TO USE the BCMOD COMMAND?***
75.	MASK	***HOW TO USE MASKS?***
76.	MASS BALANCE	***HOW TO CHECK the ENERGY and MASS BALANCE?*** ***WHAT is a ZONE?***
77.	MEASUREMENT	***HOW TO SIMULATE FACILITY MEASUREMENTS***
78.	NODALISATION	***WHAT is an AXIAL ELEMENT?*** ***HOW TO CHOOSE a NODALISATION FOR AN AXIAL ELEMENT?*** ***WHAT is a ONE-NODE PUMP?*** ***HOW TO USE the ONE-NODE PUMP SUB-MODULE?*** ***HOW TO CHOOSE a NODALISATION FOR A 3D ELEMENT?*** ***HOW TO MODEL a DIAPHRAGM?*** ***HOW TO MODEL a BREAK?***
79.	MODULE	***WHAT is a MODULE?*** ***WHAT is a SUB-MODULE?***
80.	NEUTRONICS	***HOW TO MODEL CORE NEUTRONICS?***
81.	NITROGEN	***HOW TO CALCULATE USING NON-CONDENSABLE GASES?*** ***HOW TO CALCULATE a GAZ SINGLE PHASE FLOW?***
82.	NODALISATION	***WHAT is an AXIAL ELEMENT?*** ***HOW TO CHOOSE a NODALISATION FOR AN AXIAL ELEMENT?*** ***WHAT is a ONE-NODE PUMP?*** ***HOW TO USE the ONE-NODE PUMP SUB-MODULE?*** ***HOW TO DEFINE the GEOMETRY of an AXIAL ELEMENT?***
83.	NONCONDENSABLE GAS	***HOW TO CALCULATE USING NON-CONDENSABLE GASES?*** ***HOW TO CHOOSE and TO USE a BOUNDARY CONDITION?*** ***HOW TO CALCULATE a GAZ SINGLE PHASE FLOW?***
84.	OPERATOR ACTION	***HOW TO PROGRAM SIGNALS, TRIPS and ACTIONS?*** ***HOW TO PROGRAM within CATHARE (UTILX)?*** ***HOW TO PROGRAM REGULATIONS?***
85.	OPTION	***WHAT are the AVAILABLE OPTIONS?*** ***HOW TO CONTROL TIME STEP?*** ***HOW TO EXPLAIN TIME STEP DECREASE?*** ***HOW TO REDUCE the COST of a CALCULATION***
86.	PARALLEL COMPUTATION	***WHAT are the AVAILABLE OPTIONS?***
87.	PIPE	***See AXIAL ELEMENT***
88.	PLENUM	***HOW TO MODEL a LOWER PLENUM?*** ***HOW TO MODEL an UPPER PLENUM?*** ***HOW TO MODEL a STEAM GENERATOR CHANNEL HEAD?*** ***WHY-WHEN TO USE a VOLUME?***

89.	PRESSURE DROP	***WHAT is a SINGULAR PRESSURE DROP?*** ***HOW TO MODEL a SINGULAR PRESSURE DROP?*** ***HOW TO MODEL a DIAPHRAGM?***
90.	PRESSURISER	***HOW TO MODEL a PRESSURISER?*** ***WHY-WHEN TO USE a VOLUME?***
91.	PRINTING	***HOW TO CONTROL PRINTING?*** ***HOW TO CHECK the ENERGY and MASS BALANCE?*** ***HOW TO USE a PERSONAL FILE ?***
92.	PUMP	***WHAT is a ONE-NODE PUMP?*** ***HOW TO USE the ONE-NODE PUMP SUB-MODULE?***
93.	PWR (pressurised water reactor)	***HOW TO MODEL a SEPARATOR and a DRYER?*** ***HOW TO DEFINE the GEOMETRY of a VOLUME?*** ***WHY-WHEN TO USE a VOLUME?***
94.	QUENCH	***WHAT is the REFLOODING SUB-MODULE?*** ***HOW TO USE the BOTTOM-UP REFLOODING SUB-MODULE?*** ***HOW TO USE the TOP-DOWN REFLOODING SUB-MODULE?*** ***WHY-WHEN TO USE a REFLOODING SUB-MODULE?***
95.	REACTOR	***WHAT is a REACTOR?***
96.	REACTIVITY	***HOW TO MODEL CORE NEUTRONICS?***
97.	REFLOODING	***WHAT is the REFLOODING SUB-MODULE?*** ***HOW TO USE the BOTTOM-UP REFLOODING SUB-MODULE?*** ***HOW TO USE the TOP-DOWN REFLOODING SUB-MODULE?*** ***WHY-WHEN TO USE a REFLOODING SUB-MODULE?*** ***HOW TO MODEL CORE THERMAL-HYDRAULICS?*** ***HOW TO CHOOSE a NODALISATION FOR AN AXIAL ELEMENT?***
98.	REGULATIONS	***HOW TO PROGRAM REGULATIONS?*** ***WHAT are the AVAILABLE OPTIONS?***
99.	RESTART	***HOW TO PERFORM a RESTART?***
100.	REWETTING	***See REFLOODING***
101.	ROD	***WHAT ARE THERMAL STRUCTURES?*** ***HOW TO MODEL FUEL RODS?*** ***HOW TO INITIALISE COLD/HOT FUEL DATA?*** ***WHY-WHEN TO USE THE STAND ALONE FUEL?***
102.	ROD BUNDLE	***HOW TO MODEL CORE THERMAL-HYDRAULICS?*** ***HOW TO CHOOSE the HYDRAULICS of an AXIAL ELEMENT?*** ***HOW TO CHOOSE the HYDRAULICS of a 3D ELEMENT?***
103.	SAFETY VALVE	***HOW TO MODEL a SAFETY VALVE?*** ***HOW TO MODEL a SINK?*** ***HOW TO MODEL a VALVE?*** ***HOW TO CHOOSE and TO USE a BOUNDARY CONDITION?***
104.	SAVE	***HOW to SAVE the RESULTS?***
105.	SENSITIVITY STUDY	***HOW TO PERFORM SENSITIVITY STUDIES?***
106.	SEPARATOR	***HOW to model a SEPARATOR and a DRYER?***
107.	SG CHANNEL HEAD	***HOW TO MODEL a STEAM GENERATOR CHANNEL HEAD?*** ***WHY-WHEN TO USE a VOLUME?***
108.	SGTR (steam generator tube rupture)	***HOW TO MODEL a SGTR?***
109.	SIGNALS	***HOW TO PROGRAM SIGNALS, TRIPS and ACTIONS?*** ***HOW TO PROGRAM within CATHARE (UTILX)?***
110.	SINGULARITY	***HOW TO MODEL a SINGULAR PRESSURE DROP?*** ***HOW TO MODEL a DIAPHRAGM?***

		HOW TO MODEL an ABRUPT AREA CHANGE? ***WHAT is a SINGULAR PRESSURE DROP?*** ***HOW TO MODEL a BEND?*** ***HOW TO USE THE CCFL SUB-MODULE?*** ***HOW TO CONTROL CCFL and FLOODING LIMIT***
111.	SINK	***HOW TO MODEL a SINK?***
112.	SOURCE	***HOW TO MODEL a SOURCE?***
113.	SPRAY	***HOW TO MODEL a PRESSURISER?***
114.	STAGGERED MESH	***WHAT is an AXIAL ELEMENT?***
115.	STATIONARY STATE	***HOW TO REACH the INITIAL or STEADY STATE?***
116.	STEADY STATE	***HOW TO REACH the INITIAL or STEADY STATE?***
117.	STEAM GENERATOR	***WHAT is a ONE-NODE STEAM GENERATOR?*** ***HOW TO MODEL a HEAT EXCHANGER?*** ***HOW TO MODEL a STEAM GENERATOR?*** ***HOW TO MODEL a STEAM GENERATOR CHANNEL HEAD?***
118.	SUB-MODULE	***WHAT is a SUB-MODULE?***
119.	TEE	***WHAT is a TEE?*** ***HOW TO DEFINE THE GEOMETRY OF A TEE?*** ***WHY/WHEN TO USE a TEE?*** ***HOW TO MODEL a BREAK?***
120.	THERMAL STRUCTURES	***WHAT ARE THERMAL STRUCTURES?*** ***HOW TO MODEL FUEL RODS?*** ***HOW TO INITIALISE COLD/HOT FUEL DATA?*** ***WHY-WHEN TO USE THE STAND ALONE FUEL?*** ***HOW TO MODEL a HEAT EXCHANGER?*** ***HOW TO MODEL HEAT LOSSES?***
121.	THREED	***HOW TO MODEL the 3D VESSEL?*** ***HOW TO MODEL a GUIDE TUBE?*** ***WHAT is a 3D ELEMENT?*** ***HOW TO DEFINE the GEOMETRY of a 3D ELEMENT?*** ***HOW TO DEFINE the CONNECTIONS of a 3D ELEMENT?*** ***HOW TO CHOOSE a NODALISATION FOR A 3D ELEMENT?*** ***HOW TO CHOOSE the HYDRAULICS of a 3D ELEMENT?*** ***HOW TO CHOOSE the PHYSICAL LENGTH SCALES of a 3D ELEMENT?*** ***WHY-WHEN TO USE a 3D MODULE?***
122.	TIME STEP	***HOW to control TIME STEP?*** ***HOW to explain a TIME STEP DECREASE?*** ***WHAT are the AVAILABLE OPTIONS?*** ***HOW TO REDUCE the COST of a CALCULATION***
123.	TRANSIENT	***HOW TO PROGRAM SIGNALS, TRIPS and ACTIONS?*** ***HOW to program within CATHARE (UTILX)?***
124.	TRIPS	***HOW TO PROGRAM SIGNALS, TRIPS and ACTIONS?*** ***HOW TO PROGRAM within CATHARE (UTILX)?***
125.	TUBE	***WHAT is an AXIAL ELEMENT?*** ***HOW TO CHOOSE the HYDRAULICS of an AXIAL ELEMENT?*** ***HOW TO DEFINE the GEOMETRY of an AXIAL ELEMENT?***
126.	TURBINE	***HOW TO MODEL a TURBINE?*** ***HOW TO MODEL a TURBO-MACHINE?***
127.	TURBO-MACHINE	***HOW TO MODEL a TURBO-MACHINE?***
128.	UPPER PLENUM	***HOW TO MODEL an UPPER PLENUM?*** ***WHY-WHEN TO USE a VOLUME?***

129. VALIDATION DOMAIN	***WHAT is the CATHARE VALIDATION DOMAIN?***
130. VALVE	***HOW TO MODEL a VALVE?*** ***HOW TO MODEL a SINK?*** ***HOW TO MODEL an ABRUPT AREA CHANGE?*** ***HOW TO initialise a DEADZONE?***
131. VESSEL	***HOW TO MODEL the 3D VESSEL?*** ***HOW TO MODEL the VESSEL?***
132. VOLUME	***WHAT is a VOLUME?*** ***HOW TO MODEL a LOWER PLENUM?*** ***HOW TO MODEL an UPPER PLENUM?*** ***HOW TO MODEL a PRESSURISER?*** ***HOW TO MODEL a STEAM GENERATOR CHANNEL HEAD?*** ***HOW TO DEFINE the GEOMETRY of a VOLUME?*** ***WHY-WHEN TO USE a VOLUME?***
133. VVER	***HOW TO MODEL a STEAM GENERATOR CHANNEL HEAD?***
134. WALL	***WHAT ARE THERMAL STRUCTURES?*** ***HOW TO MODEL FUEL RODS?***
135. ZONE	***WHAT is a ZONE?***

1. WHAT IS THE USER GUIDE?

REFERENCE:	
KEY WORD	
See also	

This USER GUIDE has several aims:

- To help you find your way in the forest of the CATHARE documentation.
- To give you some basic ideas of what is what.
- To give answers to some basic questions for people building an input deck.
- To help you make the right choice between several possibilities.

It gives the CATHARE modelling recommendations and aims to reduce the code user effect.

2. WHAT IS THE CATHARE VALIDATION DOMAIN?

REFERENCE:	
KEY WORDS	**VALIDATION DOMAIN,**
See also	

The CATHARE validation domain is summarised in the following tables. For each separate-effect test of the qualification matrix, the bounds of the main physical parameters are given. The integral-effect test matrix is also presented:

2.1 Qualification domain

Main Phenomena (CATHARE module)	EXPERIMENT	Mechanical Transfer	Interfacial Heat Transfer	Wall Heat Flux	Component	Validation documentation
Critical flowrate 0-D, 1-D modules	SMD long nozzle	•	•		Break	98-035 2000-021
	SMD short nozzle	•	•		Break	98-035 2000-021
	BETHSY short nozzle	•	•		Break	98-035 2000-021
	BETHSY long nozzle	•	•		Break	98-035 2000-021
	MARVIKEN 17. 24	•	•		Break	98-035 2000-021
	REBECA diaphragm	•	•		Break	2000-009 2000-021
	REBECA orifice	•	•		Break	2000-009 2000-021
Interfacial friction flow regimes 1-D module	CANON vertical tube	•			Break	99-047
	CANON vertical rod bundle	•			Core	99-053
	TAPIOCA	•			Break	99-047
	PERICLES boil up	•		•	Core	97-016
	PERICLES boil off	•		•	Core	97-016
	BETHSY core	•		•	Core	97-016
	PATRICIA SG2	•		•	SG	98-037
	ECTHOR intermediate leg	•			IL	99-085
	SMD horizontal (90, 135)	•			HL	2001-015
	MHYRESA droplet	•		•	HL	2000-023
	MHYRESA entrainment	•			HL	2001-019
Wall friction 1-D module	MD air-water	•			PIPE	99-034
	CISE	•			PIPE	99-034
	COLLIER-HEWITT	•			PIPE	99-034
CCFL 0-D, 1-D modules	MHYRESA CCF B118	•			HL	2001-011
	MHYRESA CCF R350	•			HL	2001-011
	ECTHOR hot leg	•			HL	2001-011
	MHYRESA UCP	•			Core	99-055
	MHYRESA SG tube	•			SG tubes	99-069
	HANNOVER	•			Core	99-055
	UPTF hot leg 11-26c	•			HL	99-018
Reflooding	PERICLES reflooding	•	•	•	Core	98-033

Main Phenomena (CATHARE module)	EXPERIMENT	Mechanical Transfer	Interfacial Heat Transfer	Wall Heat Flux	Component	Validation documentation
1-D module	ERSEC rod bundle	•	•	•	Core	2000-043
	ROSCO constant and oscillatory flowrates	•	•	•	Core	98-033
	REWET II	•	•	•	Core	2004-005
Wall flux *1-D module*	OMEGA rod bundle	•	•	•	Core	98-010
	COSI	•	•	•		98-002
	PATRICIA SG1	•	•	•	SG tubes	2000-019
	COTURNE	•	•	•	SG tubes	2002-085
	COTURNE 3 tubes	•	•	•	SG tubes	2001-017
	MIT	•	•	•		98-002
	TOSHIBA	•	•	•		98-002
	KIT	•	•	•		2000-030
	FLECHT SG	•	•	•	SG tubes	2000-07
Rewetting-Film boiling 1-D module	WINFRITH (hot patch)	•	•	•	Core (tube)	99-014
	INEL rewetting	•	•	•	Core (tube)	2000-018
	TPTF	•	•	•	BWR Core (rod bundle)	2002-036
Condensation *1-D module*	COSI (Accu, NC)		•		ECC	98-047 2002-65
	COSI tee		•		ECC	98-046
Fuel model *1-D module*	REBEKA			•	Fuel	99-071
	EDGAR			•	Fuel	98-053 99-021
Downcomer refilling *1-D module*	UPTF (5a, 5b, Z3E)	•	•		Downcomer	98-45
Phase separation *Tee and sink*	SMD tee	•			Break	99-003
	INEL tee	•			Break	99-003
	MHYRESA tee	•			Break	2003-059
Lower plenum voiding *0-D module*	PIERO	•			Lower Plenum	2000-011
Upper plenum de-entrainment *0-D module*	UPTF 10C	•			Upper Plenum	2004-014
Pump *0-D module*	EVA 0-D	•	•	•	Pump	2004-016
Pressuriser *0-D module*	DAMPIERRE	•	•	•	Pressuriser	2000-16
SG secondary side *1-D module*	PALUEL	•	•	•	SG	2000-51
Downcomer refilling *3-D module*	UPTF (6, 7)	•	•		Downcomer	99-050
Core uncovery *3-D module*	PERICLES 2-D boil up	•	•	•	Core	98-059
	PERICLES 2-D reflooding	•	•	•	Core	98-061
Lower plenum *3-D module*	PIERO	•			Lower Plenum	98-060
Containment (0-D module)	DEHBI (MIT)		•	•	Containment	2003-050
	COPAIN		•	•	Containment	2003-050
Fuel ballooning (3-D module)	CEGB air	•			Core	2004-058
	CEGB steam		•	•	Core	2004-058
	CEGB steam-droplets	•	•	•	Core	2004-058
	FLECHT (reflooding)	•	•	•	Core	2004-058
	FEBA (reflooding)	•	•	•	Core	2004-058
	SEFLEX (reflooding)	•	•	•	Core	2004-058
	ACHILLES (reflooding)	•	•	•	Core	2004-058

2.1.1 Critical flowrate: 0-D, 1-D modules

BETHSY short nozzle	3< P <10 MPa	D=5.24*10^{-3} m	L/D=14.313	subcooled and saturated fluid
BETHSY long nozzle	3< P <10 MPa	D=15.43*10^{-3} m	L/D=9.536	subcooled and saturated fluid
SUPER MOBY DICK long nozzle	2< P <12 MPa	D=20*10^{-3} m	L/D=18.48	subcooled and saturated fluid
SUPER MOBY DICK short nozzle	2< P <4 MPa	D=15.73*10^{-3} m	L/D=0	subcooled fluid
REBECA orifice STEAM-WATER and AIR STEAM WATER	3< P <8 MPa		0<Xvap<1	0<Xair<0.1
REBECA diaphragm STEAM-WATER and AIR STEAM WATER	2< P <8 MPa	30*10^{-3} <D<70 *10^{-3} m 0<L/D<20	0.17<Xvap<1	0<Xair<0.1
MARVIKEN	MARVIKEN tests N 17 Blowdown transient input deck K = 0 ÷ 0.25 D = 0.3 m L/D = 3.72 P = 5.1 ÷ 2.4 MPa Inlet temperatures: subcooled and saturated fluid		MARVIKEN tests N 24 Blowdown transient input deck K = 0 D = 0.500 m L/D ≈ 0.39 P = 4.6 ÷ 2.8 MPa Inlet temperatures: subcooled and saturated fluid	

2.1.2 Interfacial friction flow regimes: 1-D module

BETHSY core	4< P <16 MPa	W =2*10^{-4} W/m2	NITROGEN	
PERICLES boil off	0.345< P<6 MPa	1*10^{-4} < W <3 *10^{-4} W/m2	2< DTsat <30°C	2.35< Z <3.66 m
PERICLES boil up	1< P <6 MPa	1.42*10^{-4}< W<2*10^{-4} W/m2		
CANON VERTICAL rod bundle	P=15 MPa	3*10^{-3} < Dbreak < 5*10^{-3} m	230<TL<300°C	
CANON VERTICAL tube	5.7<P<14.9 MPa	3*10^{-3}< Dbreak< 15*10^{-3} m	232<TL<320°C	
TAPIOCA	14.5<P<15 MPa	10*10^{-3} <Dbreak< 20*10^{-3} m	TL=280°C	
ECTHOR IL	P=0.1 MPa	0.3<Level/D<0.58 m	0.14<Qair<0.56 kg/s	
MHYRESA HL	P=0.1 MPa	D=0.118 m	-0.01<Z<0.03 m	194<QG<11800 m3/h 0<QL<4.5 m3/h
		D=0.351 m	-0.02<Z<0.10 m	3200<QG<1414 m3/h 0<QL<32 m3/h
MHYRESA entrainment	P=0.1 MPa	D=0.351 m	5<QL<20 m3/h	767<QG<5000 m3/h
		D=0.118 m	0.3 <QL<2.5 m3/h	25<QG<300 m3/h
SUPER MOBY DICK horizontal	2<P<10 MPa	D=0.135 m	400<G<2400 kg/m2/s	0.22<ALFA<0.91
	2<P<10 MPa	D=0.090 m	3.7<G<16.5 kg/m2/s	0.17<ALFA<0.93
PATRICIA SG2	1<P<7 MPa	0<W<800000 W	20<G<400 kg/m2/s	0<ALFA<1 0<X<2

2.1.3 Wall friction: 1-D module

CISE	2<P<9 MPa	0.918<D<1.52 c	$50*10^{-3}$<G<$390*10^{-3}$ kg/s/m2	0.03<X<0.98
COLLIER HEWITT	P= 0.1 MPa	D=$3.175*10^{-2}$ m	0.03<QG<0.08 kg/s 3.2410^{-4}<QL<0.1613 kg/s	62<TG<74°C 55<TL<69°C
MOBY DICK	0.113<P<0.27 MPa	D=$1.4*10^{-2}$ m	7950<G<12230 kg/s/m2	5.56<X<50.1 33<TL<40°C

2.1.4 CCFL: 0-D, 1-D modules

MHYRESA CCF B118 R351	P=0.1 MPa	0.118<D<0.351 m	190<QG<7100 m3/h	3<QL<15 m3/h
ECTHOR HL	P=0.1 MPa	D=0.25 m	265<QG<3000 m3/h	QL=144 m3/h
HANNOVER	P=0.48 MPa	0.230 m × 0.230 m with 16 × 16 rod bundle and an upper tie plate. The plate has 229 holes of diameter 0.0104 m, so the open flow cross-section is about 37%.		
MHYRESA UCP	P=0.1 MPa	Rectangular cross-section channel 0.214 m × 0.214 m with a 17 × 17 rod bundle, a core top nozzle and an upper tie plate: different diameters of the upper tie plate ϕ = 0.140 m, corresponding to an average cross-section (0.0154 m^2) of the plate in case of 157 fuel assemblies in a FRAMATOME reactor ϕ = 0.175 m, corresponding to the cross-section (0.024 m^2) of the plate in the BETHSY geometry		
MHYRESA SG tube	P=0.1 MPa	D=0.0201 m		
UPTF HL	UPTF11	0.3<P<1.5 MPa		
	UPTF26c	P=0.4 MPa		

2.1.5 Reflooding: 1-D module

ERSEC rod bundle	0.1< P <0.6 MPa	$3*10^{-4}$< W <$4.5*10^{-4}$ W/m2	0.002<G<0.006 kg/m2/s	0.25<(T_{SAT}-TL)/<0.2°C
REWET II	P=0.1 or 0.3 MPa	W =30000 W	Downcomer injection	Upper plenum injection
ROSCO constant flowrate	P= 0.3 MPa	W=$3*10^{-4}$ W/m2	10<G<44 kg/m2/s	Inlet subcooling =16.6°C
ROSCO oscillatory flowrate			26<G<44 kg/m2/s	
PERICLES reflooding	0.2< P <0.4 MPa	$1.5*10^{-4}$< W <$4.2*10^{-4}$ W/m2	10<G<190 kg/m2/s	Inlet subcooling =60°C

2.1.6 Wall flux: 1-D module

COTURNE-3 TUBES	NITROGEN, HELIUM	0.5< P <2 MPa	0.01PN<W<0.02PN	
COTURNE	PURE STEAM, NITROGEN, HELIUM CO or COUNTER CURRENT	0.2<P<7 MPa	$0.1*10^{-4}$<W<$13.6*10^{-4}$ W/m2	
MIT	AIR,HELIUM	0.1< P <0.5 MPa	0.< X <0.34	$3*10^{-3}$<QG<$13*10^{-3}$ kg/s
TOSHIBA	NITROGEN	P=3 MPa	0.< X <0.147	QG=$30*10^{-3}$ kg/s
COSI	NITROGEN	2 < P <2.4 MPa	0.< X <0.32	$170*10^{-3}$<QG<$238*10^{-3}$ kg/s

FLECHT SG		P=5.86 MPa	TG=147.7°C QG=0.179 kg/s	QL=0.045 kg/s
PATRICIA SG1	NITROGEN	4< P <7 MPa	$5*10^{-4}$<W<$17*10^{-4}$ W/m2	0<Q nitrogen<0.25 cm3/s 20<Q inlet<850 kg/s/m2
KIT		4.3 < P <11 MPa	$43*10^{-4}$<W<$180*10^{-4}$ W/m2	130<TL<300°C 400<G<2130 kg/m2/s
OMEGA rod bundle		13< P <15 MPa	0<W<1000 W/m2	10<Q<19 kg/s Tinlet=285°C
	hot break or double break: break size: 5% or 14% of the test section hydraulic area: break section ratio for the double break: S_h/S_c= ¼, 4, ∞.			

2.1.7 Rewetting-Film boiling: 1-D module

INEL tube rewetting	0.3< P <0.8 MPa	$0.48*10^{-4}$<W<$15*10^{-4}$ W/m2	12.3<G<61 kg/m2/s	0< quality <0.46
WINFRITH tube (hot patch)	0.5< P <7 MPa	$163*10^{-4}$<W<$440*10^{-4}$ W/m2	100<G<1000 kg/m2/s	24< subcooling <47°C
TPTF rod bundle	3< P <12 MPa	$1.5*10^{-4}$<W<$26*10^{-4}$ W/m2	20<G<413 kg/m2/s	0< quality <0.9

2.1.8 Condensation: 1-D module

COSI: ACCU, NC	CL- D=0.118 m	P= 4.1MPa	10% saturated nitrogen	0.<Q<5 kg/s	T=20°C
COSI tee	CL- D=0.118 m	P= 2 or 7 MPa	0<H/D<0.6	0.1<Q<0.6 kg/s	22<T<83°C

2.1.9 Fuel model

EDGAR	FRAMATOME: stress-relieved Zr CEA: recrystallised Zr PHEBUS: stress-relieved Zr	**creep tests:** -temperature ramp tests (constant pressure from 1 to 12.5 MPa, constant heating rate from 0.1 to 100°C/s) -pressure ramp tests (constant temperature, only T=800°C, constant pressure rate from 0.01 to 0.2 MPa/s) -tests for the effect of heat treatment due to the first temperature peak
REBEKA 6	48 simulator fuel rods with gap (Pgap=6 MPa)	Fuel rupture analysis with reflooding phase at W=6600 W/rod

2.1.10 Downcomer refilling: 1-D module

UPTF 5a	Simulates the end of depressurisation with flashing occurring in the lower plenum, accumulator discharge (Q= 500 kg/s, T = 30°C) and water penetration into the lower plenum. As there is no steam injection simulation (in the core) but only flashing occurring in the lower plenum, the resistance to the water down flow is non-existent and all the ECC water injected flows to the lower plenum. Pressure decreases from 0.18 MPa to 0.25 MPa during the test.
UPTF 5b	A constant pressure of nearly 0.5 MPa is imposed as well as the steam flowrate injected into the core (15<Q<19 kg/s). Sub-cooled ECC water (Q=500 kg/s, T= 30°C) is injected in the 3 intact cold legs together with nitrogen.
UPTF Z3E	Analytical test with injection of nearly saturated water (Q=750 kg/s, T=163°C) in one of the intact cold legs (the closest one to the break), no nitrogen injection, a constant steam flowrate injected into the core (2<Q<4 kg/s) and a pressure of nearly 0.8 MPa imposed in the circuit.

2.1.11 Phase separation:Tee and sink

MHYRESA tee	P= 0.1 MPa	D=0.351 m	DTEE=0.13 m	UPWARD TEE PRZ open SG closed	0.06<LEVEL<0.28 m	0<QL<20 m3/h
				UPWARD TEE PRZ open SG open	0.08<LEVEL<0.15 m	0<QL<25 m3/h
SUPER MOBY DICK tee	P= 0.2 and 0.7 MPa	D=0.135 and 0.080 m	DTEE=0.020 and 0.012 m	Vertical upward and downward and horizontal	0<Xl<0.144	3<QT<13 kg/s
INEL tee	P= 3.45 and 6.21 MPa	D=0.284 m	DTEE=0.034 m	Vertical downward and horizontal	0.14<Xl<0.62	2.5<QT<9.3 kg/s

2.1.12 Lower plenum voiding: 0-D module

PIERO (Air injection)	P= 0.1 MPa	100<Qair<3442 m3/h	0<QL<0.2 l/s	0<Xliquid<0.2 0.96<Xair<0.99

2.1.13 Upper plenum de-entrainment: 0-D module

UPTF 10c	The UPTF separate effect test 10C, run 082, was performed to investigate the counter-current flow limitation at the full-scale upper core tie plate. The test focused on the behaviour of water injected through the core simulator nozzle which can be either de-entrained below or above the tie plate or carried over to the hot leg and the SG simulator with the steam flow. De-entrained water in the upper plenum will fall back through the tie plate into the core, counter to the two-phase up-flow.	Pinit=0.35 and 0.38 MPa	49<QLSI<148 kg/s 80<QVcore<82 kg/s 21.7<Qdownco<76.6 kg/s

2.1.14 Pump: 0-D module

EVA	0.815< P <2.9 MPa	0<QV<500 % of NV	0<ALFA<1	Direct or reverse flow

2.1.15 Pressuriser: 0-D module

DAMPIERRE	Normal spraying	8< P <15.5 MPa
	Auxiliary spraying	14< P <15.5 MPa
	Piston effect	12< P <15.5 MPa
	PRZ heaters	154< P <160 MPa

2.1.16 SG secondary side: 1-D module

PALUEL	7.19< P <8.13 MPa	58000 <W<957000 W	540.2<QFW<30.8 kg/s	228<TFW<114°C
	A transient test corresponding to a full power emergency shutdown, performed on PALUEL 1 on 25th February 1985, has been selected			

2.1.17 Downcomer refilling: 3-D module

UPTF 6 and 7	Large break LOCA (200%) separate-effect tests to understand -counter-current flow phenomenon -the scale effect -the condensation effect	Series No.7, nearly-saturated ECC-water (Q= 500 or 740 kg/s for 1, 2 or 3 loops), steam injection in the core (30<Qs<100 kg/s)	0<Dtsat<22°C
		Series No.6, subcooled water, ECC-water (Q= 500 kg/s per loops), steam injection in the core (100<Qs<450 kg/s)	26<Dtsat<59°C

2.1.18 Core uncovering: 3-D module

PERICLES 2D reflooding	0.3< P <0.4 MPa	$*10^{-4}$< W <4.2*10^{-4} W/m2 for HA 2.27*10^{-4}< W <3*10^{-4} W/m2 for CA	36<G<50 kg/m2/s for HA 31.5<G<50 kg/m2/s for HA	60<Inlet subcooling<90°C
PERICLES 2D boil up	P=0.3 MPa	2.45*10^{-4}< W <3.5*10^{-4} W/m2	1<Fxyz<1.85	2.74<Zg<3.45 m

2.1.19 Lower plenum: 3-D module

PIERO	P= 0.1 MPa	100<Qair<3442 m3/h	0<QL<0.2 l/s	0<Xliquid<0.2 0.96<Xair<0.99

2.1.20 Containment: 0-D module

DEHBI-MIT	AIR	0.15< P<0.45 MPa	90<Tair<150°C	0.3<Xair<0.9
COPAIN	Natural conv.	0.1< P<0.7 MPa	70<Tair<160°C	0.1<Xair<0.9
	Forced conv.	0.1< P<0.7 MPa	70<Tair<170°C	0.1<Xair<0.9

2.1.21 Fuel ballooning: 3-D module

CEGB air	P = 0.1 MPa	Long balloon	Blockage rate = 90% and 61%	0.016<Qair<0.045 kg/s
		Short balloon	Blockage rate = 90%	0.017<Qair<0.045 kg/s
CEGB steam	P = 0.1 MPa	Long balloon	Blockage rate = 90% and 61%	0.020<QG<0.050 kg/s

		Short balloon	Blockage rate = 90%	0.020<QG<0.050 kg/s
CEGB steam-droplets	P = 0.1 MPa	Long balloon	Blockage rate = 90% and 61%	0.055<QG<0.063 kg/s 0.010<QL<0.048 kg/s
		Short balloon	Blockage rate = 90%	0.045<QG<0.050 kg/s 0.023<QL<0.075 kg/s
FLECHT (reflooding)	0.14<P<0.28 MPa	2<Blockage rate<40% non-coplanar	50<Inlet subcooling<80	0.017<Reflooding rate<0.03 m/s
ACHILLES (reflooding)	0.15<P<0.4 MPa	2<Blockage rate<40% coplanar	20<Inlet subcooling<50°C	Reflooding rate = 0.02 m/s
FEBA (reflooding)	0.2<P<0.61 MPa	62<Blockage rate<90%	0<Inlet subcooling<50°C	0.022<Reflooding rate<0.095 m/s
SEFLEX (reflooding)	0.21<P<0.41 MPa	62<Blockage rate<90%	Inlet subcooling = 60°C	0.038<Reflooding rate<0.058 m/s

2.2 Verification domain

2.2.1 System test facilities for CATHARE verification

LOOP	VERT. SCALE	VOLUME SCALE	POWER	PRESSURE MPa	LOOB NB	CORE
LOFT	½	1/48	100%	16	2	Nuclear
LSTF	1/1	1/48	14%	16	2	Electrical
BETHSY	1/1	1/100	10%	16	3	Electrical
LOBI	1/1	1/700	100%	16	3	Electrical
PACTEL	1/1	1/305		8	3	Electrical

2.2.2 System test facilities for CATHARE containment verification

LOOP	VOLUME	HEIGHT	Nb containment zones	Containment material
CVTR	V=6500 m^3	Z=36 m	3 ZONES	CONCRETE
HDR	V=11000 m^3	Z=50 m	64 ZONES	CONCRETE

2.2.3 Selected tests for the assessment of CATHARE

LB LOCA	
LOFT L2-5	LBLOCA (0-D/1-D and 3-D modelling of the vessel)
LOFT LP02-6	LBLOCA (0-D/1-D and 3-D modelling of the vessel)
BETHSY 6.7c	Reflooding, base case

INTERMEDIATE BREAK	
BETHSY 4.2b	1.6” Break, Lower Plenum
BETHSY 9.1b	2” Break, Cold Leg, without HPSI
BETHSY 8.1	3” Break, Cold Leg, delayed MCP trip
BETHSY 7.3b	3” Break, Hot Leg, without HPSI, nitrogen injection
BETHSY 6.2Tc	6” Break, Cold Leg, Counterpart Test
LSTF SBCL 21	6% Break, Cold Leg
LSTF SBCL 09	10” Break, Cold Leg
OTHER TRANSIENTS	
BETHSY 6.8	4” Break, RHR Line, RHR system in operation
BETHSY 4.3b	SGTR, 6 tubes
BETHSY 4.1aTc	Natural Circulation, Counterpart Test
PACTEL ISP33	Natural Circulation, VVER
BETHSY 5.2e	Total Loss of Feedwater

BETHSY 6.9c	Loss of RHR (saturated), (1-D and 3-D modelling of the core)
BETHSY 6.9d	Loss of RHR (non-condensable gases)
LOBI BT 12	Steam Line Break
LSTF TR 03	Total Loss of Electrical Power

CONTAINMENT TRANSIENT	
CVTR 3	BREAK in the primary circuit and saturated steam removed to the containment.
CVTR 5	BREAK in the primary circuit and saturated steam removed to the containment with containment spray actuation
HDR T31.5	ISP23: LB-LOCA in the primary side and fluid removed to the containment

3. WHAT IS AN IMPLICIT CODE?

REFERENCE:	**0 –The CATHARE code: SMTH/LMDS/EM/2001-063**
KEY WORDS	**above**
See also	

The CATHARE code uses an implicit numerical scheme for solving the system of equations (except for the 3D-module). For instance an equation of the form:

$$\delta F/\delta t + \delta(F.V)/\delta z = G \quad \text{with F, G, V functions of time t and space z}$$

is discretised:

$$(F(t+\delta t)-F(t))/\delta t + \Delta(F(t+\delta t).V(t+\delta t))/\Delta z = G(t+\delta t)$$

This means that the numerical scheme is not centred in time. This method has the following properties:

- In fluid mechanics this method is known to be **unconditionally stable**, in which case time steps are not limited by stability.
- High velocities in short meshes are possible with large time steps, thereby enabling **critical flows** through a nozzle to be calculated with a 1-D model.
- The numerical scheme is known to be **diffusive** and dissipative. In certain applications, the user must pay attention to numerical diffusion. If it is necessary to reduce the numerical diffusion, use small dt and small meshes (see *HOW TO CONTROL the TIME STEP?*, *HOW TO CHOOSE a NODALISATION FOR AN AXIAL ELEMENT?* and *HOW TO CHOOSE a NODALISATION FOR A 3D ELEMENT?*).
- An implicit code makes it very easy way to determine whether a **stationary state** has been reached. Just apply constant boundary conditions and let the time step increase by itself. When a large time step is reached (100 sec or better 1000 sec) the stationary state is reached.
- **Very long transients can be calculated with large time steps**. Mass and energy balances are rigorously verified in CATHARE (a small balance error for energy when using the reflooding sub-module and for mass and energy in the 3D module may be encountered). This enables very slow and long transients with large time steps to be calculated without introducing any numerical pressure or temperature drift.
- In an implicit code, there is always something explicit. Trips actuated from the input deck are explicit, as well as heat exchanges between walls and hydraulic elements in reflooding and radio-chemical components transport calculation. The heat exchangers also can make an explicit coupling between two hydraulic elements or two circuits – the choice as to be made by the user (see *HOW TO MODEL a HEAT EXCHANGER?*). This may require some time step limitations (see *HOW TO CONTROL the TIME STEP?*).
- The 3D element time discretisation is semi-implicit. Using this scheme, the time step is limited by the Courant-Friedrich-Levy (CFL) time step. This also means that at the junctions between a 3D element and the rest of the circuit (1D or 0D and fully implicit), mass and energy balances may not be strictly respected (structural loss in circuit balances - see *HOW TO CHECK the ENERGY and MASS BALANCE?*)

4. WHAT ARE THE AVAILABLE OPTIONS?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019** **0 –The CATHARE code: SMTH/LMDS/EM/2001-063**
KEY WORDS	**CONVERGENCE, PARALLEL COMPUTATION, REGULATIONS,OPTION**
See also	

Several calculation management options are available in CATHARE.

You can assign:

4.1 Numerical options

- time step limits (DTMIN, DTMAX options)
- numerical features: minimum number of iterations per time step (ITERMIN option), maximum number of time step divisions (MAXREP option)

4.2 Physical options

- choice of physical grid (set of closure equations validated by the CATHARE team) (GRIDS)
- special treatments to make easier/faster regulations (ROCP, REGUL options)

4.3 Computing options

- OMP parallel computing level (PARALLEL option)
- not saving CPU time used for each time step (NCPUSAV option) for binary comparisons.
- the format of real numbers saved in result file (NCPUSAV option)

You can access advanced information about:

- simulated scenario (HISTORY option)
- convergence (IMPI option, VERBOSE directive)

5. WHAT ARE THE CATHARE COMPUTATIONAL VARIABLES (CCVS)?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019**
KEY WORDS	**CCV, REGULATIONS**
See also	

The **CATHARE Computational Variables** (**CCVs**) are variables calculated by the code which are accessible from the input deck.

They can be used in **read and/or write** mode.

Arithmetic operations between CCVs are possible.

In the READ option, they can be used for:

- representing a measurement, a signal;
- programming trips, operator actions;
- regulations;
- transient management.

In the WRITE option, they can be used for:

- reinitialising variables;
- imposing flowrate and enthalpy of an external source or total flowrate of an external sink.

For radio-chemical components, the WRIBA and VALBA directives can be used instead of the WRITE and VALUE directives. But access to the values is faster using WRITE and VALUE directives.

WCIRCBA is used to apply a radio-chemical CCV to an entire circuit.

Examples:

- represent a level measurement from calculated pressures or from calculated void fractions and densities;
- actuate Emergency Core Coolant System (ECCS) flowrate from a pressure signal, open steam relief valves when certain conditions are reached, etc;
- regulate SG level, primary pressure, etc;
- start reflooding calculations when ECCS water enters the core;
- reinitialise temperatures, void fractions in order to reach a desired state faster.

Comments about the use of the WRITE directive:

- The WRITE directive used to impose source, sink or other user controlled variables constitutes a normal use of the code.
- The WRITE directive used to change principal variables of the code is a violation of the physics of the code and must not be considered as a normal use of the code. It is a trick which does not follow quality assurance principles.
- The WRITE directive changes the variables calculated by the code and may destabilise the equations. After a WRITE directive, the time step may be reduced drastically as the new thermal-hydraulic state may be far from an equilibrium state.

- **Never consider that you have a steady state after a WRITE directive. A stabilised transient is always required after a WRITE directive.**

- Be careful with the new thermodynamic state, and avoid creating sub-cooled vapour conditions.
- Be careful with the new thermodynamic state, and avoid creating sub-cooled liquid in the presence of gas or superheated vapour in the presence of liquid and avoid creating superheated liquid conditions.
- Changing gas variables P, H_G, Xi when non-condensable gases are present may be very difficult and dangerous.
- If you change a variable in all nodes of an element, change also the value at the downstream junctions.
- Do not try to use a WRITE directive at every time step to simulate a boundary condition such as: imposing a fluid temperature, imposing a wall temperature, etc. The directive will not do correctly what you want it to do.
- The WRITE directive may change the mass and the energy in an element and in a circuit. Consequently, after a WRITE directive, the mass errors and energy errors printed by CATHARE may be very high.

6. WHAT ARE THE FILES USED BY CATHARE?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019**
KEY WORD	**FILES**
See also	

A CATHARE calculation uses three executables, and each of them requires/generates some files containing data:

- Files for **reader** execution
- Files for **code calculations** (also **restart** or **stand alone** calculation)
- Files for **post-processing** execution

Remark: Many files other than those which follow can be opened by CATHARE (during the three previously defined executions). Consult the overall list in the User Manual. Only the most important or necessary files are given below.

6.1 Files for reader execution

Input files: Input deck

Output files:

- File containing the image of the input deck data block (V25.INIT)
- Fortran file PILOT.f, translation of input deck execution block

6.2 Files for code calculations

Input files:

- Dictionary of key words used by CATHARE (DICO)
- CATHARE memory dimensioning files FAST.H and FASTSIZE.f

And also

Standard calculation

- File containing the image of the input deck data block (V25.INIT)
- Fortran file PILOT.f, translation of input deck execution block

Restart calculation

- Files saved (or stacked) by the user. They contain what is necessary for a restart calculation together with objects or numbers not attached to the circuit and saved in a table for use after a restart.
- File containing the saved state of the reactor V25.RESTART
- Fortran file PILOT.f, translation of input deck execution block

Fuel stand-alone calculation

Input file containing calculated hydraulic parameters necessary for a stand-alone fuel calculation

Output files:

- Listing
- Result file (if RESULT directive used, FORT21)

6.3 Files for post-processing

Input files:

- Result file containing saved values (FORT21)
- File containing the chosen definitions of variables for post-processing
- Files containing the variables corresponding to experimental values to be compared to calculated values - may be used for post-processing

Output file:

- Formatted file with the chosen variables (FORT07)

7. WHAT ARE THE COMMON SYNTAX ERRORS?

REFERENCE:	
KEY WORD	**ERROR**
See also	**HOW TO READ ERROR MESSAGES?**

Some SYNTAX ERRORS are very common when writing an input deck:

7.1 General:

- Do not forget the ; (semi-colon) at the end of a directive or an operator.
- Do not exceed 80 characters per line.
- The comment character * must be in the first column.
- Some editors put a tabulation character after a return. You must erase it.
- Do not define a variable with the name of an operator or a directive or a key word.
- Do not define anything with a name having more than 8 characters. CATHARE uses only the first 8 characters and does not stop during the reader process. Only upper case and lower case alphabetic letters, numbers and the "_" (underline) characters are allowed.

7.2 Sub-modules:

Gadgets have not only to be defined by the corresponding operator in the data block but also generally have to be activated to interfere in calculation in the executable block (see What is a sub-module?).

Do not forget to define thermal-hydraulics laws (especially non-condensable gas mass fraction if necessary) for the gadget before activating it.

7.3 Restart calculation:

- Do not forget to initialise time and time step before any calculation.
- We recommend that you avoid putting any directive (such as activating a gadget, using RESETIME directive, changing laws…) at the end of a calculation (between TRANSIENT and SAVE directive).

8. WHAT ARE THE FLUIDS AVAILABLE IN THE CATHARE CODE?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019**
KEY WORD	**GAS COOLED REACTOR**
See also	

The CATHARE code is devoted to best estimate calculations of thermal-hydraulic transients in nuclear reactors.

The main objective of CATHARE 2 V2.5_1 is to perform safety analyses for Pressurized Water Reactors postulated accidents or other incidents, such as primary LOCAs, SGTR, loss of RHR, secondary breaks, loss of feed-water, etc.

The code is based on a water 2-fluid 6-equation model including non-condensable gases equations and additional equations for radio-chemical components transport. It allows the coolant circuits of any reactor to be represented by assembling 1D, 0D and 3D modules.

CATHARE may also be used to perform gas cooled reactor calculations (Gas Turbine Modular Helium Reactor) and gas single phase flow may be simulated using the MONOPHASE directive (see *HOW TO CALCULATE a GAS SINGLE PHASE FLOW?*) with steam properties modifications (option WATERMOD of FLUID directive).

In the future, new fluids (HELIUM, SODIUM, HYDROGEN etc) will be available to model other nuclear reactors (GFR, SFR, SCWR, VHTR) or other facilities.

9. WHAT IS A REACTOR?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019**
KEY WORD	**REACTOR**
See also	

A reactor is an assembly of several circuits. You can define only one reactor per input deck. The number of circuits is limited to 4; only the first one will be considered as the primary circuit in terms of time-step management.

When a steam generator tube rupture is defined or when circuits are implicitly coupled, the same non-condensable gases and radio-chemical components have to be defined in both circuits. In particular, the GOBORA directive will be launched for the reactor.

The MANAGE directive can be used to manage the time cycle for the reactor. It compels all circuits of the reactor to reach/converge with a given time step – cycle time step. Several sub-time steps may be computed, if necessary, to reach the cycle time. This option is generally used in cases where CATHARE is coupled with simulators.

Specific directives for a reactor are:

- GOBORA: launching radio-chemical components transport calculation,
- GOFUEL: launching thermo-mechanical calculation for FUELCHAR elements of the primary circuit,
- INIBIL: (re)initialisation of all balances (water, non-condensable gas and radio-chemical component MASS and ENERGY balances),
- MANAGE: management of time step,
- REACTOR: loading reactor data after RESTORE directive,
- RESETIME: changing simulation time,
- TRANSIENT / TRANSENS: calculation of one time step with/without sensitivity calculation preparation

10. WHAT IS A CIRCUIT?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019**
KEY WORDS	**NONCONDENSABLE GAS, ACTIVITY, BORON, JUNCTION**
See also	

For the computation, a circuit is modelled with several **elements**: pipes, capacities, 3D or boundary conditions.

In CATHARE, a **circuit** is the assembly of all these elements connected by **junctions**.

A circuit may be open (with boundary conditions) or closed.

Non-condensable gases may be defined in a circuit. Physically, they will come from the accumulator (nitrogen), or from outside (nitrogen coming from outside through a break), or from the oxidation of claddings (hydrogen). In a calculation it can be injected in a circuit through a boundary condition or by a source.

Don't forget to define non-condensable gases in the circuit if they are necessary in the following cases (this will avoid error messages):

- **ACCU** (accumulator) defined with the NOCLOSE option if gas flows into the circuit after the injection of water.
- **ASSIGN** directive with the gas concentration in the containment for a guillotine RUPTURE.
- **MODEL** and **BCMOD**: boundary condition models. If necessary, give the mass fraction (Xi) for each non-condensable gas defined.
- **SOURCE**: (injection in an element) defined with non-condensable gases.

Boron and/or **activity** may be defined in a circuit. The boron will be transported by the liquid phase and the activity by both phases. Physically, it is used to represent the core leaks of activity and boron injection due to safety facilities and Chemical and Volume Control System (see How to model Radio-chemical components?).

Connection of circuits:

- thermal connection with heat exchangers
- hydraulic connection with SGTR (steam generator tube rupture).

Specific directives for a circuit are:

- ACTEMIS: activation of activity emission in the core.
- GOBORA: launching radio-chemical component transport calculation. The GOBORA has to be used with the REACTOR when an IMPLICIT heat exchanger is used.
- INIBIL: (re)initialisation of all balances (water, non-condensable gas and radio-chemical component MASS and ENERGY balances).
- INIBORA: (re)initialisation of radio-chemical component mass concentrations or activities.
- WCIRCBA: imposes radio-chemical component mass concentrations or activities in the entire circuit (mass balances should be reinitialised in the circuit after this operation).

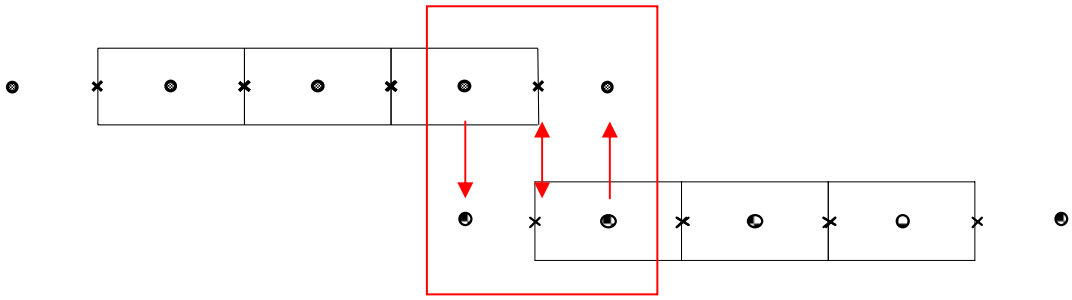
11. WHAT IS A JUNCTION?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019**
KEY WORD	**JUNCTION**
See also	

A **junction is the connection** between two adjacent modules.

A junction is either upstream or downstream of the element. This choice is of no importance in terms of flow and is a topological control for the element and the circuit. In an element (except boundary condition) at least one upstream and one downstream junction should be defined; in a circuit, each junction should appear in two, and only two, different element definitions, each with a different direction key word.

For each element, a junction consists of an assembly vector node and a scalar node which is a copy of the adjacent module node.



The two phase velocities Vl & Vg are defined at the vector node and the two momentum equations are centred on this node. Due to the solution method, the momentum equations must be written by one of the adjacent modules.

More precisely, in the general case, these two momentum equations are written by the module which is upstream of the liquid velocity.

Exceptions:

- The axial element always writes these equations when it is connected with a THREED element or a volume element.
- BC4 or BC5 boundary condition never writes these equations.
- A blind boundary condition or a safety valve boundary condition always writes these equations.
- The THREED element always writes these equations when it is connected to a VOLUME element.

In element definitions, the user has to ensure continuity of weight, gravity and geometry (sections, perimeters) at junctions.

Don't confuse junctions with boundary conditions

- Boundary conditions are modules without internal variables which write equations.
- Junctions are only variables common to two modules.

12. WHAT IS A MODULE?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019**
KEY WORD	**JUNCTION**
See also	

A module is a thermal-hydraulic model adapted to a certain type of element.

A module is a set of equations.

A module is a group of subroutines which writes and solves this set of equations.

A module has internal variables and junction variables. It writes equations on both internal and junction variables.

The following modules are used in CATHARE:

- AXIAL module (1D) for pipes and any element with a 1-D flow.
- VOLUME module (OD) to describe capacities with one or several connections.
- THREED module (3D) for description of PWR vessel in 3-D.
- BOUNDARY CONDITION module for extremity of a circuit.
- RUPTURE: double-ended break (approximately equivalent to two boundary conditions with a pressure law)

13. WHAT IS A SUB-MODULE?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019**
KEY WORD	**GADJET**
See also	

A sub-module is connected locally to a module and modifies its thermal-hydraulic behaviour.

A sub-module calculates additive terms for the equations of the connected module.

A sub-module is a group of subroutines, which calculates these additive terms.

A sub-module may have internal variables and internal equations to calculate the additive terms.

Two kinds of sub-module exist:

- **extended sub-modules (having a wide interaction with the module):**
  - multi-layer walls (WALL, WALL3D)
  - heat exchangers (EXCHANGER, *HOW TO MODEL a HEAT EXCHANGER?*)
  - fuel pins (FUEL, FUEL3D, FUELCHAR, FUELPLAQ)
  - a REFLOODING 2-D conduction calculation (REFLCHAR, REFLCH3D)
  - point kinetics (CORE)
- **sub-modules (or gadgets) connected to one mesh:**
  - pump (PUMPCHAR, *WHAT is a ONE-NODE PUMP?*)
  - gas turbine (TCOMCHAR, *HOW TO MODEL a TURBINE?*)
  - compressor (TCOMCHAR, *HOW TO MODEL a COMPRESSOR?**)***
  - 0D steam generator (SGCARACT, *WHAT is a ONE-NODE STEAM GENERATOR?*)
  - accumulator (ACCU, *HOW TO MODEL an ACCUMULATOR?*)
  - source (fill) and sink (loss) (mass: SOURCE/SINK; energy: CANDLE) (and other kinds of sink/source: PIQARE, PIQBREAK, PIQSOUP, PIQSEB, PIQREV, PIQVANNE, SINKRRI, SOURIS, *HOW TO MODEL a SOURCE?* and *HOW TO MODEL a SINK?*)
  - counter-current flow limiter CCFL
  - break (BREAK, *HOW TO MODEL a BREAK?*)
  - valves (FLOW-LIMITER, CHECK VALVE, CONTROL VALVE, ECHECK, ECVALVE, *HOW TO MODEL a VALVE?* and *HOW TO MODEL a SAFETY VALVE?*)
  - steam generator tube rupture (SGTR, *HOW TO MODEL a SGTR?*)
  - tee (TEE, *WHAT is a TEE?*)
  - turbine (TURBINE, *HOW TO MODEL a TURBINE?*)
  - sensor (SENSOR, *HOW TO SIMULATE FACILITY MEASUREMENTS*)

Except for walls, exchangers, sensors and tees, sub-modules do not interfere in calculation before user activation in the input deck.

For some extended sub-modules, specific key words are used for activating a calculation (they not only add source terms in hydraulic equations but also use a personal set of equations):

- fuel pins: GOFUEL directive
- reflooding 2-D conduction calculation: REFLOOD directive
- point kinetics: GONEUT directive

To activate gadgets, use the directives:

- ❖ OPEN or OPENBREAK /CLOSE for accumulators, break, candle, sink, source, pump, SGTR, turbine and valves (OPENBREAK is used for SGTR or PIQBREK).
- ❖ ENABLE/DISABLE for counter-current flow limiter.

Remark: a point steam generator is taken into account by using the POINTSG key word in the related wall.

More information can be found in the cross-reference table and in the thematic index of the dictionary.

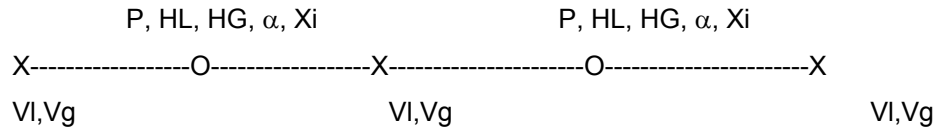
14. WHAT IS AN AXIAL ELEMENT?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019**
KEY WORDS	**AXIAL ELEMENT, NODALISATION**
See also	***WHAT is a SUB-MODULE?***

The axial element module is suitable for any pipe or duct in which the flow has a 1-D aspect.

Main features of the axial element **module:**

- It uses the classical 1-D, 2-fluid, 6-equation model (0 to 4 non-condensable gases).
- 6 equations: mass, momentum, energy balance equations for each phase + 0 to 4 transport equations for non-condensable gases.
- 6 main variables: (P, HL, HG, α, Vl, Vg) + (X1, X2, X3, X4 mass fraction of non-condensable gases). These principal variables represent average values over the cross-section.
- Qualified set of constitutive laws for interface transfers and wall transfers of mass momentum and energy.
- The discretisation is based on a staggered mesh with scalar nodes (for P, HL, HG, α, X1, X2, X3, X4) and vector nodes (for Vl and Vg)



Momentum balance equations are integrated from one scalar node to the next and centred on a vector node.

Mass and energy balance equations are integrated from one vector node to the next and centred on a scalar node.

Sub-modules may be connected to an axial element. See *WHAT is a SUB-MODULE?*

15. WHAT IS A VOLUME?

REFERENCE:	**-CATHARE 2 V2.5_1 - Description of the Volume module: SSTH/LDAS/EM/2005-037**
KEY WORD	**VOLUME**
See also	

15.1 Purpose

The volume module represents a capacity with several connections. It is adapted for capacities with large dimensions compared to the diameters of the junctions.

15.2 Main assumptions:

All thermal-hydraulic quantities are assumed uniform in horizontal planes.

- The velocities inside the volume are small compared to the velocities at the junctions.
- Inertial forces are assumed to be negligible compared to gravity forces. Consequently, the momentum equations are simplified and the pressure field is hydrostatic.
- Phase stratification occurs (void fraction stratification).

15.3 Model

The stratification is represented by a two-node model with two sub-volumes (Ω^-,Ω^+). The interface between the sub-volumes has a variable level. In each sub-volume, enthalpies and void fractions are assumed to be uniform but not the pressure which has a hydrostatic gradient. It is assumed that there is liquid in the lower sub-volume possibly with gas rising towards the interface. In the upper sub-volume there is mainly gas, possibly with liquid drops or falling jets.

Scalar variables are calculated inside the volume in front of each junction (at a distance from the junction equal to 2% of the volume height for vertical junctions). Flow distribution between sub-volumes and the phase sorting phenomena are modelled at each junction, taking into account the two-phase jet effect pull-through process.

Mass and energy transfer between the two sub-volumes (bubble rise, fall of droplets, condensation, evaporation) are modelled.

For the one-phase liquid case, the upper sub-volume is residual (height =1 cm). Respectively the lower sub-volume is residual for one-phase gas fluid (height = 1 mm).

15.4 Variables

- 4 +N X main variables for each sub-volume: P, HL, HG, α + N X (N number of non-condensable gases and X mass fraction of non-condensable gases),
- the interface level Zc,
- (6 + N non-condensable) variables at each junction (the velocities being no longer negligible at the junction).

15.5 Steady state

When the level value is specified for the steady state (LEVEL directive), both sub-volumes are initialised with saturation conditions.

15.6 Available sub-modules

Sub-modules may be connected to a VOLUME. See *WHAT is a SUB-MODULE?*

16. WHAT IS A 3D ELEMENT?

REFERENCE:	**-Physical improvement of the 3D module of CATHARE 2 V1.5B: SMTH/LMDS/EM/2002-084** **-CATHARE 2 V2.5 - Description of the 3D module: SSTH/LDAS/EM/2004-048**
KEY WORD	**THREED**
See also	**WHY-WHEN to use a 3D ELEMENT?**

16.1 Purpose:

The main purpose is to represent large-scale thermal-hydraulic 3D effects in nuclear power plants. One of the main applications is the modelling of a PWR vessel. The 3D module has been only validated for PWR vessel modelling. The use of the 3D module can be extended to other geometries but the validation of such geometries has to be performed by users.

16.2 Main assumptions:

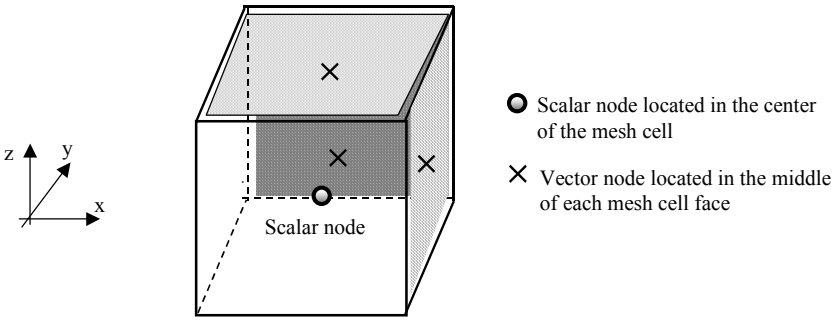
- The Z-axis is vertical.
- The main phenomena to be addressed are the three main phases of a large break LOCA, i.e., blowdown, downcomer refill and core reflooding phases, for which turbulent phenomena are not dominant. Therefore the turbulent model is dedicated to very specific applications where the model is applied to a single phase.

16.3 Main features:

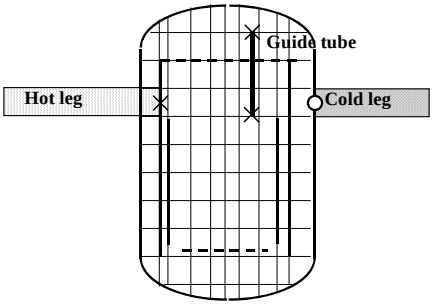
- A 3D ELEMENT is based on the two-fluid 6-equation model. The basic set of equations consists of 10 thermal-hydraulic differential equations:
  - two mass balance equations (one for each phase)
  - two internal energy balance equations (one for each phase)
  - two momentum balance equations (one for each phase) for the three coordinate directions.

  The mass and energy balance equations are of primary form whereas the momentum equations are of secondary form.
- Up to 4 non-condensable gas transport equations can be added.
- (k,ε) model for each phase (liquid and gas mixture) is available but there is no general model for two-phase flow turbulence.
- The main CATHARE variables are: P, H_l, H_g, α, $\mathbf{V}_l$, $\mathbf{V}_g$, X_i (i=1,4), (k_l,ε_l), (k_g,ε_g)
- A qualified set of constitutive relationships directly extrapolated from those of the axial element, the 1D correlations, extended over the three directions, with some specific features (no stratification, no added mass term).
- The numerical features are:
  - finite volumes
  - structured mesh
  - rectangular and cylindrical coordinates
  - first order discretisation in space and time

- o staggered spatial mesh
- o donor cell principle



- ❖ The time discretisation scheme is semi-implicit.
- ❖ Mass and energy flux terms are time-explicit (time step bounded by CFL stability criterion related to fluid velocity).
- ❖ Pressures and velocities are time-implicit in the momentum equations.
- ❖ Organisation equivalent to other hydraulic modules (0D, 1D, BC).
- ❖ Two types of junction are available:
  - external (standard) junctions, for cold leg connections for instance
  - internal junctions, for guide-tubes or hot leg connections



x: internal junction
o: external junction

- ❖ Junctions based on a standard 10 principle variable map (p, hl, hg, α; vl, vg; p, hl, hg, α)
- ❖ 3D-3D junctions allowed.
- ❖ PERMINIT + STEADY limited to single-phase flow situations.

16.4 Available sub-modules:

- ❖ walls: defined either over the entire nodalisation or locally, mesh cell by mesh cell: *WALL3D*
- ❖ fuel rod models (thermal-mechanical behaviour): *FUEL3D* …
- ❖ point kinetics model for core neutronic behaviour
- ❖ reflooding model (quench front model + 2D local conduction): *REFLCH3D* …
- ❖ CCFL model
- ❖ Heat exchangers: internal (in a single 3D hydraulics)
- ❖ Pure energy source/sink (CANDLE)

16.5 Not available: standard source/sink

16.6 3D module user interfaces:

- Reader: zone definition for input data specifications
- Listing for input data control (VERBOSE LIST3D)
- Calculation: CCVs, listing and result on the basis of RESULT+PERIOD
- Post-processing: based on a 1D approach (3D mesh columns) => FORT07 variations

17. WHAT ARE THERMAL STRUCTURES?

REFERENCE:	**-CATHARE 2 V2.5 - Description of the wall, heat exchanger and 0D steam generator sub-modules: SSTH/LDAS/EM/2004-049**
KEY WORDS	**EXCHANGER, FUEL, HEAT LOSS, HEAT SOURCE, NEUTRONICS, WALL**
See also	***HOW TO MODEL CORE NEUTRONICS?*** ***HOW TO MODEL FUEL RODS?*** ***HOW TO MODEL WALLS?*** ***HOW TO MODEL a HEAT EXCHANGER?*** ***HOW TO INITIALISE COLD/HOT FUEL DATA?*** ***WHY/WHEN TO USE THE STAND ALONE FUEL?***

Three kinds of thermal structure are defined in CATHARE: walls, heat exchangers and fuel rods.

17.1 Main characteristics

The main characteristics of thermal structures are as follows:

- They exchange heat with fluids (AXIAL, THREED or VOLUME elements).
- Heat conduction is calculated in 1D (in a direction perpendicular to the fluid flow).
- Heat conduction equations are implicitly coupled with the fluid equations (through the thermal heat fluxes).
- The wall structures are either **cylindrical** or **plane**. The same physical law are used in the two cases.
- In case of a cylindrical geometry, the walls can be :
  - **external**: heat is exchanged with the fluid at the smallest radius.
  - **internal**: " " " " " " " largest "

The geometrical data which define a thermal structure are:

- two radii (cylindrical case) or two lengths (plane case) which define a **wall thickness** used for thermal conduction calculations.
- an **exchange area** with the fluid (this is obtained from a heating perimeter multiplied by a mesh length)

Each elementary wall, exchanger, fuel, is assigned to a hydraulic scalar mesh in the axial direction. Each scalar mesh may have several elementary walls, exchangers, fuels in the direction perpendicular to the fluid flow.

A wall-structure, an exchanger-structure or a fuel-structure is thus defined as a set of elementary walls, exchangers or fuels respectively.

17.2 Characteristics for walls:

- Several layers of different materials
- Power generation in one heating medium
- External heat exchanges by user-defined heat losses (*HOW TO MODEL FACILITY HEAT LOSSES?*). By default, there are no external heat exchanges
- For a WALL connected to an AXIAL or a VOLUME (and not for WALL3D) or a plate fuel (FUELPLAQ for a 1D module), a contact resistance (option RESIST of WALL) can be added between two layers. The heat transfer coefficient (h) input in the data deck is used to add a term (1/h*e) to the thermal conductivity calculation of the last or first mesh of one of the two considered layer (e is the thickness of the considered mesh) . The considered layer is defined using the LEFT or RIGHT keyword. The heat transfer coefficient is either given by the user (VALUE) either calculated by the code from the user's

material properties (MODEL). This model can be used to model an actual resistance contact or to avoid the modelling of a given material (a fine painted layer for example).

17.2.1 Walls defined on AXIAL elements:

- Walls are cylindrical or plane.
- Walls are defined by segments between two vector points.
- Several walls can be connected to each scalar node of the hydraulic 1D mesh.
- The exchange area is defined by the product of a heating perimeter (HPERIM) and the axial length of the hydraulic mesh (MESH).

17.2.2 Walls defined on 3D elements:

- Walls are cylindrical or plane.
- Walls are defined by segments between two vector points in the directions (X, Y, Z) or (R, THETA, Z).
- Several walls can be connected to each scalar node of the hydraulic 3D mesh.
- The exchange areas are given directly by the user (HSURF).

17.2.3 Walls defined on VOLUME elements:

- Walls are vertical and cylindrical or plane.
- They are defined between two elevations (zmin, zmax).
- Several walls can be connected to a volume element.
- The exchange area is defined by the product of a heating perimeter (HPERIM) and the length of the wall (zmax - zmin).

17.3 Characteristics for heat exchangers:

Two types of exchanger can be defined:

- an (external) heat exchanger between two circuits (defined between two axial elements belonging to two different circuits)
- an (internal) heat exchanger within a circuit or within a 3D element.

In each previous case, one side of the heat exchanger is considered as primary while the other side is considered as secondary (*HOW TO MODEL a HEAT EXCHANGER?*).

17.4 Characteristics for fuel rods:

Fuel rods can be defined on axial and 3D elements (see *HOW TO MODEL FUEL RODS?*). The objectives of fuel rod modelling are to describe:

- The thermal behaviour of the rod (UO2, cladding)
- The mechanical behaviour of the rod (cladding deformation and rupture)
- The oxidation of the cladding.

The neutronic power which is physically emitted by the fuel is either imposed in the fuel data (FUELCHAR, FUELP) or calculated by the point neutronic module of CATHARE (CORE, NEUTRO).

In CATHARE, there are two ways of calculating the fuel rods (see *HOW TO MODEL CORE NEUTRONICS?*):

- **standard fuel rod**: it behaves like a standard wall, directly coupled to the hydraulics.
- **stand-alone fuel** rod (*WHY/WHEN TO USE THE STAND ALONE FUEL?*): the hydraulic conditions, deduced from a previous CATHARE calculation, are imposed on the fuel rod; there is no feed-back from the fuel rod to the hydraulics (*HOW TO INITIALISE COLD/HOT FUEL DATA?*).

18. WHAT IS A ONE-NODE STEAM GENERATOR?

REFERENCE:	**-CATHARE 2 V2.5 - Description of the wall, heat exchanger and 0D steam generator sub-modules: SSTH/LDAS/EM/2004-049**
KEY WORDS	**STEAM GENERATOR**
See also	

18.1 Purpose:

This model gives a very simple possibility for heat exchange modelling with a secondary circuit.

18.2 Model description:

It is a sub-module of an axial element.

A point SG represents a secondary circuit in a very simplified way: the secondary circuit is represented by only one mesh. It may exchange power with several successive meshes of an axial element. It is connected to the primary meshes through wall structures where the heat conduction is calculated in the same way as in a standard wall with a heat loss.

The secondary circuit has a constant volume, and is defined by three principal variables:

- the pressure P
- the liquid mass Ml
- the steam mass Mv

Both phases are assumed to be at saturation temperature. Three equations are written:

- the liquid mass balance equation
- the steam mass balance equation
- the total energy equation

The feedwater flowrate, feedwater temperature, and the steam turbine flowrate must be defined by the user as a function of time.

The steam valve flowrate can also be defined by the user as a function of pressure.

The heat exchange coefficient from tube walls to secondary fluid is a constant. This coefficient may be optionally readjusted during the automatic steady state in order to be able to exchange the appropriate power with the appropriate temperature difference between primary and secondary fluids.

'Fouling', increasing the thermal resistance due to a deposit on tube walls may be taken into account. The corresponding additional thermal resistance may be optionally adjusted during the automatic steady state in order to be able to exchange the appropriate power with the appropriate temperature difference between primary and secondary fluids.

The secondary pressure may be optionally readjusted during the automatic steady state in order to be able to exchange the appropriate power with the appropriate temperature difference between primary and secondary fluids.

19. WHAT IS A TEE?

REFERENCE:	**-CATHARE 2 V2.5 - Description of the PIPE module and TEE sub-module: SSTH/LDAS//EM/2004-065**
KEY WORD	**TEE**
See also	

The tee sub-module is a lateral branch which must be linked to an axial element. An axial element can carry as many tees as needed, but it is not possible to connect several tees at a single location of a given axial element. Therefore, the main pipe having by default two (upstream and downstream) junctions will receive additional junctions placed at tee locations. All directions of the main pipe and of the branch and all angles between their axes are possible.

Internal variables: P, Hl, Hg, α, (X1, X2,X3,X4)

Junction variables: P, Hl, Hg, α, (X1, X2,X3,X4), Vl, Vg

Equations:

- two mass balance equations
- zero to four non-condensable gas mass balance equations
- two energy balance equations
- two momentum equations
- closure equations at the junctions (for void fraction, enthalpies, X1, X2, X3, X4, pressure) according to classical rules

The Tee sub-module uses the standard constitutive laws of the pipe module.

Specific constitutive laws were developed for phase separation phenomena when the flow is stratified in the main pipe and when the flow is exiting towards the branch.

Special attention was paid to the momentum equations in order to predict correctly the pressure losses due to a change in flow direction. These pressure losses have been validated only when a single lateral branch is connected (the tee weight is the same as the main axial weight). In case of multi-lateral branch, the pressure losses may not be accurate.

Special models are available for the tee-sub-module:

- TECOND is specially designed to model safety injection devices in which condensation phenomena around the fluid injection point play an important role.
- JETPUMP is specially designed to model jet-pump injectors where the momentum in the axial element is strongly driven by the fluid injection. This model has to be used with caution because :
  - It modifies the momentum equations in the main branch. The qualification of this model relies on restricted access geometric data and experimental results.
  - It is not validated in two phase flow situations, so it has not to be used in such situations (ENABLE/DISABLE directive can be use to respectively activated or disactivated the model).

20. WHAT IS A ONE-NODE PUMP?

REFERENCE:	**-CATHARE 2 V2.5: Description of the PUMP 0D sub-module: SSTH/LDAS/EM/2004-041**
KEY WORDS	**DEGRADATION (PUMP), PUMP, HEAD**
See also	***HOW TO USE the ONE-NODE PUMP SUB-MODULE?***

The one-node pump model describes in a simple way the interaction between a centrifugal or mixed pump and a single-phase or two-phase flow. It writes the momentum and energy source terms due to the pump calculated from the head and torque characteristics.

It is located at a vector node. The momentum source term is assigned to the momentum equations written at this vector node and influences the pressure difference DP between the two neighbouring scalar nodes. The phase distribution of this DP term is chosen so that the phase slip ratio is not significantly affected by the pump. The energy source term is assigned to the energy equations written at the scalar node which is downstream of the liquid velocity.

An equation is written to calculate the rotation speed taking into account the motor torque, hydraulic torque, friction torque and the rotating inertial term.

$$I \cdot \frac{d\Omega}{dt} = C_m - C_h - C_f$$

The head H and specific torque $T = C_h/\rho$ depend on:

- Q the volumetric flowrate at the node of the pump
- Ω the rotation speed of the pump
- the pump characteristics
- $NPSH_{3\%}$, onset of cavitation (optional)

Reference values are defined:

$$\Omega_R,\ Q_R,\ H_R,\ T_R$$

Normalised values are defined:

$$\omega = \Omega/\Omega_R,\quad q = Q/Q_R,\quad h = H/H_R,\ t = T/T_R \text{ and } n = NPSH_{3\%}/H_R$$

Homologous functions are used:

$H = h/q^2 = f(\omega/q)$	or	$H = h/\omega^2 = f(q/\omega)$
$T = t/q^2 = g(\omega/q)$	or	$T = t/\omega^2 = g(q/\omega)$
$N = n/q^2 = \gamma(\omega/q)$	or	$N = n/\omega^2 = \gamma(q/\omega)$

The user gives as input data:

- the single-phase homologous characteristics for head H_1, torque T_1 and the optional N, the net positive suction head.
- the fully degraded homologous characteristics for head H_2 and torque T_2
- the degradation functions M_H and M_T (which are simple functions of the void fraction α)
- the degradation function in case of cavitation M_c (function of the ratio $NPSH_{disp}/NPSH_{3\%}$)

- $H = H_1 + (M_H(\alpha) + M_c)\,(H_1 - H_2)$
- $T = T_1 + (M_T(\alpha) + M_c\,(T_1 - T_2)$

- the pump rotor inertia
- the data needed to calculate the motor torque
- the data needed to calculate the friction torque.

Homologous characteristics and degradation functions are given for a list of operating points in the first three quadrants. Homologous $NPSH_{3\%}$ and cavitation degradation functions are given for a list of operating points in the first quadrant. The code uses an interpolation method to calculate them at any operating point.

The user does not have to give the precise geometrical data of the pump. However, the volumes and elevations must be respected. Don't forget that the volume flow rates are calculated with the cross-sectional area of the pump vector node.

21. WHAT IS A BOUNDARY CONDITION?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019** **0 –The CATHARE code: SMTH/LMDS/EM/2001-063**
KEY WORDS	**BOUNDARY CONDITION, MANWAY, GUILLOTINE RUPTURE**
See also	***HOW TO USE the BCMOD command?*** ***HOW TO USE the BCMOD command?*** ***HOW TO CHOOSE and to use a BOUNDARY CONDITION?***

A boundary condition is an element which can be placed at the extremity of a pipe, a volume, a tee or a 3D module. It is used to impose one or more hydraulic conditions for each phase (pressure, enthalpies or temperatures, velocities or mass flowrates for gas or liquid, the void fraction or the mass fraction of non-condensable gases or radio-chemical components). These boundary conditions can be defined at the inlet or the outlet of an element.

The choice of the type and number of boundary conditions to impose takes account of the characteristic velocities:

- Liquid enthalpy is imposed when the liquid enters.
- Gas enthalpy and non-condensable gas qualities are imposed when the gas enters.
- Pressure is imposed when the liquid is flowing out (subsonic flow is assumed).
- Void fraction is imposed when the liquid is entering (torrential flow is assumed).
- A choked flow condition may also be imposed at a break (sonic velocity is assumed).

A particular MODEL has been developed for a MANWAY (*HOW TO MODEL a MANWAY?*).

The RUPTURE type element is a double boundary condition. Set between two pipes, it models the opening of a guillotine rupture (*HOW TO MODEL a BREAK?*)

22. WHAT IS THE REFLOODING SUB-MODULE?

REFERENCE:	**-CATHARE 2 V2.5_1 : Physical laws used in reflooding sub-module: SMTH/LMDS/EM/2005-036** **- CATHARE-2 V2.5 : Description of the reflooding sub-module**
KEY WORDS	**QUENCH, REFLOODING, REWETTING,**
See also	***HOW TO USE the BOTTOM-UP REFLOODING SUB-MODULE?*** ***HOW TO USE the TOP-DOWN REFLOODING SUB-MODULE?*** ***WHY-WHEN TO USE a REFLOODING SUB-MODULE?***

22.1 Rewetting or quenching of a hot structure

When a heated structure is in post-CHF heat transfer with high temperatures, rewetting may occur if the cooling conditions become more favourable (arrival of enough liquid water). The standard wall heat transfer constitutive relationships are able to describe this process. The wall is first cooled in film boiling or gas convection conditions. When the wall temperature becomes lower than the rewetting temperature or minimum stable film temperature T_{mfs}, the transition heat flux provides much more rapid wall cooling and the wall temperature decreases below the burn-out temperature T_{bo}.

For the core reflooding phase of an LBLOCA, these standard heat transfer constitutive relationships underestimate the cooling of the rods and the velocity of the quench front along the rods. This has been attributed to two main reasons:

1. The axial heat conduction in the wall helps quenching. Heat from the hot dry zone of the wall is transferred axially towards the cold rewetted zone. This transfer accelerates wall pre-cooling and the rewetting temperature is reached earlier. This is not described with standard 1D-walls (only radial conduction is calculated).
2. In the vicinity of the quench front, the quenching process enhances heat transfers.

The classical boiling curve (Nukiyama's curve) is not unique and should depend on the sign of variation of the wall temperature. The critical heat flux value is valid in a burn-out or dry-out process but not in a slow rewetting process.

A reflooding sub-module has been implemented in CATHARE where the heat exchanges are different from the standard constitutive relationships. The reflooding sub-module is used to calculate fairly accurately the rewetting of a hot wall or rod or fuel (see *WHY/WHEN TO USE a REFLOODING SUB-MODULE?*).

Progressive quenching of the heated structure can be expected from bottom to top if water comes from the bottom or from top to bottom if water comes from the top. Two slightly different reflooding sub-modules are available for bottom-up or top-down quenching. They can be used separately or simultaneously.

22.2 Characteristics of the reflooding sub-module:

Compared to standard constitutive relationships (used in rewetted regions), the main modifications are:

Local 2d-conduction (axial and radial) calculation in the heated structure in the vicinity of the quench front. A very fine moving 2d-mesh is superimposed on the classical 1D-wall radial calculation and moves at the velocity of the quench front along the heated structure. The 2d-conduction calculation is automatically performed at each time step of the hydraulic calculation.

Dry regions: Specific constitutive laws for dispersed and inverted (annular or slug) flows. Effects of the spacer grids on the droplet diameter. Specific heat transfer model near the quench front, to take into account droplet impingement on the wall (Qpvi term). Standard radiative heat flux model.

Bottom-up reflooding sub-module and heat transfer enhancement: A local specific heat flux is added in the near vicinity of the quench front to represent heat transfers due to the flow perturbation by quenching. It is taken proportional to the axial wall temperature gradient ($\phi = K_2\ \delta Tw/\square z$). This term may be very high in a very limited area. In a more extended area above the quench front, an enhanced film-boiling type heat flux is added (Qpvi term) to account for the increase in wall to fluid exchange coefficient.

Top-down reflooding sub-module: Model upstream of the quench front: Wetted wall with a descending liquid film. Immediately downstream of the quench front: a specific heat flux (from the 2d-wall to fluid) model is used, in particular the Johannsen correlation for transition boiling. The CCFL (upstream of the quench front), which can limit film progression, is modelled by a Wallis local CCFL criterion (at the quench front) which stops the quench front progression.

23. WHAT IS A SINGULAR PRESSURE DROP?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019**
KEY WORDS	**PRESSURE DROP, SINGULARITY, FORM LOSS COEFFICIENT**
See also	

Singular pressure drops (or singularities) correspond to head losses due to geometrical effects (bend, area change, etc.). They are irreversible pressure drops.

They always have the following form (for a single-phase flow):

$$\Delta P_{sing} = \Delta(P + \rho V^2 / 2 + \rho g Z) = - K \rho V^2 / 2$$

Z being the elevation, K is a constant pressure drop coefficient given by the user.

In a two-phase flow, they take the following form:

$$\Delta P_{sing} = -\frac{K}{2}\left[\alpha_g \rho_g V_g |V_g| + \alpha_l \rho_l V_l |V_l|\right]$$

The pressure drop is distributed between both momentum equations with $\Delta P_{sing,g}$ and $\Delta P_{sing,l}$ respectively for gas and liquid equations with:

$$\Delta P_{sing,k} = -\frac{K}{2}\left[\alpha_k \rho_k V_k |V_k|\right]$$

An analysis of the singular pressure drop measured at the Steam Generator support plates in the Patricia SG2 experiment showed that this formulation was not appropriate for a local flow restriction. So, another option (optional key word SGPLATE in SINGULAR Directive) was implemented which has been qualified on Patricia SG2 tests:

$$\Delta P_{sing} = -\frac{K}{2}\left[\alpha_g |V_g| + \alpha_l |V_l|\right]\left[\alpha_g \rho_g V_g + \alpha_l \rho_l V_l\right]$$

It is distributed between both momentum equations with $\Delta P_{sing,g}$ and $\Delta P_{sing,l}$ respectively for gas and liquid equations with:

$$\Delta P_{sing,k} = -\frac{K}{2}\left[\alpha_g |V_g| + \alpha_l |V_l|\right]\left[\alpha_k \rho_k V_k\right]$$

K may depend on the flow direction and two coefficients K^+, K^- must be defined. The choice between $K^+$and K^- is made for each phase depending on the phase velocity direction.

Singular pressure drops can be introduced wherever needed, i.e.:

- Each vector node of an axial component
- Each junction of a volume
- Junction of a tee
- On any internal vector node of a 3D component related to each of the 3 axes (x, y, z or r, θ, z).

Singular pressure drops **cannot** be introduced at the following places:

- On any junction (internal or external) vector node of a 3D component when this junction is connected to an AXIAL component; in this case the singular pressure drop should be assigned to the corresponding junction of the AXIAL component.

24. WHAT IS A ZONE?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019**
KEY WORDS	**MASS BALANCE**
See also	

Zones in CATHARE input decks can appear with different meanings; two types of zones can be specified. The first deals with groups of hydraulic elements while the second deals with a subset of a 3D element.

The **first type** is associated with the key word ZONE; in this case a **ZONE is a set of elements which all belong to the same circuit**. The elements defining a zone must be AXIAL, VOLUME or THREED. Tees, boundary conditions and double-ended breaks are not allowed in a zone definition.

The **second type** is associated with the sub-key words **ZONEDEF**; this type provides a convenient means for structuring 3D input data (GEOM, PHYSCALE, etc.); 3D zones relating to scalar point nodalisation can ultimately be used to perform balance checking using the **BILAN3D** directive.

A zone generates additional calculations for balance checking over a chosen region of a CIRCUIT; it provides additional printout and post-treatment variables concerning:

- fluid mass and energy;
- non-condensable gas mass,
- wall energy,
- radio-chemical products

Examples:

If the reactor vessel is built by assembling 0D and 1D elements, then balance checking is possible for the whole vessel by defining a zone covering the downcomer, the upper and lower plenum, the bypass, the core and the upper head.

If the reactor vessel is built using the 3D element, then balance checking is possible for various regions of this vessel by defining zones covering subsets of the 3D mesh. For example, these subsets can address the lower plenum, the core, or any region of interest for the current study.

The period of printouts and post-treatment saves can be defined respectively by the directives PERIOD and RESULT applied on a ZONE as well as for the CIRCUIT.

25. HOW TO MODEL THE VESSEL?

REFERENCE:	**-Physical improvement of the 3D module of CATHARE 2 V1.5B: SMTH/LMDS/EM/2002-084** **-CATHARE 2 V2.5_1 - Description of the Volume module: SSTH/LDAS/EM/2005-037** **-CATHARE 2 V2.5 - Description of the 3D module: SSTH/LDAS/EM/2004-048**
KEY WORD	**VESSEL**
See also	

The pressure vessel can be modelled with a set of AXIAL and VOLUME modules or with a 3D module.

25.1 AXIAL and VOLUME modelling:

In 0D and 1D vessel modelling,

- The downcomer is represented with 1 VOLUME and 1 AXIAL element (see *HOW TO MODEL a DOWNCOMER?*)
- The lower plenum is modelled with 1 VOLUME (*HOW TO MODEL a LOWER PLENUM?*)
- The core is described with 1 AXIAL element
  - See *HOW TO MODEL CORE THERMAL-HYDRAULICS?*
  - See *HOW TO MODEL CORE NEUTRONICS?*
  - See *HOW TO MODEL FUEL RODS?*
- The core bypass is represented with 1 AXIAL element
- The upper plenum is modelled with 1 VOLUME (*HOW TO MODEL an UPPER PLENUM?*)
- Guide tubes are represented with 1 AXIAL element (*HOW TO MODEL a GUIDE TUBE?*)
- The upper head is modelled with 1 VOLUME. If thermal stratification is suspected to occur (case of low pump rotation speed and loop mass flowrate), it can be modelled with several VOLUMES.

25.2 3D modelling: (see *HOW TO MODEL the 3D VESSEL?*)

26. HOW TO MODEL THE 3D VESSEL?

REFERENCE:	**-Physical improvement of the 3D module of CATHARE 2 V1.5B: SMTH/LMDS/EM/2002-084** **-CATHARE 2 V2.5 - Description of the 3D module: SSTH/LDAS/EM/2004-048**
KEY WORDS	**THREED, VESSEL**
See also	

This section is not aimed at giving a complete description of the 3D vessel input deck. Its objective is to list all the elements necessary for 3D PWR vessel description, giving recommendations at each step. These recommendations result from the 3D module assessment. Whenever possible, a reference to a related section of the user guide is given. The description is based on modelling of the 900 MW CP1 reactor, but the methodology can be applied to any type of reactor. The following table is a summary of the required key words for the 3D vessel definition.

THREED	**→ element, name of junction**
MESH	**→ coordinates** **→ nodalisation** **→ orientation** **→ definition of scalar and/or vector zones**
CONNECT	**→ internal and external junctions** **→ open/closed edges**
GEOM	**→ mesh fluid volume** **→ edge fluid area**
HYDR	**→ nature of the flow: standard or rod bundle for scalar and vector equations**
PHYSCALE	**Definition of length scales (Dh)** **→ mesh (scalar equations)** **→ edge (vector equations)**
SINGULAR	**→ singular friction coefficients**
WALL3D	**→ walls (HSURF, heating surface)**
FUEL3D (FUELCHAR)	**→ fuel rods**
REFLCH3D	**→ reflooding characteristics**

THREED: → **see** *WHAT is a 3D ELEMENT?*

The vessel is modelled with only one 3D element. The name of the junctions belonging to the following categories must be specified:

- ❖ junctions for cold legs
- ❖ junctions for hot legs
- ❖ junctions for the inlet and outlet of the guide tubes (relevant only when guide tubes are represented with axial elements internally connected to the 3D module).

MESH: → **see** *HOW TO CHOOSE a NODALISATION FOR A 3D ELEMENT?*

coordinates: Cylindrical coordinates must be used, as the vessel is represented by a cylinder.

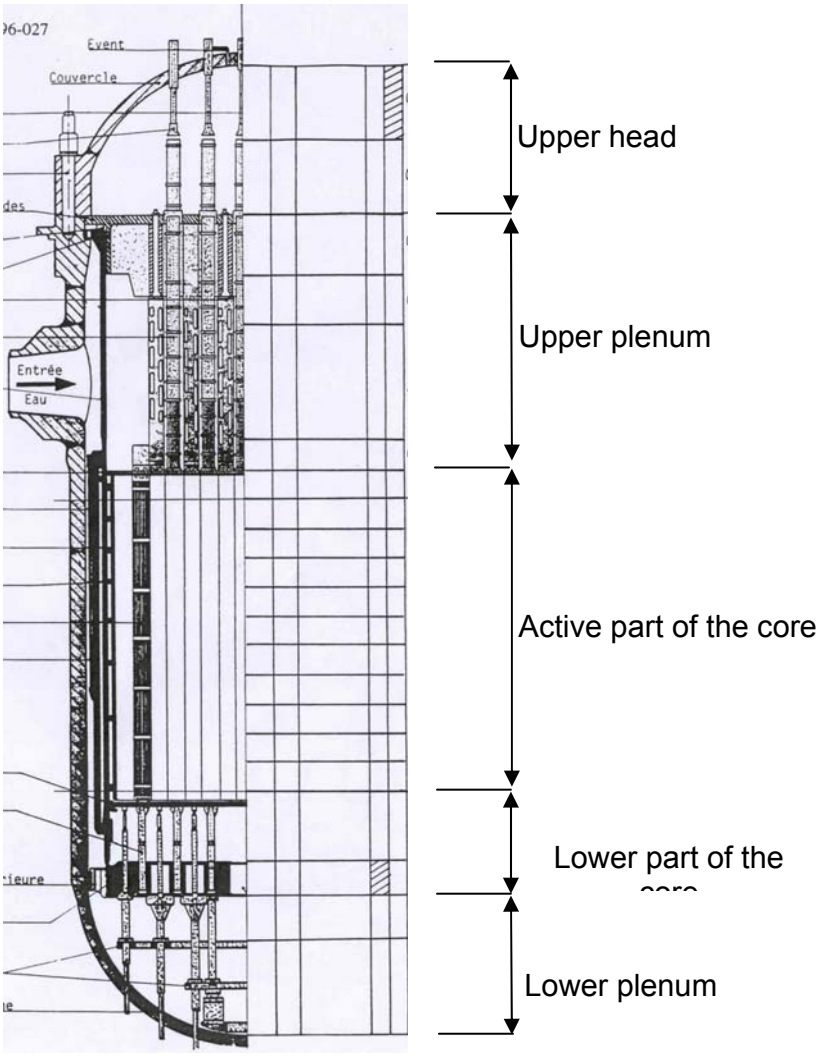
nodalisation: The standard model of the pressure vessel of a 900 MW CP1 reactor is subjected to certain constraints which are listed hereafter.

- Respect of the hot leg / cold leg elevation
- One hot leg / cold leg connection per mesh cell face. This implies that, for a three-loop reactor, there must be a minimum of three azimuthal meshes but it is recommended to use 6 azimuthal meshes as there are 3*2=6 legs (hot and cold).
- The fluid surface of the mesh cell faces where the hot leg / cold leg are connected is at least the size of the hot leg / cold leg cross-section
- In the active part of the core, the mesh must be regular and it must be coherent with the axial power profile imposed in the core.
- In the lower plenum and in the lower part of the core as well as in the upper head, the mesh is very much imposed by the geometry, but it is considered that mean flow is calculated. Therefore, the structures must not be described individually in terms of flow restriction but must be generalised. For instance when structures such as plates induce a strong flow restriction, they must not be modelled in terms of fluid surface but must be taken into account by means of singular pressure drops. The core upper tie plate is normally modelled in terms of a surface flow restriction and pressure drop.
- Because of the downcomer, it is important to model the bypass and the core in terms of hydraulic channels; there must be at least three radial meshes and the downcomer width and bypass width impose the size of the two external rings.

Therefore, the standard model of the pressure vessel of a 900 MW CP1 reactor is based on the following nodalisation:

- 5 radial meshes: one for the downcomer, one for the bypass and three for the core
- 6 azimuthal meshes: the number of connecting loops imposes the minimum number of θ-meshes, i.e., 6 θ-meshes for 3 loops
- 21 axial meshes.

Do not forget that the 3D module has been assessed on this given mesh. No nodalisation convergence has been performed. In case of others reactor or facilities modellings, it is recommended to use this kind of meshing in order to ensure the validity of the results. If it is not possible and if another meshing is used, the user has to check their nodalisation convergence.



Meshing of the PWR 900 MW 3D vessel using the 6x5x21 nodalisation

orientation: GRAVITY Z -9.81

definition of zones: It is highly recommended to define all the zones used in the definition of the 3D module all together in the input deck. It is advisable to divide the 3D vessel into zones which present a relative homogeneity. For example, a possible division of the 3D vessel is proposed hereafter:

- Lower plenum - Core support plate - Lower part of the core - Active part of the core - Upper plenum – Bypass - Upper head – Downcomer - ……

It is then possible to define the characteristics of the zones, which are independent of the characteristics of the other zones. This makes it easier to carry out possible modifications and reading of the input deck.

It is also necessary to define the zones that will be used in the BILAN3D directive.

CONNECT: → see *HOW TO DEFINE the CONNECTIONS of a 3D ELEMENT?*

The 3D vessel is connected only by means of 1D-3D junctions:

- 3 external junctions for the cold legs
- 3 internal junctions for the hot legs, declared as closed internal junctions
- internal junctions for the guide tubes, declared as open internal junctions

The position of the open/closed edges must then be given. This involves describing the flow domain, especially the impermeable walls, where no flow occurs through the edge considered. Due to the chosen modelling system, some meshes are totally closed. The fluid initially stored in these meshes cannot flow out. It should be noted that the mass balance performed with the directive BILAN3D will include the contribution of the closed meshes. To perform a correct mass balance, this contribution should be removed.

Remember that the axis of the cylindrical 3D element must be declared as closed.

GEOM: → see *HOW TO DEFINE the GEOMETRY of a 3D ELEMENT?*

In order to calculate the volumic and surface porosities, the vessel has to be considered in terms of homogeneous zones (see MESH section), described by mean quantities.

It is not recommended to attempt to obtain an exact model of the fluid surfaces (modelling of the plates for instance). An average fluid section can be used, imposing the corresponding singular pressure drop.

HYDR: → see *HOW TO CHOOSE the HYDRAULICS of a 3D ELEMENT?*

By default, the standard type of hydraulics is chosen for the vessel. Rod bundle hydraulics are used for the zone corresponding to the active part of the core. Specific hydraulics are imposed in the downcomer, using the DCOGRID key word.

PHYSCALE: → see *HOW TO CHOOSE the PHYSICAL LENGTH SCALES of a 3D ELEMENT?*

In order to calculate the volume and surface hydraulic diameter, the vessel has to be considered in terms of homogeneous zones (see MESH section), described by mean quantities.

SINGULAR: →see *HOW TO MODEL a SINGULAR PRESSURE DROP?*

As mentioned above, the different plates are not exactly modelled. Their resistance to the flow is described by means of singular pressure drops mainly imposed in the z-direction. Some of the main zones concerned are (the list not being exhaustive):

- the core support plate
- the active part of the core
- the bypass

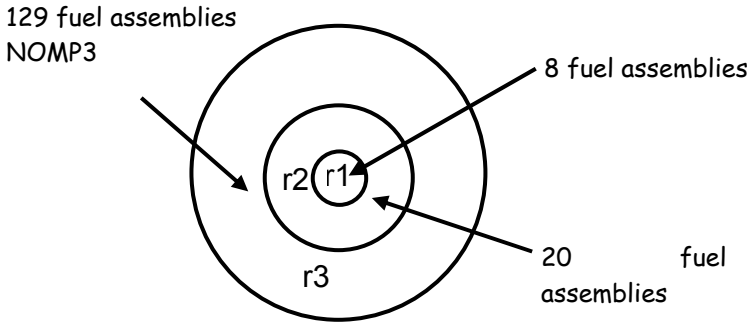
WALL3D: → see How to model WALLS?

Passive heat structures in the vessel are taken into account by means of WALL3D elements connected to the vessel hydraulic meshes. The use of the BILAN3D directive imposes the condition that a wall cannot belong to two different zones.

FUEL3D: → see How to model FUEL RODS?

The core model characteristics are:

- 180 computational cells (3 radial x 10 axial x 6 azimuthal divisions)
- core is assumed to be homogeneous (ε_v, ε_s, D_{hcell}, D_{hedge} calculated assuming uniform data)
- the number of fuel rods in each channel is based on the relative cross-sectional area of each computational cell



The fuel assemblies are modelled using the fuel 3D thermo-mechanical sub-modules FUEL3D and FUELCHAR. The fuel element description follows the three rings r1, r2 and r3. The hottest fuels are in the centre. The fuel assembly description is based on the 1D input deck. The fuel characteristic description is similar to the 1D model (INTERNAL/EXTERNAL, DATA/ISO, UO2, GAP, CLADDING, etc.).

If there are fuel structures with the same characteristics in different 3D cell columns, the description of the fuel structure does not have to be repeated in the input deck, thanks to the COPY key word of the AND operator.

REFLCH3D: → see How to use the REFLOODING SUB-MODULE?

GUIDE TUBE: → see *HOW TO MODEL a GUIDE TUBE?*

A homogeneous distribution of the guide tubes is considered. The weight of each axial element is determined in proportion to the mesh cell area. The total number of guide tubes is:

- 53 for a FRAMATOME-ANP 900 MW PWR
- 54 in the input deck (to keep a balanced distribution between the meshes: for each ring, 6 meshes along θ)

The radial distribution of the guide tubes could be as follows:

⇒

3D ring	Number of guide tubes per 3D mesh
r1 (inner ring)	1
r2	2
r3	6

The modelling of the guide tubes may depend on the physical situation. When a good geometrical representation is required (i.e., for LBLOCA transients), guide tubes may be simulated by means of axial elements internally connected to the 3D module with open internal junctions (sect, peri, hydr, singular, etc.). As this modelling process significantly increases the number of 3D-1D junctions, it also significantly increases CPU time. If the geometry of the guide tubes does not play an important role, their representation may be limited to open faces between the upper head and the upper plenum, with vertical closed faces distributed in the upper plenum.

Remarks:

- **Time step and CFL**: In the 3D module, the maximum time step is bound by the CFL time step. The variation in the CFL time step must therefore be checked before drawing any conclusion on the quality of the calculation time step. If the variation in the time step follows the variation in the CFL time step, the calculation is running normally. When the time step is very much smaller than the CFL time step, a numerical problem can be suspected.
- **2D modelling of a vessel:** In some specific applications, the vessel is considered axis-symmetrical. In that case, it is modelled using the cylindrical coordinates with only one mesh along θ. It is then compulsory to declare the θ edges as closed.

- Due to recent qualification studies, especially concerning the draining of the lower plenum during the depressurisation phase, it seems quite advisable to increase the number of meshes in the lower part of the lower plenum from 2 to 5 meshes.

27. HOW TO MODEL CORE THERMAL-HYDRAULICS (1D MODELLING)?

REFERENCE:	**-**
KEY WORDS	**CORE, HYDRAULIC**
See also	

The module representing the core of a PWR must be an AXIAL type element with hydraulics defined as RODBUNDLE (see *HOW TO CHOOSE the HYDRAULICS of an AXIAL ELEMENT?*).

Several parallel axial elements may be used to represent different power zones of the core. In this case no cross flows are modelled. Spacer grids must be represented by defining the associated singular pressure drops without representing the local flow area restriction (see *HOW TO MODEL a SINGULAR PRESSURE DROP?*).

The upper tie plate must be represented by defining the associated singular pressure drops without representing the local flow area restriction. However the CCFL is not correctly predicted and depends on the mesh size. Generally, the liquid down-flow is underestimated in counter-current flow conditions. This underestimation is more severe for fine meshes at the upper tie plate. A CCFL option may be used which controls the flooding limit according to a Wallis type or Kutateladze type correlation (see How to control CCFL?).

27.1 How to model core thermal-hydraulics for a Large Break LOCA?

- It is recommended to use a fairly fine network of nodes: 40 axial meshes.
- It is recommended to represent two parallel channels: one channel represents the maximum power fuel assembly (hot assembly) while the other represents all the other assemblies (mean assembly or mean core). Two fuel rods must be associated with the hot assembly, the maximum power rod and all the other rods of the hot assembly. Only one mean fuel rod may be associated with the mean core.
- During the blowdown phase and the refill phase of the transient, the lack of cross flows is probably slightly conservative for the hot rod.
- During the reflooding phase of the transient, the lack of cross flows would be too conservative for the hot rod given that, in these low flow conditions, the cross flows create good mixing in the rewetted zone. It is therefore recommended to move the hot rod from the hot assembly to the mean core. This calculation procedure was qualified on the PERICLES 2-D test.

27.2 How to model core thermal-hydraulics for a Small Break LOCA?

In SBLOCAs core uncovering situations may occur for low velocity conditions. Perfect mixing below the swell level was observed in PERICLES 2-D tests for these conditions. However, in the dry zone above the swell level, low or zero cross flows were observed. Thus, a single average channel model can correctly represent the region below the level but neither the single average channel nor any multi-parallel channel model can correctly represent the dry region. A single average channel model with two fuel rods (the mean rod and the hot rod) is the best compromise but probably underestimates the clad overheating in the dry region during core uncovering situations.

To achieve a relatively good prediction of the conditions occurring during such an accident, about 10 axial meshes (about 0.40 m per mesh) are recommended.

28. HOW TO MODEL GUIDE TUBES?

REFERENCE:	**-Physical improvement of the 3D module of CATHARE 2 V1.5B: SMTH/LMDS/EM/2002-084** **-CATHARE 2 V2.5_1 - Description of the Volume module: SSTH/LDAS/EM/2005-037** **-CATHARE 2 V2.5 - Description of the 3D module: SSTH/LDAS/EM/2004-048**
KEY WORDS	**THREED, VESSEL, AXIAL**
See also	***HOW TO MODEL the 3D vessel?*** ***WHAT is an AXIAL ELEMENT?*** ***WHAT is a VOLUME?***

28.1 AXIAL and VOLUME modelling of the pressure vessel:

Guide tubes are schematically represented either with 1 AXIAL element or with 1 AXIAL element for the upper part and a VOLUME element for the lower part. Lateral slots connecting the guide tube with the upper plenum can be modelled in the second case with horizontal junctions.

28.2 3D modelling of the pressure vessel

Flow communication between the upper head and the upper plenum exists through control rod guide tubes. The guide tubes may be represented by means of open faces between the upper head and the upper plenum, and vertical closed faces distributed in the upper plenum.

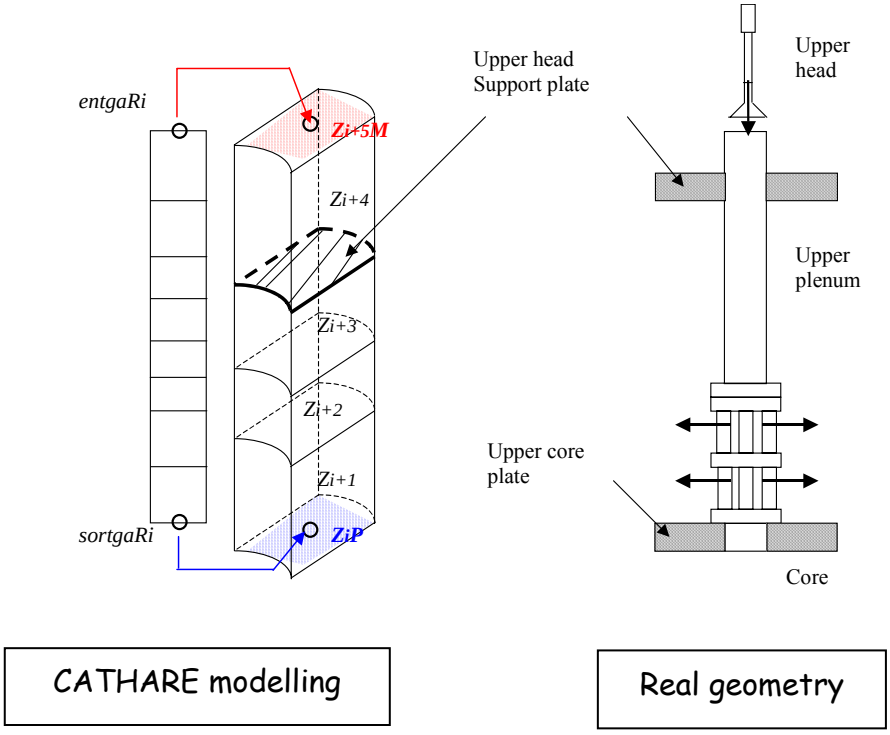
In situations where better modelling of the geometry of guide tubes is required (e.g., for a large break LOCA calculation), the 1-D flow in the guide tubes can be modelled by means of axial elements connected to the 3D module via internal junctions, as described below.

28.2.1 Modelling features for the upper plenum

A global volumic porosity can be calculated without taking into account the fluid volume of the guide tubes. This volumic porosity can be applied homogeneously to all the meshes.

28.2.2 Modelling features of the guide tubes

The guide tubes can be represented by axial elements, this being a simplified representation considering an average fluid section, hydraulic diameter, etc.



28.2.3 Layout of the guide tubes in the 3D vessel

- a homogeneous distribution of guide tubes is considered
- the weight of each axial element is determined in proportion to the mesh cell area.
- total number of guide tubes:
  - 53 for a 900 MW PWR
  - 54 in the input deck

⇒

3D ring	Number of guide tubes per 3D mesh
r1 (inner ring)	1
r2	2
r3	6

28.2.4 Connection of guide tubes to the 3D vessel

The guide tubes are connected by means of open internal junctions (SECT, PERI, HYDR, SINGULAR …)

29. HOW TO MODEL A LOWER PLENUM?

REFERENCE:	**-Physical improvement of the 3D module of CATHARE 2 V1.5B: SMTH/LMDS/EM/2002-084** **-CATHARE 2 V2.5_1 - Description of the Volume module: SSTH/LDAS/EM/2005-037** **-CATHARE 2 V2.5 - Description of the 3D module: SSTH/LDAS/EM/2004-048**
KEY WORDS	**VOLUME, LOWER PLENUM, THREED,**
See also	***HOW TO MODEL the 3D vessel?*** ***WHAT is a VOLUME?***

29.1 Volume modelling

The volume module is generally the best choice for modelling a lower plenum.

Some specific closure laws may be used with the optional key word **ANNULSPA** (GEOM directive) at the junction between the annular downcomer and the lower plenum. This makes it possible to calculate the Lower Plenum void ratio more exactly (during the blowdown and start of Refilling phases of a Large break LOCA, LBLOCA). This specific model was qualified on the PIERO tests.

However, the intrinsic limitations of this module must be borne in mind:

- It is a two-node model.
- Closure laws are very qualitative. They give good general trends but cannot be very accurate in each particular case.
- No thermal stratification is modelled with a two-node model. The use of an **AXIAL** module for the upper part of the lower plenum (above the downcomer connection) may be recommended, in particular for the reflooding phase of an LBLOCA when oscillations occur between core and downcomer. It is thus possible to track the interface between the hot water coming from the core (in the reverse flow phase of the oscillation) and the cold water coming from the ECCS through the downcomer. It also takes better account of the inertia of the fluid during oscillation which influences the oscillation period. The volume module does not model the momentum inertial term (time acceleration term).

29.2 3D modelling

When a THREED module is used to model the vessel, the lower plenum is included in the 3D element (see *HOW TO MODEL the 3D vessel?*)

30. HOW TO MODEL AN UPPER PLENUM?

REFERENCE:	**-Physical improvement of the 3D module of CATHARE 2 V1.5B: SMTH/LMDS/EM/2002-084** **-CATHARE 2 V2.5_1 - Description of the Volume module: SSTH/LDAS/EM/2005-037** **-CATHARE 2 V2.5 - Description of the 3D module: SSTH/LDAS/EM/2004-048**
KEY WORDS	**VOLUME, UPPER PLENUM, THREED,**
See also	***HOW TO MODEL the 3D vessel?*** ***WHAT is a VOLUME?***

30.1 Volume modelling

The volume module is generally the best choice for modelling an upper plenum when no 3D effects are expected.

Some specific closure laws may be used with the optional key word **UPPLEN** (GEOM Directive). However, the intrinsic limitations of this module must be borne in mind:

- It is a two-node model.
- Closure laws are very qualitative. They give good general trends but cannot be very accurate in each particular case.
- Complex phenomena such as entrainment / de-entrainment during a reflooding phase of a Large Break LOCA cannot be modelled with high accuracy.
- In case of hot leg or upper plenum ECCS injection, condensation phenomena may not be modelled accurately and there is no qualification for such situations.
- No thermal stratification is modelled with a two-node model. The use of an axial module for the lower part (below the hot leg connections) or the upper part (above the hot leg connections) of the upper plenum may be better if stratification plays an important role.
- A CCFL option may be necessary at the Core-Upper Plenum junction.

30.2 3D modelling

When a THREED module is used to model the vessel, the upper plenum is included in the 3D element (see *HOW TO MODEL the 3D vessel?*)

31. HOW TO MODEL A DOWNCOMER?

REFERENCE:	**-Physical improvement of the 3D module of CATHARE 2 V1.5B: SMTH/LMDS/EM/2002-084** **-CATHARE 2 V2.5_1 - Description of the Volume module: SSTH/LDAS/EM/2005-037** **-CATHARE 2 V2.5 - Description of the 3D module: SSTH/LDAS/EM/2004-048**
KEY WORDS	**DOWNCOMER, ANNULAR, AXIAL**
See also	***HOW TO MODEL the 3D vessel?*** ***WHAT is an AXIAL ELEMENT?***

31.1 AXIAL and VOLUME element modelling:

A volume element represents the upper part of the downcomer and connects all the cold legs. Below this volume, an axial element with annular hydraulics represents the rest of the downcomer. It is preferable to choose the separation between the volume and the 1-D element so that the cold legs are not at the bottom of the volume. This simple modelling procedure can be adopted in the following cases:

- when you represent a system test facility with an external downcomer (BETHSY, PKL,..)
- when the transient is not assumed to induce important azimuthal heterogeneities. This seems to be the case in many small break LOCAs. However, the interfacial friction model for annuli (HYDR "ANNULAR") was developed using the CREARE small scale (1/15) downcomer refill tests. In such cases, the phase separation phenomena associated with azimuthal heterogeneities are taken into account to a certain extent.

31.2 3D modelling: (see *HOW TO MODEL the 3D VESSEL?*)

The use of 3D modelling is preferable for LBLOCA. The refill phase is characterised by counter-current flow and direct contact condensation with azimuthal heterogeneities which cannot be represented in detail with a 1-D element.

32. HOW TO MODEL A PRESSURISER?

REFERENCE:	**-CATHARE 2 V2.5_1 - Description of the Volume module: SSTH/LDAS/EM/2005-037**
KEY WORDS	**VOLUME, PRESSURISER, AXIAL, SPRAY**
See also	**HOW TO MODEL a MANWAY?**

In many LOCAs not compensated for by safety injections, the pressuriser simply empties and then has no effect on primary pressure. For all these cases the volume module is appropriate.

In case of actuation of sprays or heaters, the **volume** module may still be correct if there is no temperature stratification below the mixture level.

For a transient with a break or open relief valve at the top of the pressuriser or when there is temperature stratification below the level, the use of an **axial** vertical element is better. The user has to impose the initial mixture level during the steady state phase.

32.1 Spray

The spray may be represented either by a source term, a boundary condition connected to a leg defined at the top of the volume or an AXIAL element connected from the cold leg to the top of the volume element.

Some qualification calculations of tests relative to PWR pressurisers and to the BETHSY pressuriser have shown certain shortcomings of the modelling procedure:

- Spray efficiency is sensitive to the velocity of the spray.
- By using the **PWRPRZ** option it is possible to predict a realistic heat exchange between the spray water and the walls when the spray flow does not touch the walls when dropping.
- The effects due to possible temperature stratification of the water in the pressuriser are not represented by the volume module. They can be better represented using an axial module for the pressuriser.

32.2 Heaters

The heaters can be represented either using a heated wall or a CANDLE operator (see *HOW TO MODEL a HEAT SOURCE?*). The effects due to a possible temperature stratification of the water in the pressuriser are not represented by the volume module. They can be better represented using an axial module for the pressuriser.

In case of heated wall modelling, the **PWRPRZ** option can be used to predict correctly the flashing delay due to the heaters and the vapour-wall exchange when the pressure is increasing in the case of film condensation of the steam above the level.

32.3 Initialisation

An initial level can be given with the key word LEVEL just before the GOPERM action.

33. HOW TO MODEL A CONTAINMENT?

REFERENCE:	**-CATHARE 2 V2.5_1 - Description of the Volume module: SSTH/LDAS/EM/2005-037**
KEY WORD	**VOLUME**
See also	***HOW TO MODEL a SINK?*** ***HOW TO MODEL a SOURCE?***

The reactor containment may be modelled by the CATHARE code by using a VOLUME module or a set of VOLUME modules.

Special models have been developed to take into account containment components (circuits, exchangers related to the safeguard system), a break between the primary coolant system and the containment, or specific physical phenomena occurring in the containment (film condensation on the containment wall).

33.1 Model to represent the safeguard system

The model involves three (or four) circuits:

- the containment spray system circuit (called "EAS" in French reactors),
- possibly the safety injection system circuit (called "RIS" in French reactors),
- the component cooling system circuit (called "RRI" in French reactors),
- the essential service water system circuit (called "SEC" in French reactors),

and two (or three) heat exchangers:

- the "EAS/RRI" heat exchanger, between the containment spray system and the component cooling system,
- possibly the "ISBP/RRI" exchanger, between the low pressure safety injection system and the component cooling system,
- the "RRI/SEC" exchanger, between the component cooling system and the essential service water system.

In direct running conditions, both safety circuits ("EAS" and "RIS") are fed with water from the "Reactor Cavity and Spent Fuel Pit Cooling and Treatment System" tank (called "PTR" tank in French reactors). When the "PTR" tank is empty, the operation is switched to recycling running conditions: pumps of the safety circuits ("EAS" and "RIS") draw out water which has accumulated at the bottom of the containment in the sinks. In recycling conditions, the spray water (case of French reactors of CPY, P4 or N4 type) or the safety injection system water (case of EPR reactor project) from the sinks is cooled by the intermediary circuit ("RRI"), in an exchanger ("EAS/RRI" or "ISPB/RRI"). The "RRI" circuit is itself cooled by the Essential Service Water System through the exchanger "RRI/SEC". Drawn-out mass flowrate and exchanged power with auxiliary circuits are also taken into account by the model. Moreover, the "EAS/RRI", "ISBP/RRI" and "RRI/SEC" exchangers can be co-current or counter-current (DIRECT or REVERSE key word).

As a normal sink, the SINKRRI SUB-MODULE (see *HOW TO MODEL a SINK?*), held by a VOLUME element located at the bottom of the containment, enables two-phase fluid to be removed from the mesh holding the SINKRRI. Moreover, it computes the flowrates and fluid temperatures of safeguard systems.

33.2 Model to represent a break between the PCS and the containment

To perform a CATHARE calculation of the containment behaviour in case of an accidental transient involving a break flowrate from the primary circuit into the containment, the input data to be defined are the flowrates and the enthalpies of the two phases at the break.

There are different ways to determine these data:

- by performing a CATHARE calculation of the transient in the primary circuit; this calculation may be coupled to the containment calculation, or may have been performed previously. In both cases, the calculation of the primary circuit provides the break flowrates and enthalpies at the coolant circuit conditions. It is then necessary to determine these flowrates and enthalpies (and the way they are broken down according to the two phases) at the containment conditions. This may be performed using phase distribution models of the SOURCE sub-module (see "PFM" or "VFM" models in the dictionary).
- without performing any CATHARE calculation of the primary circuit, the SOURIS sub-module can be used, based on a simple model (the "RIS overflowing" model) relating the flow characteristics at the break to the safety injection conditions and to the core residual power. A SOURIS sub-module must be defined on the volume located at the top of the containment. It is a normal source with includes, in addition, a description of the "RIS" model (see *HOW TO MODEL a SOURCE?*).

33.3 Model to calculate film condensation on the containment wall

Film condensation on the containment walls in the presence of non-condensable gas is modelled in the VOLUME module of the CATHARE code.

The standard correlation used for the VOLUME module is the CHEN correlation.

NUSSELT, COPAIN or USHIDA film condensation correlations can be used with the VFILM directive. The use of the COPAIN film condensation correlation is adviced for the containment wall to fluid heat transfer.

The convection velocity used in the correlations can be imposed in the input deck or calculated by the code. The VFILM directive must be used in the data BLOCK of the input deck. The film condensation calculation is activated in the executable block using the STARTVC directive and can be cancelled with the STOPVC directive.

34. HOW TO MODEL CORE NEUTRONICS?

REFERENCE:	**-CATHARE 2 V2.5_1 - Description of the core kinetics: SSTH/LDAS/EM/2005-042**
KEY WORDS	**FUEL, NEUTRONICS, CORE**
See also	***HOW TO MODEL FUEL RODS?*** ***HOW TO INITIALISE COLD/HOT FUEL DATA?*** ***WHY/WHEN TO USE THE STAND ALONE FUEL?***

34.1 Point Reactor Kinetics Equations

The main idea of the point kinetics is then to split the neutron flux in two factors $\Phi(\vec{r},t) = T(t)\,S(\vec{r})$: the amplitude function $T(t)$ depending only on time (representing neutron population change in reactor), and the shape function $S(\vec{r})$ which is supposed to depend only on space (it is equal to the shape of the neutron flux at the initial state of the reactor before perturbations are applied).

With this assumption, we get a set of equations able to predict the evolution of the amplitude function as a function of reactivity and delayed neutrons groups parameters and concentration. The evolution of the ***instantaneous fission power*** P_{fis} (directly proportional to the fission rate, and so to the neutron flux $\Phi(\vec{r},t)$), which is the parameter of interest in CATHARE, is then given by the following set of equations called the ***Point Reactor Kinetics Equations*** :

$$\begin{cases} \dfrac{dP_{fis}}{dt} = \dfrac{\beta}{\Lambda}\left(\rho(t)-1\right)P_{fis} + \displaystyle\sum_{i=1}^{N_D}\lambda_i C_i \;+ S_{ext}(t) \\ \dfrac{dC_i}{dt} = \dfrac{\beta_i}{\Lambda}P_{fis} - \lambda_i C_i \quad , \quad i = 1..N_D \end{cases}$$

One can notice that the instantaneous fission power P_{fis} depends very strongly on the reactivity which is supposed to be time dependant. S_{ext} represents the time dependant power increase source term (in $W.s^{-1}$) due to external neutron source. Default value for S_{ext} is zero, but user can set another value by using WRITE directive (with SOURCNEU keyword).

Constant parameters

$N_D = 6$	:	Number of delayed-neutron groups
Λ	:	Prompt-neutron generation time (s)
f_i	:	Relative effective delayed-neutron fraction of group $i = 1..N_D$
$\beta_i = f_i\beta$	:	Effective delayed-neutron fraction of group $i = 1..N_D$
$\beta = \sum_{i=1}^{N_d}\beta_i$	:	Total effective delayed-neutron fraction
λ_i	:	Decay constant for delayed-neutron precursors of group $i = 1..N_D$ (s^{-1})

34.2 Actinide and Fission Product Decay-Heat Equations

The solution of Point Reactor Kinetics Model for the instantaneous fission power is used to evaluate the decay-heat. Actinides and fission products decay power P_{res} (called also residual power when no fission occurs) is given by the following ***Decay-Heat Equations*** :

$$\begin{cases} P_{res} = \sum_{j=1}^{N_H} \lambda_j^H H_j \\ \dfrac{dH_j}{dt} = -\lambda_j^H H_j + E_j P_{fis} \quad , \quad j = 1..N_H \end{cases}$$

Constant parameters

$N_H = 11$: Number of decay-heat groups

λ_j^H : Decay constant of decay-heat group $j = 1..N_H$ (s^{-1})

E_j : Effective energy fraction of decay-heat group $j = 1..N_H$

34.3 PWR Feedback anti-reactivity

The core reactivity is the sum of the external reactivity (control-rod movements) and the reactivity based on changes in the core fuel temperature (Doppler effect), coolant density (moderator effect) and boron concentration.

$$\delta_{fdbk}\rho(t) = \delta\rho_{dop}(t) + \delta\rho_{\mathrm{mod}}(t) + \delta\rho_{bor}(t) \qquad \text{with} \qquad \delta\rho_i(t) = \rho_i(t) - \rho_i(t_0)$$

$\delta\rho_{dop}(t)$: Doppler feedback anti-reactivity (due to changes in fuel temperature)

$\delta\rho_{\mathrm{mod}}(t)$: Moderator feedback anti-reactivity or void effect (due to changes in coolant density)

$\delta\rho_{bor}(t)$: Boron feedback anti-reactivity (due to changes in soluble boron concentration)

34.4 HTR Feedback anti-reactivity

For a HTR application the way to compute the global feedback reactivity is different from the standard PWR Model. The core reactivity is the sum of the external reactivity (control-rod movements activated by the ROROD directive) and of the reactivity based on changes in the core fuel temperature (Doppler effect), in the graphite Moderator temperature (moderator effect), in the wall structures temperature (Reflector effect) but also on changes of the coolant density (void effect).

The global feedback anti-reactivity is given by :

$$\delta_{fdbk}\rho(t) = \delta\rho_{dop-T}(t) + \delta\rho_{\mathrm{mod}-T}(t) + \delta\rho_{\mathrm{mod}-\alpha}(t) + \delta\rho_{ref-T}(t) \quad \text{with} \quad \delta\rho_i(t) = \rho_i(t) - \rho_i(t_0)$$

$\delta\rho_{dop-T}(t)$: Doppler feedback anti-reactivity (due to changes in fuel temperature)

$\delta\rho_{\mathrm{mod}-T}(t)$: Moderator feedback anti-reactivity (due to changes in moderator temperature)

$\delta\rho_{\mathrm{mod}-\alpha}(t)$: Moderator feedback anti-reactivity or void effect (due to changes in coolant density)

$\delta\rho_{ref-T}(t)$: Reflector feedback anti-reactivity (due to changes in reflector temperature

34.5 External reactivity

The external reactivity due to an emergency shutdown can be imposed by using the SCRAM directive. The evolution of external reactivity is made by time interpolation of given values in the input deck (t_0 is time when Point kinetics model is activated, using GONEUT directive.

$$\boxed{\rho_{ext}(w) = law(t - t_0)}$$

The external reactivity due to the control rods movements may be also modified at any time during calculation by using ROROD directive.

34.6 Power Generated in the Overall Core

The following powers are defined for the overall core :

- The neutronic or nominal power (part of the instantaneous fission power released at time of fission) :

$$\boxed{P_{nom} = \left(1 - \sum_{j=1}^{N_H} E_j\right) P_{fis}}$$

- The residual power (or actinides and fission products radioactive decay power) :

$$\boxed{P_{res} = \sum_{j=1}^{N_H} \lambda_j^H H_j}$$

- The effective power (or total power generated in the core) : $\boxed{P_{eff} = P_{nom} + P_{res}}$
- The delayed power (part of instantaneous fission power given to the actinides and fission products and that will be released by radioactive decay) :

$$\boxed{P_{del} = \left(\sum_{j=1}^{N_H} E_j\right) P_{fis}}$$

Remark : At equilibrium, $P_{res} = P_{del}$ and $\boxed{P_{eff} = P_{fis}}$

35. HOW TO MODEL A STEAM GENERATOR?

REFERENCE:	**-** **-CATHARE 2 V2.5 - Description of the wall, heat exchanger and 0D steam generator sub-modules: SSTH/LDAS/EM/2004-049**
KEY WORDS	**HEAT EXCHANGER, STEAM GENERATOR**
See also	***WHAT is a ONE-NODE STEAM GENERATOR?***

Two ways are proposed to model steam generators:

35.1 One node steam generator:

This module is adapted to the transients where exchanges in the SG are not determinant (very fast depressurisations, etc.).

It is a simplified representation of a secondary circuit which exchanges with a primary axial element (see *WHAT is a ONE-NODE STEAM GENERATOR?*).

The secondary circuit is characterised by three variables: the liquid mass, the steam mass and the pressure in the cavity. A saturated homogeneous mixture is assumed. Normal feedwater, auxiliary feedwater, steam outlet flowrates are modelled. Steam generator isolation and valves are also modelled. See the SGFEED directive.

But the one node steam generator module is very limited in the description of the secondary side:

- no sub-cooled liquid is considered,
- no tube dry out can be described,
- no stratification, no re-circulation,
- no connection between several steam generators.

35.1.1 Steady state initialisation:

The power to exchange is initialised in the operator SGCARACT (key word SGP).

The initial values of the secondary pressure of the thermal resistance are initialised in the operator SGCARACT. In the same operator the user chooses the parameter to modify in order to find the power to exchange: these parameters are the secondary exchange coefficient, the thermal resistance and the secondary pressure.

35.1.2 Transient:

The one node steam generator module is explicitly coupled to the circuit, which means that it is calculated at the end of each time step after convergence of all the principal variables. Thus it is recommended to avoid excessively large time steps.

35.2 Full representation of the secondary circuit:

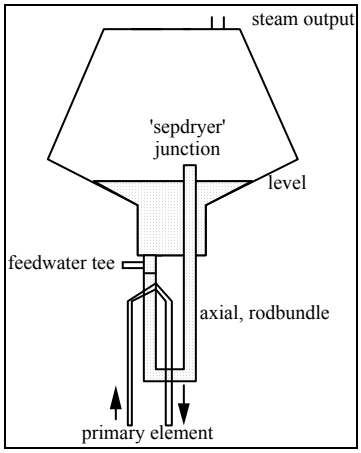
This representation is generally recommended. A heat exchanger is described between an axial element of the primary circuit and one axial element of the secondary circuit. Geometrical limitations for the mesh are described in the dictionary. For this nodalisation, the user will choose the second type of initialisation (operator PERMINIT followed by regulations)

Here we can distinguish between PWR SG and VVER SG models.

35.2.1 PWR type steam generator:

A normal representation for a PWR 900 MW reactor is:

- A volume module representing the upper cavity or steam dome. A SEPDRYER junction at the connection with the riser is used to represent the separator.
- A pipe module representing the downcomer and the riser.
- Feed-water injection, which may be represented either by a tee or by a source.



The Main Feed Water (MFW) overflow phenomena in a N4 reactor steam generator (SG) can be modelled with a PIQARE object (see *HOW TO MODEL a SOURCE*).

All the secondary elements are calculated with initialisation values: feedwater values and power to exchange for each exchanger (operator PERMINIT). The primary circuit can then be calculated.

The user then describes the regulations for the primary circuit and the heat exchanger.

- the level in the cavity is regulated with a ‘source’ at the steam generator temperature. This source is set at the bottom of the cavity.
- the equality of the flowrates is assumed explicitly by the user at each time step by modifying the feedwater flowrate.
- a regulation on fouling (thermal resistance) may be used.

35.2.2 VVER type horizontal steam generator

- Volume modules are required for the SG channel headers.
- SG tubes: several layers of tubes must be represented to describe:
  - Heat exchange degradation when the secondary level uncovers the upper layers of tubes
  - Heat exchange degradation when the primary level in the hot collector is below the upper layers of tubes
- Re-circulating flows between upper layers and lower layers of tubes in natural circulation conditions
- Secondary side of the SGs:
  - A volume module is convenient for the space above the tube bundle. A SEPDRYER junction may be required at the connection with the module representing the tube bundle.
  - The simplest representation of the tube bundle uses a single vertical tube with a number of meshes equal to the number of groups of tubes. In this case, the homogenising effects of internal re-circulation may be simply represented by

distributing the feedwater in every mesh. This may be recommended for transients without major temperature stratification. The modelling choices (such as feedwater flow distribution) must be qualified on plant data.

- In another representation, the tube bundle itself uses a single vertical tube with a number of meshes equal to the number of groups of tubes. The re-circulation is represented by a pipe connected to the volume at the top and bottom of the tube bundle. Feedwater is injected in the re-circulation pipe. The re-circulation pipe may be replaced by a Volume element. In this case, the internal re-circulation is represented as long as the tube bundle is not uncovered. No cross flows between the re-circulation pipe and each mesh of the tube bundle can be modelled.
- A more complex representation uses several TEE modules to represent cross flows between the downcomer and the tube bundle. In this case, the internal re-circulation is represented even if the tube bundle is partly uncovered.

Recommendations:

- The simplest representation without a re-circulation loop may be used for many transients where no temperature stratification is expected.

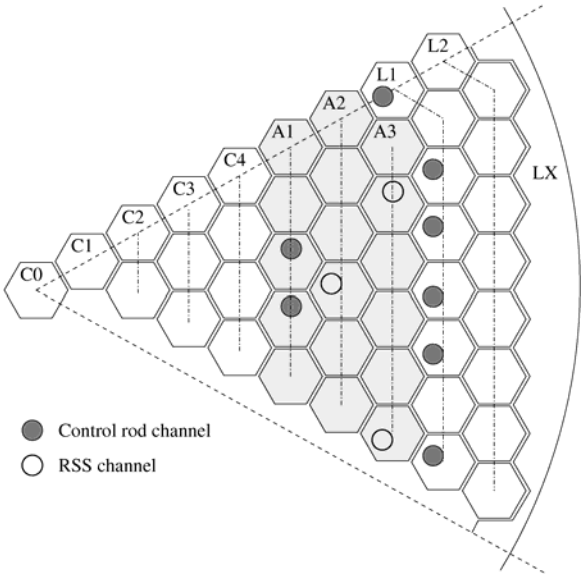
36. HOW TO MODEL A GCR CORE?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019**
KEY WORDS	**GAS COOLED REACTOR, HEAT LOSS, COMPRESSOR, TURBINE, TURBO-MACHINE**
See also	*HOW TO MODEL HEAT LOSSES?* *HOW TO MODEL a TURBINE?* *HOW TO MODEL a COMPRESSOR?* *HOW TO MODEL a TURBO-MACHINE?*

The choice of coolant apart, a major difference between a conventional PWR and gas-cooled reactors is the thermal and neutronic properties of the reactor core. A 600 MW(t) VHTR core, for example, contains more than 800 metric tons of prismatic graphite blocks, some charged with fuel, others acting as reflectors. The large mass of graphite represents a considerable thermal inertia. From a simulation point-of-view, we have to focus on solid heat transfer modelling. The solid heat transfer in the core plays a crucial role in the design of the VHTR core, since the residual heat should be evacuated via the walls of the core vessel after an emergency shutdown. The heat transfer properties also determine the maximal temperatures experienced by fuel pellets and vessel walls in normal operation and during transients. During neutronic transients the temperature distribution in the core is of great importance, since fuel pellets, graphite moderator and reflector blocks all contribute to the core reactivity via thermal anti-reactivities.

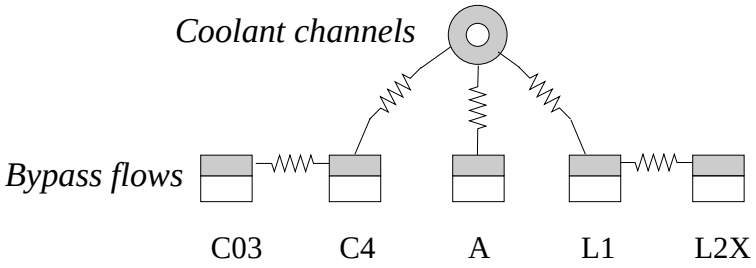
36.1 Core heat transfer modelling in CATHARE:

A typical graphite GCR core, HTGR core for example, is made up of a large number of hexagonal graphite blocks organized in concentric hexagonal rings. Some of these blocks have channels for the insertion of control rods. The blocks in the active annular part of the core have a large number of coolant channels, each of which is surrounded by channels filled with fuel pellets. The graphite blocks inside and outside of the active core constitute the central and lateral reflectors, respectively. There is a small gap of a couple of millimetres between the block columns, to allow for dimensional changes and refuelling.

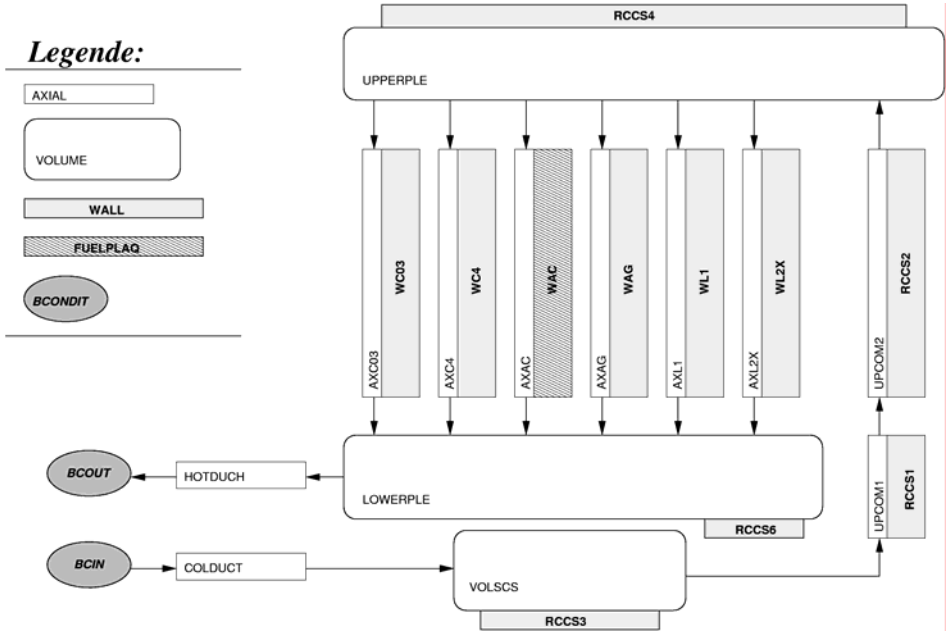


One 6th of a HTGR core in horizontal cross-section. The grey region represents the three concentric rings (A) of the active core. Inside is the central reflector (C), and outside the lateral reflector (L). Some blocks in the active core and in the lateral reflector have channels for control rods or for the reserve shutdown system (RSS).

It can be modelled as following with five radial zones: one for the active core (A) and two zones each for the central reflector (C03, C4) and the lateral reflector (L1, L2X). The coolant channel flow in the active core is modelled separately, while the flow in rod channels and block gaps has been combined in one flow path per zone. Thermal resistances indicate the thermal coupling of walls:



And corresponding to the following scheme:



A successful description of the flow and heat transfer in a graphite GCR core should address the following aspects:

1. Modelling of the different coolant flow paths, and the heat exchange between the fluid and surrounding solid structures (walls, inserts, etc.) (see *HOW TO MODEL HEAT LOSSES?*).
2. Modelling of *radial* heat transfer between blocks of the core, taking into account the presence of block gaps, coolant channels, etc. (see **EXWALINK** operator and *HOW TO MODEL WALLS?*).
3. Simple modelling of *axial* heat transfer in the block of the core (see **AXICOND** directive and *HOW TO MODEL WALLS?*)
4. Modelling of heat exchange with the exterior (for example convection/radiation heat transfer from the outer surface of the core vessel to the containment cavity) (see **RADEXT** keyword for **WALL** and **FUELPLAQ**).

36.2 Modelling of solids and walls in gas-cooled reactor cores: FUELPLAQ

All walls in a GCR core should be defined using the **FUELPLAQ** directive with the keyword **HTR**, if they should be able to (i) receive fission power, and/or (ii) contribute a thermal anti-reactivity to the core neutronics. A wall can consist of several segments with different properties. In the example, the first and the third segments act as reflectors (**REFLECT** keyword). The second segment consists of three material layers: the first acts as moderator (**MODERAT** keyword), while the third layer represents the fuel (**DOPPLER** keyword).

36.3 Point neutron kinetics:

The **FUELPLAQ** HTR walls should be used together with a **CORE** object, also with the keyword **HTR**. This module handles the point neutronics modelling in Cathare, and distributes the core neutronic and residual power to the walls in the model.

CATHARE already features a point neutron kinetics module, with six delayed neutron groups and eleven heat decay groups (for the residual heat). This module has been modified, using **HTR** option, to support the modeling of gas-cooled reactors. In particular, it should be possible to distribute the core power on several walls in the core, walls that may not have the same geometry. Each such wall structure can have its own set of temperature-dependant thermal anti-reactivity coefficients, contributing to the total core reactivity. Walls can have several axial segments and several radial material layers. The wall of a coolant channel, for example, can have an upper and a lower segment of graphite acting as passive reflectors, and a middle segment containing the fuel. The fuelled segment can have one radial layer acting as moderator, and another layer representing the actual fuel pellet with a Doppler-type anti-reactivity contribution.

The point kinetics module collects anti-reactivities from all elements of the core to compute a global core reactivity. An additional external reactivity can be introduced in the input deck, to model the action of the control rods, or effects of Xenon poisoning.

A reactor scram can be initiated using the **SCRAM** directive, which gives the control rod reactivity as a function of time.

The new directive **ROROD** can be used in the input deck during a transient to define and to modfiy an external reactivity. The intended use is the implementation of a simple control system for the insertion/extraction of the control rods for power control and load following.

37. HOW TO MODEL FUEL RODS?

REFERENCE:	**-CATHARE 2 V2.5_1 - Description of the fuel sub-module SSTH/LDAS/EM/2005-043**
KEY WORDS	**FUEL, NEUTRONICS,**
See also	***HOW TO INITIALISE COLD/HOT FUEL DATA?*** ***WHY/WHEN TO USE THE STAND ALONE FUEL?***

See the precise description in the User Manual, which includes an extensive description of the way to change the power law versus time of fuel rods or their nominal power.

You need to model fuel rods with the fuel sub-module as soon as thermo-mechanical effects are sensitive in your computation. If your transient is such that there will be no clad deformation and no clad oxidation, it is sufficient to use the multi-layer wall approach (WALL, WALL3D).

38. HOW TO MODEL THE FUEL BALLOONING?

REFERENCE:	**-CATHARE 2 V2.5_1 - Description of the fuel sub-module SSTH/LDAS/EM/2005-043** **-CATHARE 2 V2.5_1 : Description of the base revision 6.1 physical laws used in the 1D, 0D and 3D modules : SMTH/LMDS/EM/2005-038**
KEY WORDS	**FUEL, NEUTRONICS,**
See also	*HOW TO MODEL FUEL RODS?*

In the CATHARE code, fuel ballooning is taken into account only for local cladding thermo-mechanical behavior, without any induced change in core geometry that remains unaffected by ballooning.

Nevertheless, some physical laws are modified to take into account the effect of ballooning on fluid distribution around the balloon. Mechanical effect of the presence of the ballooning is modeled by adding pressure loss coefficient in the vector points close to location of balloon.

Steam to liquid and wall to fluid heat transfers are also modified to take into account the fluid acceleration, the liquid droplet break up and vaporisation due to the blockage.

The model is activated when the clad rupture is calculated both during the depressurisation phase and the reflooding phase of a large break LOCA. This model may be used when the core is modelled with a 3D module and when the core rod are represented using either simple walls or fuel sub modules.

The modifications of the physical laws are based on the deformation of the fuel rod or of a considered wall :

- In case of fuel rod, the clad deformation is calculated by the fuel sub module.
- If the model is used fo simple walls, the user has to input in the data deck the cladding deformation.

In both fuel or wall cases, users have to input in data deck (GOBALLON directive) the law defining the blockage rate in function of the cladding deformation.

Then, the added pressure loss (a modified GAMBILL law), the modified heat transfer coefficients and the broken up liquid droplets are calculated.

All the modified physical laws are given in “-CATHARE 2 V2.5_1 : Description of the base revision 6.1 physical laws used in the 1D, 0D and 3D modules : SMTH/LMDS/EM/2005-038”.

The fuel ballooning model is validated for the 3D module using the CEGB (air or steam single phase or droplet flows) and the ACHILLES, FLECHT SEASET, FEBA and SEFLEX (reflooding conditions) facility tests.

39. HOW TO MODEL WALLS?

REFERENCE:	**-CATHARE 2 V2.5 - Description of the wall, heat exchanger and 0D steam generator sub-modules: SSTH/LDAS/EM/2004-049** **-CATHARE 2 V2.5_1 - Description of the fuel sub-module SSTH/LDAS/EM/2005-043** **-CATHARE 2 V2.5_1 - Description of the core kinetics: SSTH/LDAS/EM/2005-042**
KEY WORDS	**EXCHANGER, FUEL, HEAT LOSS,HEAT SOURCE, NEUTRONICS, WALL**
See also	***HOW TO MODEL a HEAT EXCHANGER?*** ***WHAT ARE THERMAL STRUCTURES?*** ***HOW TO MODEL a HEAT SOURCE?*** ***HOW TO MODEL FACILITY HEAT LOSSES?*** ***HOW TO MODEL a HEAT SINK?***

In CATHARE, walls are cylindrical or plane.

39.1 Heating perimeter

In AXIAL elements, the heat exchange area is calculated as the product of the heating perimeter and the mesh length and only the heating perimeter is entered by the user. In VOLUME elements, the heat exchange area is calculated as the product of the heating perimeter and the wall height and only the heating perimeter is entered by the user. In THREED elements, the heat exchange area is directly entered by the user.

In an AXIAL element or a THREED element, the friction perimeter and the heating perimeter may be different.

39.2 Advice

Use a finer nodalisation on the heat exchange side of the wall than on the external side.

Caution: for external wall layers, the ISO iso-volume option defines finer meshes on the opposite side.

When power is generated in the wall, calculate the volume of the power generation layer and then deduce and enter the corresponding volumetric power.

The heat losses may be entered. They can be constant and defined either by the heat flux or by the heat flux coefficient and the external temperature. If they change with time, a FLUMOD directive must be used (See *HOW TO MODEL FACILITY HEAT LOSSES?*).

39.3 Coupling of walls

Two walls connected to an axial of a same circuit or of two different circuits (WALL or FUELPLAQ sub modules) can be coupled used the **EXWALINK** operator (**EX**plicit **WAll to wall LINK).**

The two walls are coupled thermally through this link, and they can exchange heat depending on the wall temperatures in the two walls. The communication is handled automatically by the code.

The heat exchanged can be computed using :

- a thermal resistance model (conduction),
- a simplified radiation model,
- a combination of both.

For the radiative heat transfer, it is assumed that:

- the linked walls may be considered of infinite length with respect to their distance;
- the walls are close enough to consider that the flux lost by one wall is equal to the flux received by the other. The model is not able to predict the fluxes exchanged between a small wall inside another one,
- the walls are “grey walls” so their respective emissivities do not depend on their temperatures.

39.4 Simple model of axial conduction of walls

A simple model of axial conduction in a wall is implemented in the code. It is used for GCR calculation to model the axial conduction in the core rod. The axial conduction calculation resolution is explicit. It is defined in the WALL directive but has to be enabled using the AXICOND directive in the command block.

40. HOW TO MODEL A HEAT EXCHANGER?

REFERENCE:	**-CATHARE 2 V2.5 - Description of the wall, heat exchanger and 0D steam generator sub-modules: SSTH/LDAS/EM/2004-049**
KEY WORDS	**HEAT EXCHANGER**
See also	

A heat exchanger is a thermal coupling between two elements representing the heat exchange between a primary side and a secondary side through a wall.

The primary side normally refers to the “Hot” side and the secondary side refers to the “Cold” side, for instance for the modelling of PWR steam generators.

40.1 If the two hydraulic sides of the exchanger are modelled by CATHARE:

The flux computation between each hydraulic side and the wall representing the heat exchanger is fully implicit.

Two types of exchanger are proposed: one with **a quasi implicit** coupling between the primary and the secondary sides of the wall and the other with **an explicit** coupling between the primary and the secondary sides of the wall.

40.1.1 Explicit exchanger type (default)

In this case, the two sides of the heat exchanger must belong to two different circuits (CIRCUIT operator).

An exchanger is then described between an axial element of the primary circuit and an axial element of the secondary circuit. Geometrical limitations for nodalisation are described in the dictionary.

In high-speed transients when the exchange conditions vary rapidly from one time step to the next, abnormal effects, such as heating transfer from the cold side to the hot side, may be seen. These should be solved by using the IMPLICIT option.

40.1.2 Quasi implicit exchanger type (IMPLICIT key word in EXCHANGER)

In this case, the two sides of the heat exchanger may belong to two different circuits, to two elements of a same circuit and even to two parts of a same element.

The coupling of the two elements is only partially taken into account during the steady state calculation: users have to ensure that both elements are completely coupled by including a stabilised transient at the end of the steady state calculations.

40.2 If one of the two hydraulic sides of the exchanger is not modelled by CATHARE:

In such a case, the user has to define a simple WALL representing the heat exchanger. The heat exchange must then be defined by the LOSS key word in the WALL representing the heat exchanger, and modified by the FLUMOD directive. The heat exchange may be either an imposed flux (PHIEXT) or a computed flux, with an external fluid temperature and an exchange coefficient being given. The flux computation between each hydraulic side and the wall representing the heat exchanger is fully implicit.

This model is used, for example, to ensure that CATHARE is thermally coupled with another code (e.g., to model the RHR System heat exchanger).

41. HOW TO MODEL A SEPARATOR AND A DRYER?

REFERENCE:	**-CATHARE 2 V2.5_1 - Description of the Volume module: SSTH/LDAS/EM/2005-037**
KEY WORDS	**VOLUME,PWR (pressurised water reactor), BWR (boiling water reactor)**
See also	

Separators and dryers are present in PWR steam generators and BWR vessels to prevent liquid entrainment in the steam turbines. Precise modelling of these components is not possible given the complex geometry of the various possible designs.

However, if the cavity of the steam generator is modelled by a VOLUME module, options may be used to represent at least qualitatively the effects of these two components.

41.1 Case of PWR

The key word SEPDRYER must be entered in the GEOM directive for the junction between the riser and the cavity. The β distribution coefficients at the junction of the volume will be modified in the following way:

- if the level in the cavity is below the junction
  - all the steam from the riser is sent to the upper sub-volume
  - ε_1)% of the liquid from the riser is sent to the lower sub-volume and (1-ε_1) % to the upper sub-volume
- if the level in the cavity is above the junction
  - all the water from the riser is sent to the lower sub-volume
  - ε_2)% of the steam from the riser is sent to the upper sub-volume and (1-ε_2) % to the lower sub-volume

The standard values are: ε_1 = 4%, ε_2 = 0%. They may be changed in subroutine FTRANS (EPSEP, EPSEC).

Generally this model gives the following result:

- complete separation when the level is above the junction. Only steam goes to the steam lines.
- some liquid may be entrained (maximum ε_1 %) to the steam lines when the level is below the junction.

41.2 Case of BWR

In the case of a **BWR vessel separator-dryer junction**, specific options (**SEPINLET, SEPSIDE**, **SEPOUT, MINDRYER and DRYER)** imposing the carry over, the carry under of the separator and the liquid quality of the dryer have to be used. In this case the phase flow distribution at the junction takes into account the separator and dryer behaviour.

42. HOW TO MODEL A STEAM GENERATOR CHANNEL HEAD?

REFERENCE:	
KEY WORDS	**VOLUME, AXIAL, SG CHANNEL HEAD, VVER**
See also	

There are two possibilities for modelling a steam generator channel head:

- **A VOLUME module**
- **An AXIAL module**

The volume module is recommended for Large Break LOCAs

The axial module requires abrupt area changes and this may induce physical inconsistencies for high velocity conditions. So when high velocities are expected in two-phase situations (for instance V > 20m/s), a volume module is preferable.

The Hot Collector and Cold Collector of a **VVER** type SG must be modelled with a **Volume module** particularly when there are several groups of SG tubes.

43. HOW TO MODEL A TURBINE?

	0 –The CATHARE code: SMTH/LMDS/EM/2001-063
KEY WORDS	**TURBINE, BWR (boiling water reactor),GAS COOLED REACTOR, TURBO-MACHINE**
See also	***HOW TO MODEL a TURBO-MACHINE?*** ***HOW TO MODEL a COMPRESSOR?***

43.1 Case of boiling water reactor

CATHARE has a turbine sub-module which can be attached to a vector point of an axial module.

The user must enter the nozzle section, the head loss through the turbine and the power or efficiency.

- The two momentum equations of the vector point considered are replaced by two different equations:
  - Vl = Vg
  - and a total momentum equation where the velocity is a function of the upstream and downstream pressures.
- The energy equation (for the vapour phase) of the scalar point downstream of the turbine is modified to take into account the turbine power. The user gives either the turbine power, or the efficiency η, and can change from one option to the other during the calculation (directives MODPCST and MODPNCST in the command block).

Turbines with several stages can be represented by one or several sub-modules.

The part of the pipe where the turbine is located must be horizontal.

Liquid drains or steam extractions can be modelled with sinks. A special sink with the key word 'PURGE' models a drain of pure liquid from the scalar point downstream of the turbine.

43.2 Case of gas reactor

The turbine sub-module (TCOMCHAR directive) is used to model a gas turbine in a 0D approach. The user should enter the reduced characteristics of the turbine (the pressure ratio and the isentropic efficiency).

A turbine is located at a vector node. It affects the momentum and energy equations. In the calculated Source terms, the head (H) and torque (T) are derived from the specific turbine characteristics.

The momentum source term is assigned to the momentum equations written at this vector node and influences the pressure difference DP between the two neighbouring scalar nodes. The energy source term is assigned to the energy equations written at the scalar node which is downstream of the liquid velocity.

The turbine characteristics are:

- ✓ The reduced expansion ratio which is a function of the reduced rotation speed and the reduced flowrate.

$$\left(\frac{P_{in}}{P_{out}}\right)^{*} = f\left[\left(\frac{W\sqrt{T_{in}}}{P_{in}}\right)^{*}, \left(\frac{\omega}{\sqrt{T_{in}}}\right)^{*}\right]$$

- ✓ The reduced efficiency which is a function of the reduced rotation speed and the reduced flowrate.

$$\eta^* = g\left[\left(\frac{P_{in}}{P_{out}}\right)^*, \left(\frac{\omega}{\sqrt{T_{in}}}\right)^*\right]$$

The source terms H and T which will be used in the CATHARE equations are derived from these characteristics as follows:

$$\frac{H}{H_{ref}} \approx \frac{\left[\left(\frac{P_{T,out}}{P_{T,in}}\right)^*\left(\frac{P_{T,out}}{P_{T,in}}\right)_d - 1\right]}{\left[\left(\frac{P_{T,out}}{P_{T,in}}\right)_d - 1\right]}\frac{T_{in}}{T_{in\,d}}$$

$$\frac{t}{t_{ref}} \approx \frac{1}{\eta^*}\left(\frac{W\sqrt{T_{T,in}}}{P_{T,in}}\right)^*\left(\frac{\sqrt{T_{T,in}}}{\omega}\right)^* \frac{\left[\left(\frac{P_{T,out}}{P_{T,in}}\right)^{0^{0.4}}\left(\frac{P_{T,out}}{P_{T,in}}\right)_d^{0.4} - 1\right]}{\left[\left(\frac{P_{T,out}}{P_{T,in}}\right)_d^{0.4} - 1\right]}\frac{T_{in}}{T_{in\,d}}$$

with $\frac{q}{q_{ref}} \approx \left(\frac{W\sqrt{T_{T,in}}}{P_{T,in}}\right)^* \sqrt{\frac{T_{in}}{T_{in\,d}}}$ and $\frac{\omega}{\omega_{ref}} \approx \left(\frac{\omega}{\sqrt{T_{T,in}}}\right)^* \sqrt{\frac{T_{in}}{T_d}}$

The momentum and energy source terms due to the turbine are calculated from the head (H) and torque (T) characteristics.

The user gives the following input data:

- the inertia I of the motor
- the turbine rotation speed variation
- the reference velocity Ω_{Ref}
- the reference temperature
- the gas expansion
- the turbine characteristics which can be input in three ways:
  1. by using the library (see way 3)
  2. the user enters the interpolation function coefficients giving the h = H/H_R and t = T/T_R polynomial laws as a function of $\omega = \Omega/\Omega_{Ref}$ and q = $Q/Q_{Ref,}$ the reduced flowrate and velocity, the reference flow rate Q_R, the nominal head H_R multiplied by gravity and the nominal specific torque T_R

$$h = \frac{H}{H_R} = a + bq + c\omega + dq^2 + e\omega^2 + fq\omega + gq^3 + h\omega^3 + iqw^2 + hq^2w$$

$$t = \frac{T}{T_R} = a' + b'q + c'\omega + d'q^2 + e'\omega^2 + f'q\omega + g'q^3 + h'\omega^3 + i'qw^2 + h'q^2w$$

3. by giving the reduced efficiency (η) and the reduced expansion ratio as a function of reduced rotation speed (ω) and reduced flow rate (q). The reduced efficiency and expansion ratios are input as tables.

During the transient calculation and in the executable block, the turbine rotation speed can be modified with the TCOMMOD directive.

Several compressors and (or) gas turbines can be coupled and mounted on a given shaft using a SHAFT operator to create a CATHARE turbo-machine element.

44. HOW TO MODEL A COMPRESSOR?

REFERENCE:	**0 –The CATHARE code: SMTH/LMDS/EM/2001-063**
KEY WORDS	**TURBINE, COMPRESSOR, GAS COOLED REACTOR**
See also	***HOW TO MODEL a TURBO-MACHINE?*** ***HOW TO MODEL a TURBINE?***

The sub-module (TCOMCHAR directive) is used to model a compressor in a 0D approach. The user should enter the reduced characteristics of the compressor (the pressure ratio and the isentropic efficiency).

A compressor is located at a vector node. It affects the momentum and the energy equations. In the calculated Source terms the head (H) and torque (T) are derived from the specific turbine characteristics.

The momentum source term is assigned to the momentum equations written at this vector node and influences the pressure difference DP between the two neighbouring scalar nodes. The energy source term is assigned to the energy equations written at the scalar node which is downstream of the liquid velocity. The compressor characteristics are:

- ✓ The reduced compression ratio which is a function of the reduced rotation speed and the reduced flowrate.

$$\left(\frac{P_{in}}{P_{out}}\right)^{*} = f\left[\left(\frac{W\sqrt{T_{in}}}{P_{in}}\right)^{*}, \left(\frac{\omega}{\sqrt{T_{in}}}\right)^{*}\right]$$

- ✓ The reduced efficiency which is a function of the reduced rotation speed and the reduced flowrate.

$$\eta^{*} = g\left[\left(\frac{P_{in}}{P_{out}}\right)^{*}, \left(\frac{\omega}{\sqrt{T_{in}}}\right)^{*}\right]$$

The source terms H and T which will be used in the CATHARE equations are derived from these characteristics as follows:

$$\frac{H}{H_{ref}} \approx \frac{\left[\left(\frac{P_{T,out}}{P_{T,in}}\right)^{*}\left(\frac{P_{T,out}}{P_{T,in}}\right)_{d} - 1\right]}{\left[\left(\frac{P_{T,out}}{P_{T,in}}\right)_{d} - 1\right]} \frac{T_{in}}{T_{in\,d}}$$

$$\frac{t}{t_{ref}} \approx \frac{1}{\eta^{*}}\left(\frac{W\sqrt{T_{T,in}}}{P_{T,in}}\right)^{*}\left(\frac{\sqrt{T_{T,in}}}{\omega}\right)^{*} \frac{\left[\left(\frac{P_{T,out}}{P_{T,in}}\right)^{0^{0.4}}\left(\frac{P_{T,out}}{P_{T,in}}\right)_{d}^{0.4} - 1\right]}{\left[\left(\frac{P_{T,out}}{P_{T,in}}\right)_{d}^{0.4} - 1\right]} \frac{T_{in}}{T_{in\,d}}$$

with $\boxed{\frac{q}{q_{ref}} \approx \left(\frac{W\sqrt{T_{T,in}}}{P_{T,in}} \right)^{*} \sqrt{\frac{T_{in}}{T_{in\,d}}}}$ and $\boxed{\frac{\omega}{\omega_{ref}} \approx \left(\frac{\omega}{\sqrt{T_{T,in}}} \right)^{*} \sqrt{\frac{T_{in}}{T_{d}}}}$

The momentum and energy source terms due to the turbine are calculated from the head (H) and torque (T) characteristics.

The user gives the following input data:

- the inertia I of the motor
- the compressor rotation speed variation
- the reference velocity Ω_{Ref}
- the reference temperature
- the gas expansion
- The compressor characteristics which can be input in three ways:
  1. by using the library (see way 3)
  2. the user enters the interpolation function coefficients giving the $h = H/H_R$ and $t = T/T_R$ polynomial laws as a function of $\omega = \Omega/\Omega_{Ref}$ and $q = Q/Q_{Ref,}$ the reduced flowrate and velocity, the reference flow rate Q_R, the nominal head H_R multiplied by gravity and the nominal specific torque T_R

$$\boxed{h = \frac{H}{H_R} = a + bq + c\omega + dq^2 + e\omega^2 + fq\omega + gq^3 + h\omega^3 + iqw^2 + hq^2w}$$

$$\boxed{t = \frac{T}{T_R} = a' + b'q + c'\omega + d'q^2 + e'\omega^2 + f'q\omega + g'q^3 + h'\omega^3 + i'qw^2 + h'q^2w}$$

  3. by giving the reduced efficiency (η) and the reduced compression ratio as a function of reduced rotation speed (ω) and reduced flow rate (q). The reduced efficiency and expansion ratios are input as tables.

In addition, for compressors, a surge limit can be defined as input data. By means of the table, the user gives the reduced pumping flowrate as a function of the reduced flowrate (q). The surge limit calculation can be run or stopped using the STARSURG and the STOPSURG directives.

During the transient calculation and in the executable block, the compressor rotation speed can be modified with the TCOMMOD directive.

Several compressors and (or) gas turbines can be coupled and mounted on a given shaft using a SHAFT operator to create a CATHARE turbo-machine element.

45. HOW TO MODEL A TURBO-MACHINE?

REFERENCE:	**0 –The CATHARE code: SMTH/LMDS/EM/2001-063**
KEY WORDS	**GAS COOLED REACTOR, TURBO-MACHINE, COMPRESSOR, TURBINE**
See also	***HOW TO MODEL a TURBINE?*** ***HOW TO MODEL a COMPRESSOR?***

The CATHARE turbo-machine model consists of a coupling of gas-turbines, compressors and optionally a generator which are arranged in a single vertical shaft structure completely suspended on electromagnets (operator SHAFT).

Gas turbines (*HOW TO MODEL a TURBINE?***)** and compressors (*HOW TO MODEL a COMPRESSOR?***)** are defined with their dynamic characteristics in a 0D approach.

The transient behaviour (speed, w) of the single shaft requires solving the 1D shaft equation at each time step:

$$\boxed{\frac{\partial w}{\partial t} = \frac{\sum \Gamma_{hydr} - \Gamma_{mech} - \Gamma_{elec}}{I}}$$

I denotes the moment of inertia due to the turbine, I_T, the compressors, I_C and the generator, I_G. $\boxed{I = I_T + I_C + I_G}$

The hydraulic torque, Γ_{hydr}, is the sum of the hydraulic torques for turbines and compressors.

The mechanical friction torque, Γ_{mech}, is the sum of the individual friction torques (turbines, compressors and generator if modelled) and potentially a damping factor:

$$\boxed{\Gamma_{mech} = C_{f\_turbine} + C_{f\_compressor} + C_{f\_generator} + Damping}$$

The generator may be represented and connected to the structure. In the CATHARE model, the electrical (generator) torque (Γ_{elec}) may be represented in two ways:

1. simple way: the shaft rotation speed does not change (steady state):

$$\boxed{\sum \Gamma_{hydr} - \Gamma_{mech} = \Gamma_{elec}}$$

2. more detailed way: the electrical torque (Γ_{elec}) is a function of a swing angle (Φ) and the nominal speed (w_{ref}). This angle is given by the computation of a differential equation:

$$\boxed{\frac{\partial \Phi}{\partial t} = w - w_s}. \text{ and } \boxed{\Gamma_{elec} = C_0 \sin\Phi \frac{w_{ref}}{w}}$$

To define, a CATHARE turbo-machine model, a MASTER element (turbine or compressor) has to be defined. The MASTER element imposes its rotation speed on the other turbo-machine elements. These other components are defined as SLAVE elements.

A generator (alternator) can be coupled to the turbo-machine shaft. Alternator characteristics have to be input (simple or detailed characteristics).

The friction torque coefficients of the turbines ($A_{\_turbine}$), compressors ($A_{\_compressor}$), alternator ($A_{\_1generator}$ and $A_{\_2generator}$) and the damping parameter ($A_{damping}$), the load of the alternator (LOAD) and the

synchronism velocity have to be input. These parameters can be changed in the executable block by using the SHAFTMOD directive.

$$C_{f\_turbine} = \frac{A_{turbine}}{\omega_{ref}} \left(\frac{\omega}{\omega_{ref}} \right)^2$$

$$C_{f\_compressor} = \frac{A_{compressor}}{\omega_{ref}} \left(\frac{\omega}{\omega_{ref}} \right)^2 , C_{f\_generator} = \frac{A_{1\_generator}}{\omega_{ref}} \left(\frac{\omega}{\omega_{ref}} \right)^2 + \frac{A_{2\_generator}}{\omega_{ref}} LOAD$$

$$Damping = A_{damping} \left[1 - \frac{\omega_{ref}}{\max\left(\omega ; 4.2\,10^{-4}\right)} \right] LOAD$$

Calculation of the turbo-machine model can be de-activated using the STOPSHAF directive.

The coupling of the alternator to the turbo-machine can be de-activated or re-activated using the STOPALTR and STARALTR directives.

46. HOW TO MODEL AN ACCUMULATOR?

REFERENCE:	**-CATHARE 2 V2.5_1 - User Manual: SSTH/LDAS//EM/2005-035**
KEY WORDS	**ACCUMULATOR,VALVE, CHECK VALVE, TEE, DEAD ZONE**
See also	

There are two ways to model an accumulator with CATHARE:

- using the 0D accumulator sub-module;
- using the volume and pipe standard modules.

The 0D accumulator sub-module is the simplest way to represent the accumulator (no meshing and less CPU time). But in this case, the accumulator discharge line and wall effects are not taken into account. The user has to input an expansion coefficient to calculate a similar effect.

The volume-pipe way is a more detailed model (meshed modules) and needs to well know all geometries and topologies of the accumulator components (walls, discharge line, check valve, pressure drop and heat losses) to take into account of all hydraulic and thermal coupling between the accumulator, the external and the primary coolant system. Calculation CPU time is greater than in case of use of the 0D sub-module.

46.1 Modelling an accumulator with the accumulator sub-module

CATHARE has an **accumulator sub-module** which may be attached to an AXIAL module.

- ❖ It calculates the discharge flowrate at every time step and adds the appropriate mass, momentum and energy source terms to the equations for the mesh to which it is connected.
- ❖ It is explicitly coupled to the axial module (the discharge flowrate at time n+1 is calculated from the accumulator pressure at time n+1 and the pipe pressure at time n).
- ❖ It calculates an isentropic nitrogen expansion: γ= 1.4 but different values of γ may be imposed by the user (γ= 1 for an isothermal expansion). The heat released from the tank walls is not calculated.
- ❖ It takes into account time acceleration, friction and singular pressure losses and gravity head in the discharge line.
- ❖ The check valve (non-return valve) is modelled. The accumulator water will be discharged only if conditions for opening the check valve are fulfilled (Paccu > Ppipe)
- ❖ If condensation occurs at the injection point, a specific condensation model is used locally which takes into account the local increased mixing processes due to the jet. This model was qualified on the COSI experiment.
- ❖ The diluted nitrogen can be modelled and injected in the circuit. It is assumed that there is complete de-gassing when entering the circuit. At the end of the water discharge, the discharge of nitrogen can also be modelled with the option NOCLOSE. Do not forget to define the non-condensable gas in the circuit.
- ❖ The hysteresis in the opening or closure of the valve can be taken into account (VALVE keyword).

46.2 Modelling an accumulator with the standard modules

An accumulator may also be represented using the standard modules of the code:

- ❖ a volume module for the tank;

- an axial module for the discharge line;
- a tee module for the connection between the discharge line and the main pipe where water is injected;
- the check valve is modelled with a CHECK VALVE sub-module which is defined at the end of the discharge line (see *HOW TO MODEL a VALVE?*).
- an isolation valve is defined with a CONTROL VALVE at the end of the discharge line at the same location as the CHECK VALVE. It is used to initialise the accumulator DEADZONE. (see *HOW TO MODEL a VALVE?*).

To calculate the steady state, a DEADZONE including the elements which belong to the accumulator (tank and discharge line) must be defined in the PERMINIT directive.

- The CHECK VALVE is disabled
- The CONTROL VALVE is enabled in the closed position in order to isolate the accumulator completely.
- The thermal-hydraulic conditions in such an accumulator are imposed by the REALVO, REALAX and LEVEL directives (see *HOW TO initialise a DEADZONE?*).

To calculate the transient step, the CONTROL VALVE is fully opened and the CHECK VALVE is enabled. The check valve management system will allow the accumulator to discharge when the pressure conditions are reached ($P_{line} > P_{pipe}$).

At any moment the accumulator can be isolated by closing the CONTROL VALVE (CLOSE directive).

This model can be used to calculate the heat released from the accumulator walls and its effect on the pressure.

This model can represent the nitrogen discharge at the end of the water discharge but not the diluted nitrogen.

47. HOW TO MODEL RADIO-CHEMICAL COMPONENTS?

REFERENCE:	**0 –The CATHARE code: SMTH/LMDS/EM/2001-063**
KEY WORD	
See also	**BORON, ACTIVITY**

Boron and/or **radio-active elements** may be defined in a circuit or in a reactor. A maximum of 12 different elements can be defined. When using several circuits, these elements must be specified for each of circuit.

Physically, this function is used to represent core activity leaks and boron injection due to safety facilities and the Chemical and Volume Control System.

The **RADCHEMI** operator is used to define all radio-chemical components used in the calculation. A set of pre-defined elements is available and personal user components defining certain properties can be included.

The **INIBORA** directive is used to define the initial concentration (or to reinitialise this concentration during the calculation) of each element defined in the RADCHEMI operator in any part of the circuit.

The **GOBORA** directive launches the transport calculation. The boron will be transported by the liquid phase and the other elements by both phases. This calculation does not interfere with the standard thermal-hydraulic calculation, but simply adds transport and transfer between phases (liquid/solid for boron and liquid/gas for the other elements) in the system of equations to be solved.

As boron deposition is calculated using an explicit test on the solubility threshold, it is recommended to check that the boron mass in the solute is not time-step dependent.

48. HOW TO MODEL A BREAK?

REFERENCE:	**-**
KEY WORDS	**BREAK, GUILLOTINE RUPTURE,TEE**
See also	***HOW TO CHOOSE a NODALISATION FOR A PIPE?***

Break modelling depends on the size and geometry of the break. When a pipe is fully broken it requires a boundary condition such as BC4 or RUPTURE for a double-ended break. When there is a break in an element with a smaller cross-section than the element itself there are several possibilities:

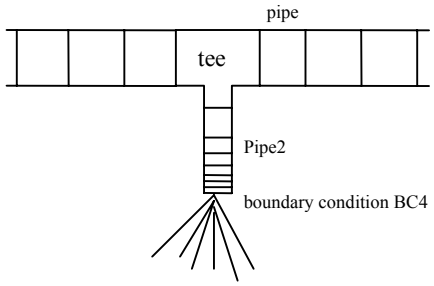
- If the broken element is a pipe, the break can be modelled with
  - ✓ a TEE, a PIPE and a BC4
  - ✓ a BREAK or PIQBREK sub-module.
- If the broken element is a volume, the break can be represented with
  - ✓ a PIPE and a BC4
  - ✓ a SAFETY SINK or PIQBREK sub-module

BC4 or RUPTURE boundary conditions impose either the containment pressure or the sonic blockage. If the sonic condition is reached with a higher pressure than the containment pressure, then the sonic condition is imposed. The change in containment pressure must be specified by a time-dependent law. In the RUPTURE model, the pressure may also be imposed by WRITE.

48.1 Break modelled with a TEE, a PIPE and a BC4

This is the recommended modelling procedure, particularly for breaks on horizontal pipes; since it takes into account the phase separation effects which are modelled in the TEE (see qualification reports). No specific phase separation law was developed in the TEE for a vertical pipe.

Even without a phase separation phenomenon, the critical flow prediction is more accurate when calculated with 1-D modelling of the flow in the nozzle (see Qualification reports) than by using a critical flow correlation function of upstream parameters (Break sub-module).



Nodalisation recommendation

The pipe2 mesh must decrease in size when approaching the break. According to some numerical rules, the length ratio of two successive meshes should not exceed 1.2. In practice it is recommended not to exceed 1.5.

The last mesh element before the break should not exceed 1mm. The two last meshes before the break should have the same length. The length of the first mesh after the TEE sub-module should be close to the length of the TEE component. The user has to adjust the total length according to all the previous meshing rules.

These recommendations are the result of some mesh convergence tests performed for the qualification tests. These recommendations are somewhat subjective since they depend on how much error is allowed. Users may also perform their own mesh convergence tests.

48.2 Break modelled with a BREAK sub-module

The BREAK sub-module can be defined on a scalar point of a pipe. The break is activated (or opened) with the OPEN directive. The break section may be modified with the VALVSECT CCV and its position with the IPSINT CCV.

The BREAK sub-module is treated as a sink term using a critical mass flowrate correlation (Gros d'Aillon correlation): it always assumes that the difference between the internal pressure and the outside pressure is high enough to ensure a sonic flowrate.

Coupling is implicit. It allows a simple representation of a break which saves CPU time. There is no need for fine nodalisation at the break location. **But the critical flowrate may not be sufficiently accurate; it cannot take into account phase separation effects**.

48.3 Break modelled with an external VALVE SINK sub-module

The EXTERNAL VALVE SINK sub-module can be defined on a volume. The break is activated (or opened) with the OPEN directive. The break section may be modified with the VALVSECT CCV but its position cannot be changed.

The EXTERNAL VALVE SINK is treated as a sink term using a critical mass flowrate correlation (Gros d'Aillon correlation): it always assumes that the difference between the internal pressure and the containment pressure is high enough to ensure a sonic flowrate.

Coupling is implicit. **But the critical flowrate may not be sufficiently accurate; it cannot take into account phase separation effects.**

48.4 Break modelled with a SAFETY SINK sub-module

The SAFETY SINK sub module can be defined on a volume. The break is activated (or opened) with the OPEN directive. The break section may be modified with the VALVSECT CCV but its position cannot be changed.

The SAFETY SINK is treated as a sink term using a critical mass flowrate correlation (Gros d'Aillon correlation): it always assumes that the difference between the internal pressure and the containment pressure is high enough to ensure a sonic flowrate. The critical mass flowrate depends on the safety valve opening rate (defined by the calibration pressure and the pressure difference at the maximum opening).

Coupling is implicit. **But the critical flowrate may not be sufficiently accurate; it cannot take into account phase separation effects**.

48.5 Break modelled with a PIBREK sub-module

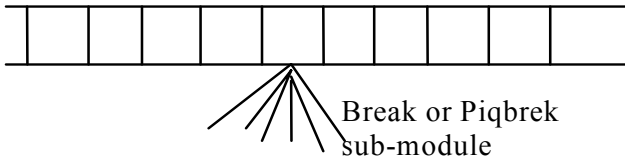
The PIQBREK sub-module can be defined on a scalar point of a pipe or in a volume. The break is activated (or opened) with the OPENBREK directive. The break section and position are chosen on activation.

In the PIQBREK sub-module, flow reversal is calculated. This depends on the outside pressure conditions. The user has to define the outside conditions by CCV (WRITE): temperature of liquid, gas, void fraction, non-condensable mass fraction and pressure. They will be used:

- if the outside pressure is higher than the pressure in the mesh that the PIQBREK is connected to, the PIQBREK sub-module is treated as a source term (mass and energy)
- if the outside pressure is lower than the pressure in the mesh that the PIQBREK is connected to, the PIQBREK sub-module is treated as a sink term (mass and energy) using the minimum value of a critical mass flowrate correlation (Gros d'Aillon correlation) and a classical subsonic flowrate estimated from the pressure difference between the inside and outside of the circuit.

Coupling is implicit. It allows a simple representation of a break which saves CPU time. There is no need for fine nodalisation at the break location. **But the critical flowrate may not be sufficiently accurate; it cannot take into account phase separation effects**.

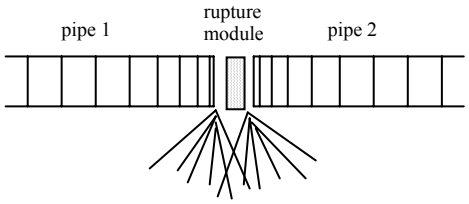
Common errors: do not forget to define the external thermal-hydraulic conditions in case of PIQBREK



48.6 Double-ended break modelled with a guillotine rupture module

LARGE BREAK LOCAs are modelled by the **'RUPTURE'** module, which is a double-ended break between two pipes. Before the break opens, pipe 1 and pipe 2 are just connected. After the break opens, this module consists of two BC4 boundary conditions. The time-dependent outside pressure, temperature (and nitrogen mass quality in case of N_2 calculation) must be given by the user.

Nodalisation recommendations near the two breaks are the same as above: 1 mm for the last mesh and a mesh length ratio lower than 1.5



49. HOW TO MODEL A BEND?

REFERENCE:	
KEY WORDS	**BEND, INTERMEDIATE LEG**
See also	***HOW TO CHOOSE a MESH FOR A PIPE?*** ***HOW TO MODEL an INTERMEDIATE LEG?***

The nodalisation of a bend depends on the importance of the phenomena which occur there during the transient. It also depends on the orientation of the bend:

- if the bend is in a horizontal plane, only the singular pressure drop due to the bend need be taken into account.
- if there are changes in elevation along the bend axis, there might be more complex phase separation phenomena in two-phase situations - like in the intermediate leg of a PWR- which require a finer nodalisation procedure. The analysis of the ECTHOR tests resulted in the following recommendations for proper calculation of voiding tests:
  - at least 3 mesh elements are recommended,
  - each of these 3 meshes must have its own orientation (with respect to the vertical axis).

50. HOW TO MODEL AN INTERMEDIATE LEG?

REFERENCE:	
KEY WORDS	**BEND, INTERMEDIATE LEG**
See also	***HOW TO CHOOSE a MESH FOR A PIPE?*** ***HOW TO MODEL a BEND?***

The intermediate legs of many PWRs are characterised by a U shape which may behave as a loop seal.

In many SBLOCA transients, a liquid plug forms in this loop seal preventing steam from flowing from the SGs to the cold legs. Then, pressures and temperatures in the hot and cold legs become more independent and this quite often results in a lower pressure in the cold legs than in the hot legs.

This situation is dangerous for core cooling, as there is a level depression in the core. Different liquid levels in the upstream and downstream vertical legs of the U tubes are established and the liquid slug is removed when the liquid level in the upstream vertical leg reaches the horizontal pipe at the bottom of the U. This process of loop seal clearing may be very sensitive to small differences in the various loops. One loop seal clearing may be sufficient to re-establish equal pressures in hot and cold legs.

The situation was studied in the ECTHOR and IVO test facilities. CATHARE gives good predictions provided that the nodalisation is sufficiently fine. Three mesh elements per bend are necessary.

51. HOW TO MODEL AN ABRUPT AREA CHANGE?

REFERENCE:	
KEY WORDS	**VALVE, FLOW LIMITER**
See also	

An abrupt area change is a local flow restriction or flow enlargement with sharp edges in a duct. The AXIAL module of the code cannot describe them. This module describes only continuous cross-section area variations along the axis through successive truncated cones. If the large area side may be described by a volume module and the smaller area side by a pipe, all balance equations including the momentum equations are written correctly. If such an abrupt are change has to be located within an AXIAL module, certain precautions must be taken.

If the abrupt area change is used as a flow limiter, you should refer to *HOW TO MODEL a VALVE?*

Recommendations:

- Use conical shape area changes. The cone angle must not be too sharp (maximum 45° for example but sensitivity tests may be required)
- The singular pressure drop must be taken into account (SINGULAR directive). Different singular pressure drop coefficients are necessary for the two possible flow directions (See *HOW TO MODEL a SINGULAR PRESSURE DROP?* and *WHAT is a SINGULAR PRESSURE DROP?*).
- Numerical problems and physical inconsistencies are likely to occur in case of a strong flow area enlargement with high velocities in two-phase situations (for instance V > 20m/s).

52. HOW TO MODEL A DIAPHRAGM?

REFERENCE:	
KEY WORDS	**FLOW LIMITER, BREAK, SINGULARITY, PRESSURE DROP**
See also	

A diaphragm is a local flow restriction with sharp edges in a duct. It can be at the downstream end of a pipe (at a break for example) or inside a pipe. SG support plates, core spacer grids and upper tie plates may be considered as diaphragms.

The nodalisation depends on which phenomena are expected to occur in this diaphragm during the calculated transient.

Recommendations:

- If the diaphragm is used as or is part of a flow limiter, the user should use the **FLOW LIMITER** operator that takes into account the area restriction and the singular pressure drop (refer to *HOW TO MODEL a VALVE?*).
- In other cases, the singular pressure drop must be taken into account (SINGULAR directive). The SGPLATE option is recommended (See *HOW TO MODEL a SINGULAR PRESSURE DROP?* and *WHAT is a SINGULAR PRESSURE DROP?*). Most of the time, modelling of the area restriction is not necessary except:
  - if choked flow conditions are expected,
  - for stratified flows in horizontal pipes.
- **If choked flow conditions are expected to occur**
  - It is necessary to model the area restriction in the pipe geometry.
  - As abrupt area changes are not possible in AXIAL modules, use conical shape area changes on both sides of the diaphragm. The angle of the cone must not be too sharp (maximum 45° for example but sensitivity tests may be required).
  - Use locally fine nodalisation: a maximum of 1mm long meshes at the area restriction.
  - If it is located at the downstream end of a break pipe, a BC4 boundary condition can be used (See *HOW TO MODEL a BREAK?*)
- **If CCFL conditions are expected to occur (See How to control CCFL and Flooding limit?)**
  - If the diaphragm is located at a pipe-volume junction (with the volume above the pipe), the CCFL option may be used. In this case there is no need to represent the area reduction.
  - If the diaphragm is not located at a pipe-volume junction, there is no simple way of controlling the CCFL and flooding limit, regardless of the nodalisation, either with or without modelling the area reduction. If the flooding limit in this particular geometry is known, the best procedure would be to test several nodalisations and choose the most appropriate one.
- If it is a horizontal pipe and if stratified flow is expected to occur in this pipe, it is necessary to represent the flow area reduction in the geometry because the diaphragm may be an obstacle for the liquid and not for the gas. Use at least two meshes for the small cross-section area.

53. HOW TO MODEL A MANWAY?

REFERENCE:	
KEY WORDS	**BREAK, GUILLOTINE RUPTURE, BOUNDARY CONDITION, BLIND**
See also	

A MANWAY is a way on a VOLUME element allowing maintenance operations to be carried out on the circuit. For example, on a PWR, MANWAYs are located:

- at the inlet and outlet of the STEAM GENERATOR plena
- at the top of the PRESSURISER.

The best way to model a MANWAY is to define a BOUNDARY CONDITION at the level of the MANWAY, and then use a MODEL that depends on its state:

- when it is closed, it should be modelled by a BLIND MODEL,
- when it is open, it should be modelled by a BC5HOLE MODEL

Opening the MANWAY is equivalent to switching the MODEL used from BLIND to BC5HOLE. The user may specify a time interval for a smooth transition from the fully closed condition to the fully open condition.

Closing the MANWAY is equivalent to switching the MODEL used from BC5HOLE to BLIND. The user may specify a time interval for a smooth transition from the fully open condition to the fully closed condition.

The state of the MANWAY may be checked by the VALUE directive.

The BC5HOLE model requires the definition of the following external conditions: the external pressure, the external temperatures, void fraction, non-condensable gas mass fractions and radio-chemical fractions. The external values must be imposed by the WRITE directive from the input deck which allows an explicit coupling with containment codes for example.

The equations written follow the standard BOUNDARY set of equations presented in *WHAT is a BOUNDARY CONDITION?*

The model allows liquid-gas counter-current flow.

The junction of the VOLUME to which the MANWAY is connected should be declared HORIZONTAL, the elevation being that of the axis of the MANWAY and the section being the projection of the real section onto a vertical plane.

54. HOW TO MODEL A VALVE?

REFERENCE:	**- CATHARE 2 V2.5_1 - Description of the valve sub-modules: SMTH/LMDS/EM/2005-040**
KEY WORDS	**FLOW LIMITER, CHECK VALVE, SAFETY VALVE, FAILURE (VALVE)**
See also	

A valve may be located:

- ❖ in a circuit in order to partially or fully isolate a sub-circuit or a part of a circuit.
- ❖ at a boundary of a circuit for extracting mass and energy from a circuit.

The user may either impose the singular pressure drop or flowrate through the VALVE when no VALVE characteristics are known, or define the type of VALVE to be modelled and the associated values:

54.1 Modelling of a valve of unknown hydraulic characteristics

A valve within a circuit may be modelled in an AXIAL element by a VALVE directive. It is located at a vector node and simulates the valve by adding a singular pressure drop coefficient. To open the valve, the user has to reset the singular pressure drop coefficients to their initial values.

54.2 Modelling of a valve of known hydraulic characteristics

54.2.1 A VALVE in a circuit

A valve in a circuit can be modelled in an AXIAL element (including its junctions), at the junction of a TEE or at the junction of a VOLUME by a VALVE sub-module. It is located at a vector node and simulates the valve from a more industrial point of view, making it easier to simulate its position settings to control flowrates or temperatures.

The model is based on a modification of the standard momentum equations of the module to which the valve is connected: it takes into account interfacial friction effects as the valve tends to close and singular pressure drop modelling of the SGPLATE type. It is also able to limit the flowrate in critical conditions, using the Gros d'Aillon correlation.

The valve flowrate is implicitly coupled to the pressure gradient through the VALVE by the momentum equations of the vector mesh on which the VALVE is defined.

It is assumed that the flow is not stratified in the valve itself, so that a computation with a fully open valve is not completely equivalent to a computation with no valve.

The pressure drop through the valves is computed by an SGPLATE type correlation using a singular pressure drop coefficient derived from the valve position and flowrate coefficient. There are two models; one is suitable for liquid flows, while the other is suitable for vapour-gas flows. The appropriate model is selected according to the value of the reference density given in the input deck. The recommended value for the reference density is 1000 kg/m^3 for liquid flow type valves, and 1 kg/m^3 for vapour-gas flow type valves.

Definition of the valve characteristics depends on the type of valve:

- **CONTROL VALVE**: the variation in the flowrate coefficient CV is given in the input deck as a function of the valve position PU; PU is set between 0 and 1 by the user using the WRITE directive.
- **CHECK VALVE**: the opening (PU between 0 and 1) depends on the pressure gradient through the valve with respect to DELTAP (necessary pressure gradient to obtain full opening of the check valve).
- **FLOW LIMITER**: the opening is set in the DATA BLOCK through the SR key word.

When information is missing, some simplifications may be used:

- If the CONTROL VALVE is only used to isolate a part of a circuit: a large value for the nominal flowrate coefficient together with a simple law such as (PU = 0, CV = 0), (PU=1, CV=5000) is sufficient.
- If the CHECK VALVE is only used to prevent the point of the circuit from reverse flow, a CVMAX value of 3000 to 5000 and a value of 1000 Pa for DELTAP are sufficient for rough use of the model, ensuring that the pressure loss for full opening will be small.

An active VALVE (OPEN), fully closed (PU = 0.) imposes that the velocities are set to 0 on the corresponding vector mesh. This is useful for isolating parts of a circuit that will be initialised in PERMINIT as DEADZONE where the fluid may have completely different properties (temperatures, void fractions… see *HOW TO INITIALISE a DEADZONE?*). These zones may also be re-initialised by REINIT.

The valves connected to the same point may be grouped together by the ECHECK or ECVALVE directive. Each VALVE of the group follows the scheme of one VALVE. Refer to the Dictionary of Directives and Operators.

Typical applications of the valve sub-module are:

- control-valves for the pressuriser spray regulations, the steam line conditioning, temperature or flowrate regulation in the RHRS, ….
- check-valves in the safety injection system and in the RHRS, …..
- flow limiter at the outlet of steam generators, …..

54.2.2 A VALVE at a circuit boundary

A valve at a circuit boundary may be modelled on an AXIAL or a VOLUME element by a PIQVANNE sub-module to model the mass (and corresponding energy) exchange of an element with the outside which is not necessarily part of the CATHARE input deck (containment or any other system for example).

The model adds mass, energy and momentum source terms in the equations of the element to which it is connected. The model is able to predict sonic and subsonic flowrates, using the Gros d'Aillon correlation for critical conditions and the Bernoulli formula for sub-critical flows. Separation effects between phases are not taken into account.

The flowrate through the valve is computed by the code and is positive in SOURCE-type behaviour and negative in SINK-type behaviour.

The external conditions (i.e., the injected fluid characteristics) must be imposed from the input deck by WRITE. These conditions are: the external pressure, the liquid and gas temperatures, the void fraction, the non-condensable gas mass fraction. A slip ratio of 1 is assumed between the liquid and gas phases.

The valve flowrate is implicitly coupled to the pressure gradient through the PIQVANNE sub-module. The PIQVANNE sub-module simulates the valve from an industrial point of view, making it easier to simulate its position settings (to control flowrates or temperatures for example):

A PIQVANNE sub-module may be made of up to four sub-components including a CONTROL VALVE and a CHECK VALVE (refer to the dictionary for more information) which exchange with the outside of the circuit. The CHECK VALVE has no intermediate position between fully open and fully closed.

- For valve definition the user must define a flowrate coefficient as a function of the rate of opening of the valve. The pressure drop through the valve is computed from this flowrate coefficient.
- Once the PIQVANNE sub-module is connected via OPEN, the valve position is managed by the user via the 'PU' CCV, 0 meaning closed and 1 full opening.

Typical applications of the PIQVANNE sub-module are: vents, atmospheric relief valves, turbine modelling, etc.

54.2.3 Valve failures

Several kinds of failure such as fouling, leakage, inadvertent opening, etc., can be simulated (see *HOW TO MODEL VALVE FAILURES?*).

54.2.4 Remarks

Be careful: CLOSE means that the valve will no longer be modelled. To model a closed valve, activate it by OPEN, then set its position to fully closed by setting its position rate 'PU' to 0.

55. HOW TO MODEL A SAFETY VALVE?

REFERENCE:	**- CATHARE 2 V2.5_1 - Description of the valve sub-modules: SMTH/LMDS/EM/2005-040**
KEY WORDS	**SAFETY VALVE, BOUNDARY CONDITION, FAILURE (VALVE), SINK**
See also	***HOW TO MODEL a SINK?***

A safety valve is used to extract mass from a circuit and to avoid an excessive pressure increase. It opens automatically when pressure exceeds the safety valve pressure set point. Total opening of the valve requires a pressure higher than the opening pressure.

Safety valves can be modelled either by a boundary condition, by a SINK-SAFETYVA sub-module, a PIQSEB sub-module or a PIQSOUP sub-module, connected to an AXIAL or a VOLUME module.

Some safety valves have a closure pressure lower than the opening pressure:

- a particular model describes this effect for SEBIM safety valves in CATHARE. They correspond to the PIQSEB operator.
- otherwise, the only way to take this effect into account is to use a BCMOD in the input deck and to control the opening or the closing of the valve from the input deck using the appropriate logic.

55.1 Safety Valves in a Boundary condition

A boundary condition can be of the SAFETYVA type (see *HOW TO CHOOSE and to use a BOUNDARY CONDITION?*).

55.2 Sink Safety Valve sub-modules

A safety valve connected to an element may be modelled by:

- an EXTERNAL safety valve (SINK SAFETYVA directive). The valve section, the opening pressure and the DP necessary to obtain full opening of the valve are given by the user. The valve section may be modified during the transient by the WRITE directive. The valve flowrate is calculated using a critical flow correlation.
- a PIQSOUP safety valve. The valve section, the opening pressure and the DP necessary to obtain full opening of the valve are given by the user in the DATA block of the input deck. The valve flowrate will be the minimum of the sonic and subsonic flowrates. The opening rate is determined by an interpolation at the actual pressure gradient between 1 for the full opening pressure gradient and 0 for a negative pressure gradient. Many failures may be imposed on these safety valves (see *HOW TO MODEL VALVE FAILURES?*).
- a PIQSEB safety valve that precisely describes the way a SEBIM safety valve works with different pressures for the beginning of opening, full opening, beginning of closing and full closing. Many failures may be imposed on these safety valves (see *HOW TO MODEL VALVE FAILURES?*).

56. HOW TO MODEL VALVE FAILURES?

REFERENCE:	**- CATHARE 2 V2.5_1 - Description of the valve sub-modules: SMTH/LMDS/EM/2005-040**
KEY WORDS	**CHECK VALVE, FAILURE (VALVE), SAFETY VALVE**
See also	

Valve failures may be triggered using the WRITE directive in the "exec block" of the input deck, with the appropriate CCVs (CATHARE Control Variables) to identify the type of failure (corresponding to a failure index). The failure of a valve may be deactivated by setting its failure index to zero by means of the WRITE directive. Not more than one failure index can be assigned to a valve, but some failure indices involve the occurrence of several failures of different types concomitantly.

The objects which may be subjected to failures are:

- ✓ CONTROL and CHECK VALVES
- ✓ PIQVANNE sub-modules, including CONTROL AND CHECK valves
- ✓ PIQSOUP safety valves
- ✓ PIQSEB safety valves.

Possible failures for CHECK and CONTROL valves are leakage, fouling, spurious lift, spurious closure, closure sticking, opening sticking, position sticking, spurious closure + leakage and spurious lift + fouling.

Possible failures for PIQSOUP and PIQSEB safety valves are leakage, fouling, spurious lift, spurious closure, closure sticking, opening sticking, position sticking, spurious closure + leakage, spurious lift + fouling and derivatives of valve characteristics.

57. HOW TO MODEL A SINGULAR PRESSURE DROP?

REFERENCE:	
KEY WORDS	**BREAK, RUPTURE**
See also	***WHAT is a SINGULAR PRESSURE DROP?*** ***HOW TO MODEL an ABRUPT AREA CHANGE?*** ***HOW TO MODEL a DIAPHRAGM?*** ***HOW TO MODEL a VALVE?*** ***HOW TO MODEL a BEND?***

Recommendations:

- Do not confuse 'singular pressure drops' with pressure losses. A singular pressure drop is an irreversible pressure loss whereas some pressure losses are reversible such as gravity head or acceleration-deceleration terms.
- If the singular pressure drop is modelled by a particular model as a CHECK VALVE, CONTROL VALVE or FLOW LIMITER, no modification of the nodalisation or of the singular pressure drop coefficient of the component are necessary as they are computed by the model.
- When coefficients K are entered, they correspond to velocities calculated at the vector node where K is defined.
- For a singular pressure drop at a flow area restriction, it is recommended to use the SGPLATE option (See *HOW TO MODEL a DIAPHRAGM?*).

Singular pressure drop at an abrupt area change:

Define singular pressure drop coefficients so that the pressure change is monotonous.

58. HOW TO MODEL HEAT LOSSES?

REFERENCE:	**-CATHARE 2 V2.5 - Description of the wall, heat exchanger and 0D steam generator sub-modules: SSTH/LDAS/EM/2004-049**
KEY WORDS	**HEAT LOSS**
See also	

In the description of a wall (WALL operator) the user may specify an exchange with the outside by using the key word LOSS. There are three possibilities:

- To give the radiative exchange coefficient ($W/m^2/°C$) and an outside temperature (C). In this case, the option RADEXT has to be used to enable a radiative exchange heat transfer. In this case, the considered wall and the ouside may be considered of infinite length with respect to their distance. Both are close enough to consider that the flux lost by the wall is equal to the flux received by the outside. The model is not able to predict the fluxes exchanged between a small wall inside another one. The wall and the outside are “grey walls” so their respective emissivities do not depend on their temperatures.
- To give an **exchange coefficient** ($W/m^2/°C$) and an **outside temperature** (°C). During the calculation, these values may be modified by using the directive FLUMOD applied to the corresponding wall.
- To give a **heat flux** (W/m^2) with the outside.

The exchange with the outside can be modified in the executable blox using the FLUMOD directive. The EXTEND option of the FLUMOD directive can be used to change the mode of heat transfer. In this case, the radiative heat transfer can be modified by giving the new non-dimensional surface emissivity.

59. HOW TO MODEL A HEAT SINK?

REFERENCE:	**-CATHARE 2 V2.5 - Description of the wall, heat exchanger and 0D steam generator sub-modules: SSTH/LDAS/EM/2004-049**
KEY WORDS	**HEAT LOSS, HEAT SINK**
See also	

A heat sink may be modelled with CATHARE using

- A LOSS term in a wall (see *HOW TO MODEL FACILITY HEAT LOSSES?*)
- A CANDLE term which creates a pure energy sink. Two types of injection can be defined:
  - INTERNAL CANDLE: the injection is automatically controlled in the computation by giving a variation law when the CANDLE is defined.
  - EXTERNAL CANDLE: the injection is controlled by the user in the executable block of the data set, using CATHARE Computation Variables (CCV).

60. HOW TO MODEL A HEAT SOURCE?

REFERENCE:	**-CATHARE 2 V2.5 - Description of the wall, heat exchanger and 0D steam generator sub-modules: SSTH/LDAS/EM/2004-049**
KEY WORDS	**HEAT LOSS,**
See also	**HEAT SOURCE**

A heat source may be modelled with CATHARE using

- A SOURCE term in a wall. In this case, the volume power has to be input by the user in the input deck (see *HOW TO MODEL a PRESSURISER?* for PRZ heaters)
- A CANDLE term which creates a pure energy source. Two types of injection can be defined:
  - INTERNAL CANDLE: the injection is automatically controlled within the computation by giving a variation law when the CANDLE is defined.
  - EXTERNAL CANDLE: the injection is controlled by the user in the executable block of the data set, using CATHARE Computation Variables (CCV).

61. HOW TO MODEL A SINK?

REFERENCE:	**-**
KEY WORDS	**SINK,CONTAINMENT**
See also	For safety valve modelling refer to *HOW TO MODEL a SAFETY VALVE?* For valve modelling, refer to *HOW TO MODEL a VALVE?*

A sink-type object will be used to extract mass (and corresponding energy) from an element. The extracted flowrate is lost by CATHARE. Mass, momentum and energy sink terms are added in the equations of the module to which the sink is connected.

Sink can be modelled with the SINK, PIQREV, PIQVANNE, SAFETYVA SINK, PIQSOUP or PIQSEB sub-modules. The choice depends on

- The kind of sink (safety valve, break, etc.)
- Whether or not the sink mass flowrate is known
- The kind of mass flowrate law (function of time or pressure or other)
- Whether other devices have to be represented

Sinks can be connected to pipes (AXIAL elements) or VOLUMES. They must be activated by the OPEN directive.

61.1 The sink mass flowrate is known

The sink mass flowrate has to be given:

- either using a law to be input in the data block (e.g., HPSI or LPSI for PWR). In this case an INTERNAL SINK can be used. INTERNAL means that the mass flowrate is controlled by the code itself in the subroutine GBREAK (call to SLAWP).
- or the mass flowrate has to be calculated in the execution block at each time step. In this case, an EXTERNAL SINK or a PIQREV sub-module can be used. This may be a function of flow variables calculated somewhere else in the circuit. It can be controlled from the input deck through CCVs (WRITE directive). The dependence is explicit. To obtain a better estimate of the extracted flowrate in liquid single-phase condition, it is advisable to compute the flowrate from the pressure gradient between the element and "the outside" by defining a hydraulic component such as a VALVE as described in *HOW TO MODEL a VALVE?*.
    - For an EXTERNAL SINK, only a negative flow is allowed (outlet flow from the circuit).
    - In the case of a PIQREV, flow reversal is possible (negative for outlet flow from the circuit and positive for inlet flow from the circuit). In case of positive flow, the PIQREV is treated as a source model (*HOW TO MODEL a SOURCE?*)

61.2 The sink mass flowrate is unknown

The sink flowrate has to be calculated by CATHARE. The available model depends on the kind of sink to be modelled: single sink (break, etc.), or sink associated with other devices.

61.2.1 A single sink losing mass (break, sink)

- If the sink represents a break see *HOW TO MODEL a BREAK?*

- If the sink represents a single loss of mass (turbine system), the EXTERNAL VALVE SINK can be used (see *HOW TO MODEL a BREAK?*)

61.2.2 A sink with other devices

- Model of a group of valves: a PIQVANNE sub-module can be used for a condenser or atmospheric steam dump, pressuriser or upper head vents or turbine system (see *HOW TO MODEL a VALVE?*).
- Model of safety valves: EXTERNAL SAFETY SINK, PIQSOUP (steam line safety valves) or PIQSEB (pressuriser safety valves) can be used (see *HOW TO MODEL a SAFETY VALVE?*)

61.3 SINKRRI special sink model

A SINKRRI sub-module is a special sink designed to model the safeguard systems in recycling conditions. By these systems, water is drawn out of the bottom of the containment, cooled through exchangers and re-injected into the containment spray system or into the safety injection system. A SINKRRI must be defined on the volume located at the bottom of the containment. It is a normal sink including, in addition, a description of safeguard systems.

62. HOW TO MODEL A SOURCE?

REFERENCE:	**-**
KEY WORDS	**SOURCE,CONTAINMENT**
See also	

Sources of water, steam, non-condensable gases or radio-chemical components in liquid or steam phases may be represented by CATHARE. Such fluids may also be injected into a circuit through boundary conditions.

The flowrate may be imposed by the user via the WRITE directive, or computed from the pressure in the element.

Sources can be connected to pipes (AXIAL elements) or VOLUMES. They must be activated by the OPEN directive.

Mass, momentum and energy source terms are added in the equations of the module.

When a source injects subcooled water in a mesh of an AXIAL element, a specific local condensation model is used in the injection mesh taking into account the increased mixing effects due to the jet. This specific model was qualified on COSI tests.

SOURCE terms can be modelled either with the SOURCE sub-module or the PIQREV sub-module or the PIQARE sub-module.

62.1 SOURCE sub-modules

Sources can be INTERNAL or EXTERNAL:

- INTERNAL means that the mass flowrate is a function of time or of local variables (pressure, etc.). The dependence is implicit. The injection is controlled by the code itself in the subroutines EINJEC (for AXIAL or BIDIM elements) or VINJEC for a volume. These subroutines may be modified by the user.
- EXTERNAL means that the mass flowrate is a function of flow variables calculated elsewhere in the circuit and can be controlled from the input deck through VCC's. The dependence is explicit.

Be careful

Sources are taken into account in the steady state calculations when they have been activated by OPEN before the GOPERM directive. The user has to check that a stable state is possible under the chosen conditions (flowrate or no flowrate, etc.).

62.2 PIQREV sub-modules

- The total mass flowrate and the injected fluid characteristics (temperature, void fraction, non-condensable mass fraction, radio-chemical fractions) are always imposed from the input deck (WRITE directive). They may be functions of flow variables calculated elsewhere in the circuit or by an other code and can be controlled from the input deck through CCV's. The dependence is explicit and a no-slipping condition is assumed between the liquid and gas phases.
- The sign of the flowrate must be positive to have a SOURCE-type behaviour; otherwise it switches to a SINK type behaviour (See HOW to model a SINK?)
- To obtain a better estimate of the injected flowrate in a liquid single-phase condition (water hammer risks), it is better to compute the flowrate from the pressure gradient between the element and “the outside” by defining a hydraulic component such as a VALVE with a rather small fictitious capacity, as described in *HOW TO MODEL a VALVE?*.

62.3 Feedwater overflow modelling in case of N4 reactor (PIQARE sub-module)

A special model is available for modelling N4 steam generator main feedwater: the user should refer to the PIQARE operator in the Dictionary.

The PIQARE object simulates Main Feed Water (MFW) overflowing phenomena in a steam generator (SG). The overflow model is relevant for an SG equipped with an economiser type feedwater system (i.e., the downcomer is divided into two separated legs, and the MFW injection devices are designed to send the main feedwater into the cold leg). The aim of the PIQARE object is to take into account the imperfect operation of the MFW injection devices leading to MFW overflow (i.e., MFW spilling over into the hot leg). The model has been derived for a 73/19 type SG (French reactor N4). The PIQARE sub-module is a combination of two PIQREVs:

- The main one on the DOWNCOMER cold leg.
- The other one receiving the overflow, on the DOWNCOMER hot leg.

62.4 Break modelling in case of containment calculation

To perform a CATHARE calculation of containment behaviour in case of an accidental transient involving a break flowrate from the primary circuit into the containment, the flowrates and enthalpies of the two phases at the break need to be defined as input data.

There are different ways of determining these data:

- by performing a CATHARE calculation of the transient in the primary circuit; this calculation may be coupled with the containment calculation, or may have been performed previously. In both cases, the calculation of the primary circuit provides the break flowrates and enthalpies under the coolant circuit conditions. It is then necessary to determine these flowrates and enthalpies (and the way they are broken down according to the two phases) under the containment conditions. This may be performed using the phase distribution models of the SOURCE sub-module.
- without performing any CATHARE calculation of the primary circuit, the SOURIS sub-module may be used, based on a simple model (the "RIS overflowing" model) relating the flow characteristics at the break to the safety injection conditions and to the core residual power. This model is relevant only for situations where the safety injection and the core residual power are indeed the only effects governing the break flow (typically the long-term phase of an LBLOCA). A SOURIS must be defined on the volume located at the top of the containment. It is a normal source but includes, in addition, a description of the "RIS" model.

63. HOW TO MODEL AN SGTR?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019** **-**
KEY WORD	**SGTR (steam generator tube rupture)**
See also	

A partial, single or multiple Steam Generator Tube Rupture (SGTR) may be modelled using the **SGTR operator**. This consists of a hydraulic connection between a primary side and a secondary side. It uses simply sink terms on one side and source terms on the other side between two AXIAL elements. The two axial elements can belong to two different circuits (in which case both circuits must be assigned the same components: same non-condensable gas and same radio-chemical components). They can also belong to a single circuit representing the whole reactor (in that case the IMPLICIT option must be used).

Flowrates are calculated by the code using a critical flowrate correlation or a flow characteristic function for subsonic conditions. The flowrates cannot be controlled by the user with CCVs.

The SGTR operator deals with two quite different cases:

- Partial rupture of steam generator tubes.
- Double-end break of steam generator tubes.

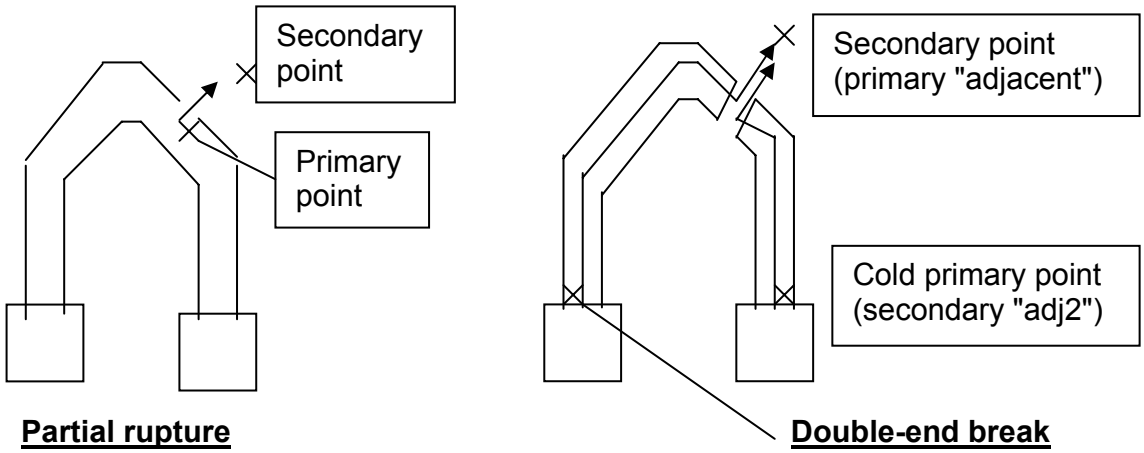
63.1 Partial rupture of steam generator tubes.

In case of a partial rupture of steam generator tubes (defined in the BLOCK DATA with the GVFUIT key word), only one flowrate is calculated at the break. The SGTR simulated in the BETHSY facility (defined with the BETHSY key word in the BLOCK DATA) is taken as a partial rupture.

63.2 Double-end break of steam generator tubes.

In case of a double-end break of steam generator tubes (defined in the BLOCK DATA with the GVGUIL key word), two flowrates are calculated:

- The first is the flowrate exiting the part of the broken tubes connected to the hot plenum of the steam generator; this flowrate is calculated at the break location on the secondary side; on the primary side, it is calculated in the AXIAL element representing the mean SG tube, at the node closest to the hot plenum (i.e., the first node).
- The second is the flowrate exiting the part of the broken tube connected to the cold plenum; this flowrate is calculated at the break location on the secondary side; on the primary side, it is calculated in the AXIAL element representing the mean SG tube, at the node closest to the cold plenum (i.e., the last node).



Partial rupture **Double-end break**

In both cases, if there are a total of n tubes in the Steam Generator and p broken tubes, the n tubes must be modelled with a single AXIAL element and the total break area calculated:

- $S_{br} = p^* S_{1\ tube}$ in case of double-ended break
- $S_{br} = p * S_{break\ 1\ tube}$ in case of partial break ($S_{break\ 1\ tube} < S_{1\ tube}$)

and the break must be located along the axis of the axial element.

There are two possible ways to define the total break area:

- either defining a maximum break area in the BLOCK DATA, and giving a percentage of maximum opening at its opening in the COMMAND BLOCK
- or defining the total break area in the BLOCK DATA.

There are two possible ways to locate the break:

- either defining a table of corresponding nodes between PRIMARY and SECONDARY sides in the BLOCK DATA (TABLE key word), and locating the break (by defining primary node) at its opening in the COMMAND BLOCK
- or locating the break in the BLOCK DATA (UNIQ key word).

At break opening in the COMMAND BLOCK an opening time can be defined (OPENBREK directive).

It is not recommended to represent two types of SG tubes:

- n-p intact tubes
- p broken tubes with an SGTR

There would be a risk of the imposed flowrate exceeding the critical flowrate in the broken tube. This would lead to code failure.

64. HOW TO MODEL SPECIFIC COMPONENTS

REFERENCE:	**DOC 3-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019**
KEY WORD	**GAS COOLED REACTOR, COMPONENT**
See also	

Specific components (core or heat exchanger, etc) can be modelled using the COMPONENT operator.

The **COMPONEN**(T) operator enables the user to define technical data describing specific hydraulic and thermal components like heat exchangers. These data will be used to calculate wall friction and/or forced convection exchange coefficients with specific correlations, which are not those used by the standard revision 6.1 of CATHARE.

In the CATHARE 2 V2.5_1 version , the different types of components are :

- the tube side of a heat exchanger : the standard correlations of CATHARE 2 are used for the computation of the wall friction and the convective heat transfer coefficient.
- the shell side of a heat exchanger : modified correlations are available for the computation of the wall friction and/or the convective heat transfer coefficient. The choice of the laws is done by the ASSIGN directive.
- the finned side(s) of a heat exchanger (PLATE and FINNED or TUBE and FINED) : modified correlations are available for the computation of the wall friction and/or the convective heat transfer coefficient. The choice of the laws is done by the ASSIGN directive.
- the bundle side of a helicoidal tube bundle heat exchanger : modified correlations are available for the computation of the wall friction and/or the convective heat transfer coefficient. The choice of the laws is done by the ASSIGN directive.
- both sides of a Printed Circuit Heat exchanger : modified correlations are available for the computation of the wall friction and/or the convective heat transfer coefficient. The choice of the laws is done by the ASSIGN directive.
- the plate-type hydraulic core : modified correlations are available for the computation of the wall friction coefficient only.

The wall friction correlations do not take into account pressure drops due to inlet and outlet structures or internal structures which are considered as singular pressure drops. They must be imposed via the **SINGULAR** directive.

The user must be aware that the modified correlations for forced convection heat exchange coefficients are mainly validated under single phase flow conditions, either liquid phase or gas phase conditions, which are those of operating and accidental transient conditions in a Gas Cooled Reactor.

65. HOW TO USE THE CATHARE DOCUMENTATION?

REFERENCE:	
KEY WORD	**DOCUMENTATION**
See also	

The CATHARE documentation includes:

- User documents
- Code description documents
- Qualification reports
- Verification reports
- Published papers, Review articles, Conference presentations, etc.

65.1 User documents:

DOC 0 –The CATHARE code: SMTH/LMDS/EM/2001-063	CATHARE main characteristics CATHARE assessment strategy
DOC 1-CATHARE 2 V2.5_1 - User Manual: SSTH/LDAS//EM/2005-035	CATHARE main characteristics Description of the DATA BLOCK (reactor, circuit, modules, sub-modules and controls in the REACTOR topology) Description of the command block Description of the stand-alone CATHACOMB fuel
DOC 2-CATHARE 2 V2.5_1 - User guide: (this document)	
DOC 3-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019	Description of the syntax of operators and directives
DOC 4 CATHARE 2 V2.5_1 - Post-processing of results: SMTH/LMDS/EM/2005-020	Description of the post processing operators and directives and the list of parameters available to plot
DOC 5-CATHARE 2 V2.5_1 - Implementation manual: SMTH/LMDS/EM/2005-017	Description of how to install the CATHARE code on WINDOWS, UNIX and LINUX platforms

65.2 Code description documents

DOC 6-CATHARE 2 V2.5_1 : Description of the base revision 6.1 physical laws used in the 1D, 0D and 3D modules : SMTH/LMDS/EM/2005-038	Description of the 1D physical constitutive laws. Description of the exact formulation of correlation. Description of certain justification elements (physical and numerical), Description of the zone indexes printed on the output listing.
DOC 7-CATHARE 2 V2.5 - Description of the 3D module: SSTH/LDAS/EM/2004-048	Description of the 3D physical models Description of the 3D space and time discretisation for all balance equations. Description of the 3D boundary conditions (walls). Description of the solution algorithm in the 3D

DOC 8-Physical improvement of the 3D module of CATHARE 2 V1.5B: SMTH/LMDS/EM/2002-084	Description of recent improvements for the 3D physical laws
DOC 9-CATHARE 2 V2.5_1 : Physical laws used in reflooding sub-module: SMTH/LMDS/EM/2005-036	Description of the physical laws used during a reflooding calculation Description of the specific physical laws of reflooding
DOC 10- CATHARE-2 V2.5 : Description of the reflooding sub-module	Description of the reflooding module
DOC 11-CATHARE 2 V2.5_1 - Description of the Volume module: SSTH/LDAS/EM/2005-037	Description and equations (internal and junctions) of the VOLUME module Numerical analysis of the VOLUME module Description of the closure equations (distribution and influence coefficients, mass transfer between sub-volumes) Description of a wall transfer in the VOLUME module Description of specific models in the VOLUME module
DOC 12-CATHARE 2 V2.5 - Description of the PIPE module and TEE sub-module: SSTH/LDAS//EM/2004-065	Basic equations of the 1D model Terms used in the discretised equations Discretised equations at a 1D point and at the TEE branch TEE closure laws Physical laws used in the TEE sub-modules
DOC 13-CATHARE 2 V2.5_1 : Description of mass source and sink sub modules: SSTH/LDAS/EM/2005-039	List of the mass and sink sub-modules (sink, source, SGTR, break) Physical equation and specific model used in mass and sink sub-modules Numerical features and discretisation Data processing
DOC 14-CATHARE 2 V2.5 - Description of the wall, heat exchanger and 0D steam generator sub-modules: SSTH/LDAS/EM/2004-049	
DOC 15-CATHARE 2 V2.5: Description of the PUMP 0D sub-module: SSTH/LDAS/EM/2004-041	
DOC 16-CATHARE 2 V2.5_1 - Description of the fuel sub-module SSTH/LDAS/EM/2005-043	
DOC 17-CATHARE 2 V2.5_1 - Description of the core kinetics: SSTH/LDAS/EM/2005-042	
DOC 18- CATHARE 2 V2.5_1 - Description of the valve sub-modules: SMTH/LMDS/EM/2005-040	Description of valve, flow limiter and check valve models Physical modelling of valve sub-module Terms added to 1D equations Specific laws used in the valve sub-module Description of valve failures
DOC 19-CATHARE 2 V2.5 - Fluid and material constitutive relationships: SSTH/LDAS/EM/2004-032	Description of steam liquid tables Description of non-condensable gases Description of wall materials

65.3 When building an Input Deck:

- ❖ Look in the User guide whether your problems are treated or not.

- Look in the User Manual for the description of the code capabilities and in order to define the information required by the code.
- Look in the Dictionary for the precise syntax of the input file.
- Look at the CATHARE data set samples.

66. HOW TO USE THE CCFL SUB-MODULE?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019**
KEY WORDS	**CCFL,FLOODING LIMIT,SINGULARITY**
See also	**HOW TO CONTROL CCFL and FLOODING LIMIT**

In the CATHARE code, the flooding equation is given by specifying the parameters M, C, E and X (Bo is the BOND number):

$$\left[J_G^* Bo^{\frac{E}{2}}\right]^X + M\left[J_L^* Bo^{\frac{E}{2}}\right]^X = C$$

66.1 CCFL at a VOLUME-PIPE junction

Using liquid and gas mass flowrate measurements from the facility, a CCFL correlation (Wallis or Kutateladze or Bankoff or Tien form according to the X and E parameters) can be determined. For a PWR upper tie plate and if the facility sections are known (core upper tie plate, S_{UTP}, and core section, S_{CORE}), a Kutateladze law is often used for a PWR upper tie plate CCFL. In this case the DIAM parameter is not used.

- If the same facility has to be modelled with CATHARE, the area restriction due to the core upper tie plate should not be modelled. The CCFL model is used at the core-upper plenum junction with the same values of the C and M parameters. The presence of the core upper tie plate is taken into account in the SRATIO parameter which is SRATIO = S_{UTP}/ S_{CORE}.
- If the same correlation has to be used in another case (extrapolation to the reactor), some assumptions are required in order to use the correlations. The C and M parameters depend on the geometry but also on the form of the upper tie plate (multi-holes or central hole or other). If the reactor form is assumed to be identical to the facility form, CCFL is also assumed to occur for the same velocity as in the facility and the C and M parameters are the same. The upper tie plate and the flow area restriction are not modelled and the parameter SRATIO = $Sreactor_{UTP}$/ $Sreactor_{CORE}$.

66.2 CCFL at an internal mesh of a pipe

A Wallis type CCFL is often used in the hot leg and at the inlet of the PWR SG-U tubes.

The CCFL at the SG U-tube inlet is able to predict the appropriate limit of liquid down flow to the core as a function of the steam up flow in a reflux condensation situation.

The CCFL in the hot leg may occur in situations without CCFL in SG U-tubes. No general correlation describing all existing data could be found. But a Wallis CCFL module can be used although the flooding limit calculated by standard momentum equations can be more severe than the CCFL module and this has to be checked if abnormal results are obtained.

67. HOW TO USE THE BOTTOM-UP REFLOODING SUB-MODULE?

REFERENCE:	**-**
KEY WORDS	**QUENCH, REFLOODING, REWETTING,**
See also	***WHAT is the REFLOODING SUB-MODULE?*** ***WHY/WHEN TO USE a REFLOODING SUB-MODULE?*** ***HOW TO MODEL CORE THERMAL-HYDRAULICS?*** ***HOW TO USE the TOP-DOWN REFLOODING SUB-MODULE?***

The bottom-up reflooding sub-module can be used with or without the top-down reflooding sub-module.

67.1 Recommendations:

- The reflooding 2d-conduction calculation must be attached to a wall or a fuel, already defined, which has a larger thermal inertia than the other thermal structures of the element. With the 3D-module, there are as many reflooding calculations as fluid columns where the 2d-conduction is attached.
- During reflooding, the other thermal structures of the element, which may be numerous when modelling the vessel with a 3D element, can be (options NOREFLAW or WREFLINK for WALL, WALL3D, FUELCHAR operators) either autonomous (standard correlations) or linked to the reflooding sub-module (the walls follow the quench front progression and the specific reflooding physical relationships are used). However, as these walls have an axial mesh similar to the standard hydraulic mesh, and not the fine mesh for 2D-conduction, the heat released after quenching will be more abrupt. This WREFLINK option is advised when modelling the shroud as in a BWR core.

67.2 Hydraulic and 2d-conduction axial meshes

See general recommendations in *HOW TO CHOOSE a MESH FOR AN AXIAL ELEMENT?* and in *HOW TO CHOOSE a MESH FOR A 3D ELEMENT?*.

The reflooding sub-module is mainly used to model the reflooding phase occurring during of a large break LOCA. In this case, the total axial length of the 2d-conduction mesh must be of the same magnitude as an axial hydraulic mesh. The 2D conduction axial nodalisation includes29 axial meshes; the quench front location is located in the 8^{th} mesh from the bottom of the 2D conduction. It is finer near the quench front location.

67.2.1 Core modelling with 1D elements

The example given in the dictionary (REFLCHAR operator) is a reference for a hydraulic mesh of length equal to 0.09 (m), i.e., corresponding to the nodalisation of a CP1 French PWR with 40 meshes, a relatively fine nodalisation which is advisable when the core is modelled with 1D elements. The total length of the 2D conduction axial mesh is about 0.09 m. The length of the 2D conduction mesh varies from 0.025 to 0.003 m from the bottom to the top and it is 0.00004 m at the quench front location.

67.2.2 Core modelling with a 3D element

The axial hydraulic nodalisation is chosen equal to 10 mesh cells. The total length of the 2D conduction axial mesh is about 0.35 m. The length of the 2D conduction mesh varies from 0.1 to 0.02 m from the bottom to the top and it is 0.00016 m at the quench front location.

For the Pericles2D qualification, a sensitivity to the axial hydraulic nodalisation (40 mesh cells) shows that 3D 10-mesh nodalisation does not converge. Nevertheless, the results obtained with 10 or 40 meshes are not very different.

67.3 Maximum time step DTMAX during a reflooding phase

See general recommendations in *HOW TO CONTROL the TIME STEP?*

As there cannot be a unique recommendation valid for all transients and all circuits, it is the responsibility of the user to check the time convergence of the calculation. This requires time-step sensitivity tests.

The same DTMAX is chosen for 3D and 1D modelling of the core (except when the time step must be low because of the CFL due to the semi implicit numerical method of the 3D module).

Gravity driven Oscillatory flowrate tests: "DTMAX=0.01 s" is recommended (e.g., PWR or integral test facility such as BETHSY or LOFT). Moreover, the "OPTION ITERMIN 6" must be chosen.

Reflooding CFL: The time step is limited by the code to 0.1 * dz / |quench front velocity|, dz being the length of the hydraulic vertical axial mesh just being reflooded.

67.4 Summary of automatic actuation of the 2d-conduction calculation

The CATHARE validation tests (e.g., oscillatory flowrate tests with a large inlet liquid flowrate, which are of great physical interest) show that the results are very sensitive to the initial location of the quench front and to the actuation time (in the V1.3L version, the actuation was controlled by the user). In V2.5, the transition between the 1D reflooding and the 2D reflooding is managed by the code (only for bottom-up reflooding). The user need only define the time of the start of reflooding (REFLOOD directive - see *HOW TO USE the TOP-DOWN REFLOODING SUB-MODULE?*).

67.4.1 Without 2D-conduction calculation (1D reflooding)

If wall temperatures are between T_{bo} and T_{mfs}, there is no axial heat conduction effect on quenching. When water enters the core, i.e., $\alpha < 0.995$, the bottom cold part of the rods is quenched without 2d-conduction calculation: The quench front position follows T_{bo}.

67.4.2 2D-conduction ACTUATION criteria:

- The quench front reaches a mesh with: Tw > Tmfs.
- There is enough water upstream of the quench front: (α < 0.995).
- No water downstream of the quench front.

67.4.3 2D-conduction STOP criteria:

When the quench front reaches a mesh such that: Tw < T_{mfs} – 125 (K) or moves too fast (quench front speed > 1 m/s) the 2D conduction calculation stops. There may be alternately 2D-conduction actuations (max 3) and 1D reflooding (max 3).

67.5 When to start the bottom-up reflooding calculation (REFLOOD directive)?

It is recommended to use the REFLOOD directive when water enters the core ($\alpha < 0.995$). In LBLOCA conditions and when the water released by the accumulator enters the core, there might be a first phase of large oscillatory reflooding (flow oscillations between core and downcomer). In this case, the conditions may be difficult for the reflooding calculation and it is possible to delay the start of reflooding after damping these oscillations (by delaying the call to the REFLOOD directive).

68. HOW TO USE THE TOP-DOWN REFLOODING SUB-MODULE?

REFERENCE:	**-CATHARE 2 V2.5_1 : Physical laws used in reflooding sub-module: SMTH/LMDS/EM/2005-036**
KEY WORDS	**QUENCH, REFLOODING, REWETTING,**
See also	***WHAT is the REFLOODING SUB-MODULE?*** ***WHY-WHEN TO USE a REFLOODING SUB-MODULE?*** ***HOW TO MODEL CORE THERMAL-HYDRAULICS?***

Compared to the BOTTOM UP reflooding sub-module, there is no automatic actuation and the positive direction is downward. So upstream means above and downstream means below. The TOP DOWN reflooding sub-module can be used with or without the BOTTOM UP reflooding sub-module.

Recommendations: Nodalisation and time step management recommendations are the same as for the bottom-up reflooding sub-module (see: *HOW TO USE the BOTTOM-UP REFLOODING SUB-MODULE?*)

68.1 Where to initialise the quench front?

The first wall temperature downstream of the quench front must be high enough. The first wall temperature upstream of the quench front must be low enough and lower than the downstream wall temperature. The code does not accept the reflooding calculation if these conditions are not satisfied:

Tw, upstream < Tsat + 100. (°C)
Tw, downstream > Tsat + 40. (°C)

But these conditions are not very severe and do not guarantee that the 2D-conduction calculation will run correctly. It is better to start with conditions such as:

Tw,upstream < Tsat + 40. (°C)
Tw,downstream > Tsat + 100. (°C)

If the conditions cannot be satisfied, consider the first criterion. If all meshes of the element have a high wall temperature, initialise the quench front elevation in the first mesh from the top.

68.2 When to start the top-down reflooding calculation (REFLOOD directive)?

There must be enough water in the mesh just upstream of the initial location of the quench front: (α < 0.995) and no water downstream of the quench front. In case of simultaneous reflooding calculations but without water top-injection, it is recommended to activate the TOP DOWN REFLOOD directive after the core begins to be BOTTOM UP reflooded, because of TOP DOWN rewetting by droplet deposition.

68.3 Which parameters limit quench front progression?

- Quench front progression is limited by a CCFL limit upstream of the quench front which is modelled by a Wallis local CCFL criterion.
- The quench front is also stopped when α,'quench front' > 1. - 10^{-4}.

69. HOW TO USE THE BCMOD COMMAND?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019**
KEY WORDS	**BOUNDARY CONDITION, MANWAY**
See also	***HOW TO MODEL a MANWAY?*** ***WHAT is a BOUNDARY CONDITION?*** ***HOW TO CHOOSE and to use a BOUNDARY CONDITION?***

The BCMOD directive may be used for:

- changing the type of a boundary condition during a transient
- changing some physical values of a boundary condition during a transient without changing the type of boundary condition.

If you want to change some physical values of a boundary condition at each time step (for example to program a regulation), put the BCMOD directive before the TRANSIENT directive in the time step loop with the appropriate new values.

In particular, the BCMOD directive is used to open and close MANWAYs.

70. HOW TO USE THE ONE-NODE PUMP SUB-MODULE?

REFERENCE:	**-CATHARE 2 V2.5: Description of the PUMP 0D sub-module: SSTH/LDAS/EM/2004-041**
KEY WORDS	**DEGRADATION (PUMP), PUMP, HEAD**
See also	

The use of the one-node pump module requires knowledge of:

- the nominal or reference head, torque, volume flowrate and rotation speed
- the motor torque, friction torque and rotor inertia
- the head and torque single-phase characteristic functions of homologous flowrate in the first three quadrants (6 octants)
- the fully degraded head and torque characteristic functions in the first three quadrants (6 octants)
- the degradation functions of the void fraction in the first three quadrants (6 octants)
- the homologous $npsh_{3\%}$ as a function of homologous flowrate ν/α or α/ν in the first quadrant (2 octants)
- the cavitation degradation as a function of the ratio $npsh_{disp}/npsh_{3\%}$.

Both 1-phase and two-phase pump characterisation tests are then required. These characteristic functions are known by the code for the LOBI pump and for the LOFT pump. They are available for the users by the key words LOBI or LOFT. The EVA, CP1, N4, P4 and RRA pump characteristic functions are also available for authorised users.

The one-node pump module can also be used with an imposed rotation speed as a function of time.

Recommendations:

- The pump characteristic functions are entered for a certain number of operating points in each octant. The code interpolates between these points. The result of this interpolation must be plotted to verify that it does not give unrealistic oscillations of the function or strong discontinuity of the derivatives at the change of octants. In such cases, it may be helpful to add operating points to force the interpolation in certain domains.
- The pump sub-module should be located at an inner vector node of an AXIAL element. It is recommended to locate it on a vertical part of the pipe (even if this is not the case in the real circuit). The reason is that, in case of stratification in a horizontal pipe, the pump model is not consistent with the low mechanical coupling condition between liquid and gas in the pipe, and this may cause convergence problems.
- It is important to respect the elevations in the pump particularly when the elevation of the diffuser prevents stratified water in the cold leg from flowing into the intermediate leg.
- Do not use a long mesh upstream of the vector node of the pump. It could entrain water from too great a distance.

71. HOW TO USE MASKS?

REFERENCE:	**0 –The CATHARE code: SMTH/LMDS/EM/2001-063**
KEY WORDS	**INPUT DECK**
See also	***HOW TO PROGRAM within CATHARE (UTILX)?***

A mask is a modified CATHARE subroutine. The language used in CATHARE and in its subroutines is FORTRAN 77.

The use of masks is under the user's responsibility.

A mask can simply be added in the calculation directory: it will be taken into account and compiled through (*cathar.unix* (UNIX) or *catharw.bat* (PC)) link and execution procedures.

The modifications of a subroutine can rapidly lead to a code failure if some basic rules are not respected:

- Before creating a variable, check that it does not already exist in the subroutine or in the included files.
- Check the number and type of arguments of the mask; they must be consistent with those of the original subroutine.
- Check the calls within the masks; the number and type of arguments of the calls must be consistent with those of the original subroutine.
- Make sure that you pre-compile the masks with the included files of the version with which it is to be linked.
- Use an IMPLICIT NONE statement before your variable declaration to avoid using wrong symbols or non-initialised variables.

Remark: For use of UTIL Fortran subroutines, see *HOW TO PROGRAM within CATHARE (UTILX)?*.

72. HOW TO USE THE COMMON USER?

REFERENCE:	**0 –The CATHARE code: SMTH/LMDS/EM/2001-063**
KEY WORDS	**INPUT DECK**
See also	

The common USER is a table used to store real or integer user variables.

The common USER may contain up to 750 double-precision reals (XUSER) and 250 integers (KUSER) and 250 character*8 (CUSER).

Warning: To be able to use the COMMON USER table in an UTILx subroutine, you should add the USER.H included at the top of the UTILx sub-routine.

72.1 Using the common user table

The common USER can be filled:

- from the input deck (WRITE)
- from an UTIL subroutine (XUSER(1) = 3794.).

It can be read:

- from the input deck (VALUE)
- from an UTIL subroutine (X1=XUSER(1)).

72.2 Post-processing common user variables

If the user wants to fill the common USER in order to be able to post-process the written values, they have to be declared as used (using the CXUSER table). This can be done:

- automatically using the WRITE directive in the input deck: for example, to use the 1 to 10 real user variables: WRITE XUSER 1 10;
- directly in UTILx.f files: for example, to use the 1 to 10 real user variables:
  Do i= 1, 10
  XUSER(i) = my_value(i) * 2.D0
  CXUSER(i) = ‘SET’
  Enddo

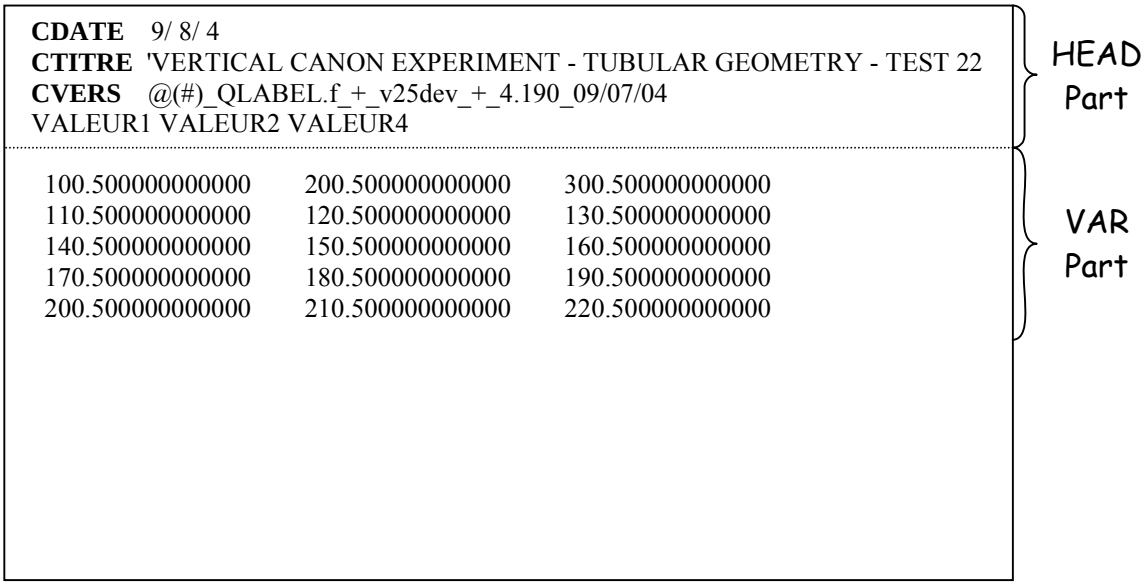
The used variables of this common table are saved in the result file (FORT21) using the REACTOR element result frequency. Changes in the variables contained in the common table can thus be plotted.

evol1 = CHRONO XUSER 1;

73. HOW TO USE A PERSONAL FILE?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019**
KEYWORD	**FILES, INPUT DECK, PRINTING**
See also	

A personal file can be used during calculation in order to store or read a list of variables. It looks like following :



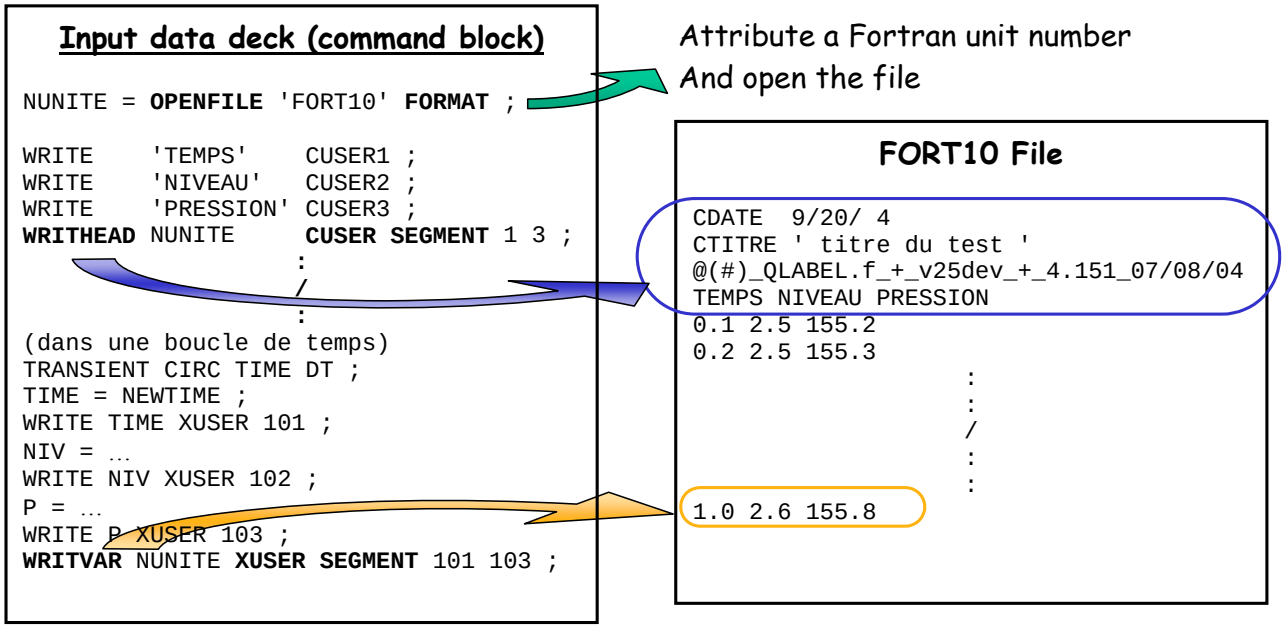
Avalilable operators and directives to manage, read and write in the personal file are :

- OPENFILE to attribute a Fortran unit number,
- READHEAD to read the heading part,
- READVAR to read the variable part,
- WRITHEAD to write the heading part,
- WRITVAR to write the variable part,
- REWIND to rewind the file and go back to the first variable line.

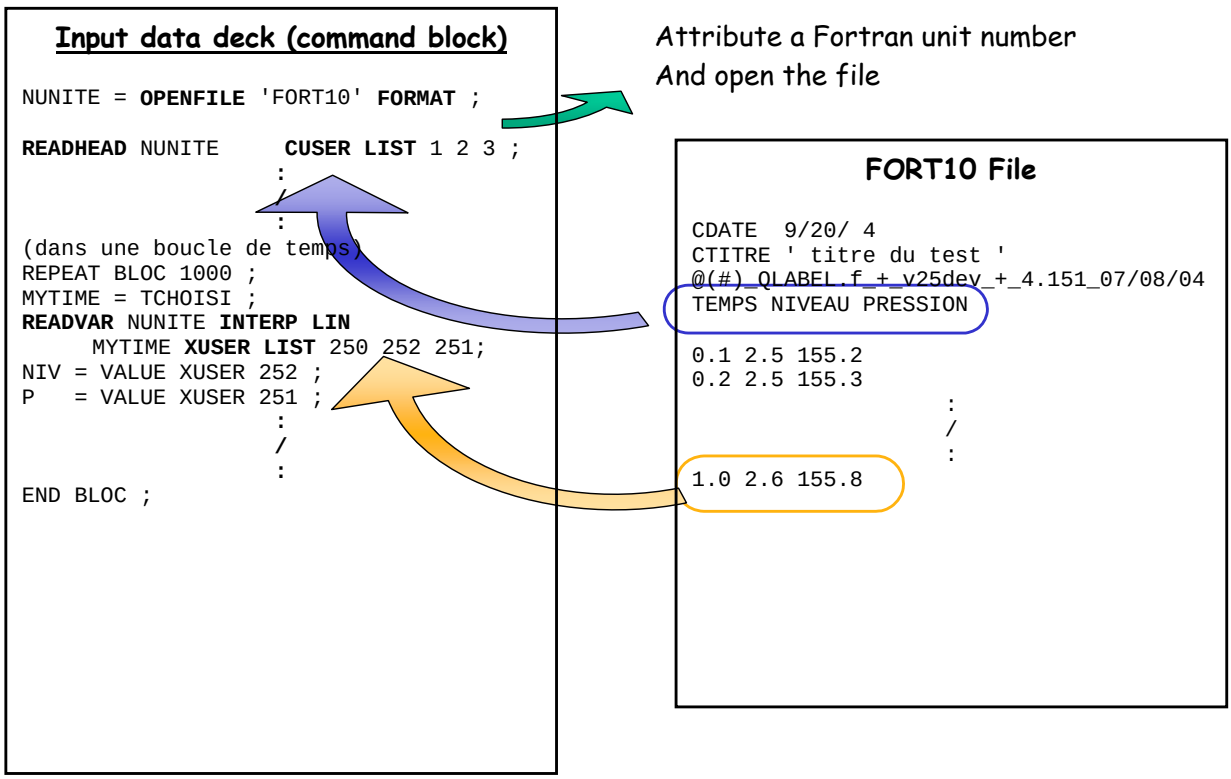
Furthermore, there are many rules to respect:

- The first column (abscissa data) must be filled as a monotone function (increasing or decreasing),
- In case of formatted file, the number of XUSER variables is limited to 10,
- It is forbidden to read or write in CATHARE standart files (FORT21, V25.INIT, …),
- It is not possible to read and write simultaneously in the same file.

The writing methodology is the following :



The reading methodology is the following:



The file is read according to the first column. The research of the target interval is done according the "xval" value of the READVAR directive (in the example, the MYTIME variable). The target values can be calculated either by linear interpolation, or logarithmic interpolation, or no interpolation (operator INTERP). In this case, the first value of the interval is chosen. The READVAR directive automatically fills the XUSER and CUSER variables.

74. HOW TO DEFINE THE GEOMETRY OF AN AXIAL ELEMENT?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019**
KEY WORDS	**AXIAL, NODALISATION**
See also	

An axial element is composed of truncated cones. Each truncated cone is defined by:

Nodalisation data:

- length L
- cosine of the angle with the vertical upward direction

Geometrical data:

- two cross-sections S1, S2 : SECT (m^2)
- friction perimeter PERI (m)
- scale length SIZE (m) used only in case of annular hydraulic geometry

The volume of the truncated cone is defined by:

$$V=L\ [\ S1+S2+(S1.S2)^{0.5}]\ /\ 3$$

Sometimes it is not possible to respect elevations, volumes and curvilinear abscissa along the mean flow. In that case it is generally recommended to **respect the volume** (important for stored mass and energy) **and the elevations** (important for all the gravity-driven flow situations).

Never give excessively abrupt area changes, it is better to smooth them. And do not forget to give singular pressure drops associated with the area changes.

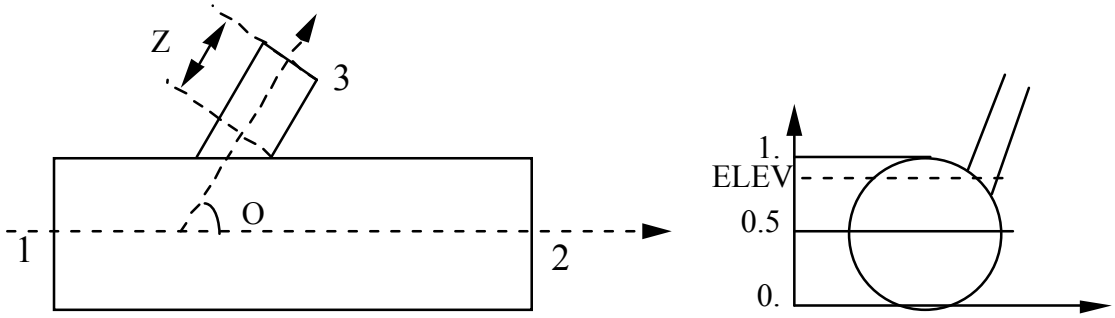
Note: the minimum number of meshes in an axial element is two.

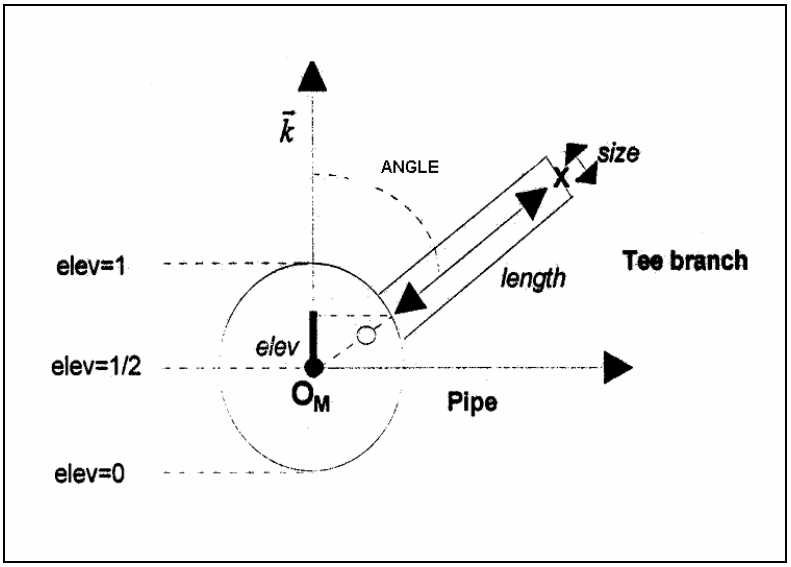
75. HOW TO DEFINE THE GEOMETRY OF A TEE?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019** **-CATHARE 2 V2.5 - Description of the PIPE module and TEE sub-module: SSTH/LDAS//EM/2004-065**
KEY WORD	**TEE**
See also	***WHAT is a TEE?***

A Tee is a lateral branch which has to be linked with an axial element (directive CONNECT). For the main pipe and the lateral junction, we define (directive GEOM):

the length calculated from the edge of the main pipe	**LENGTH (m)**
the tee section	**SECT (m^2)**
the friction perimeter	**PERI (m)**
the tee diameter	**SIZE (m)**
the standardised elevation of the tee branch with respect to bottom of the main pipe	**ELEV (m)**
the angle between axial element and tee branch	**ANGLE (radians)**

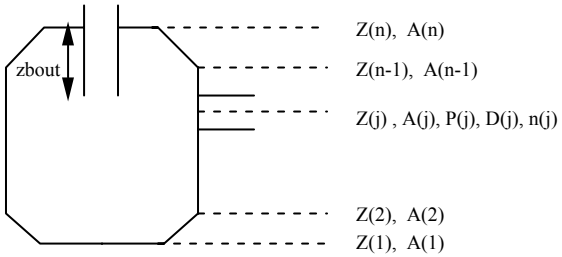




For the branch length, it is recommended to take half of the length of the first vector mesh of the adjacent element.

76. HOW TO DEFINE THE GEOMETRY OF A VOLUME?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019** **-CATHARE 2 V2.5_1 - Description of the Volume module: SSTH/LDAS/EM/2005-037**
KEY WORDS	**VOLUME, PRESSURISER, UPPER PLENUM, LOWER PLENUM, SG CHANNEL HEAD, PWR (pressurised water reactor), BWR (boiling water reactor)**
See also	***WHAT is a VOLUME?***



Data for the volume: zmax: total height

couples (Z(i),A(i)) : elevation and cross-sectional area (cylindrical or conical)

For each junction:

Z(j) elevation (m)

A(j) cross-sectional area (m^2)

P(j) wetted perimeter (m)

D(j) size (m)

n(j) weight factor

zbout(j) length of penetration

76.1 Hydraulic diameter in the volume

A hydraulic diameter is calculated in the volume and used in the calculation of heat transfer coefficients, which are themselves used in calculating the energy and mass transfers between both sub-volumes and with the walls.

The hydraulic diameter is the average diameter from bottom to top of the volume on the basis of the cross-sections A(i), assuming a circular shape. Depending on the actual shape of the volume and on the presence of internal structures, this default value of the hydraulic diameter may be far too big. The key word **HYDRDIAM** can be used to enter another value. An excessively large hydraulic diameter may give, for example, too low a heat transfer coefficient with the internal or external walls.

A **junction** can be **top** (coming from the top)**, bottom** (coming from the bottom), **horizontal** (connected on the side) or **inclined**. Junctions may penetrate into the volume; in this case, it is necessary to give the length of penetration.

76.2 Length of penetration:

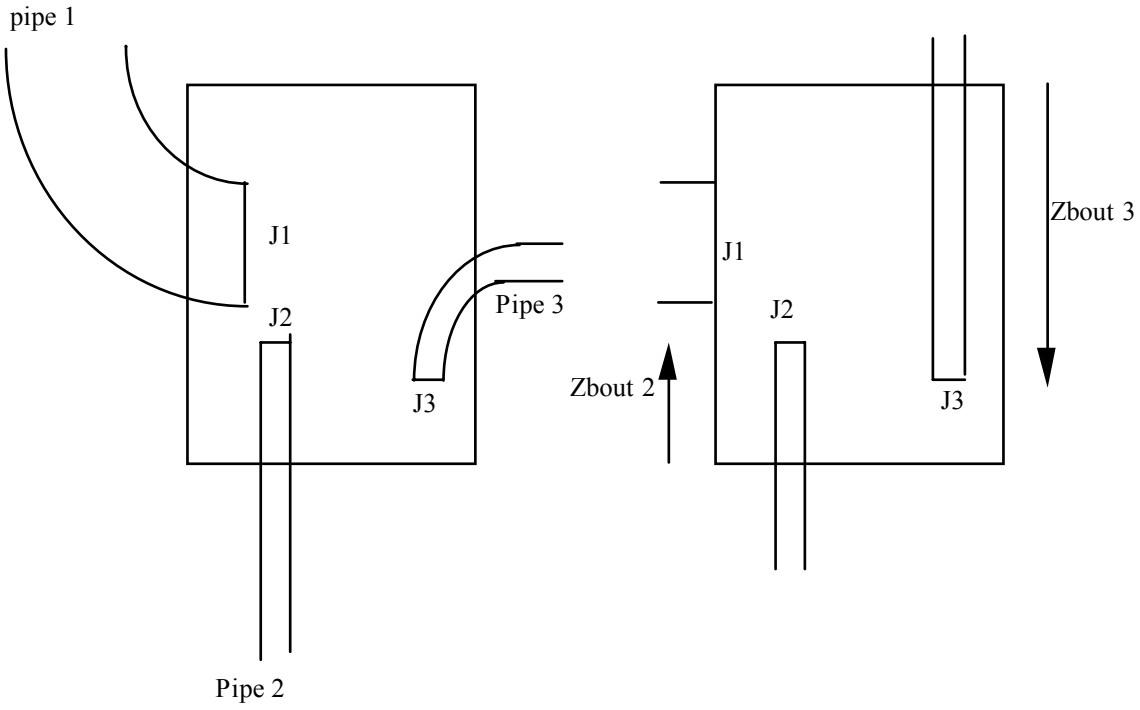
- for a vertical top junction the length of penetration corresponds to the vertical distance from the top of the volume to the level of the junction.
- for a vertical bottom junction the length of penetration corresponds to the vertical distance from the bottom of the volume to the level of the junction.
- for a horizontal junction it is pointless defining a length of penetration as all thermal-hydraulic parameters are assumed to be uniform in a horizontal plane.

- for an inclined junction, a length of penetration can be input. CATHARE will take into account the penetration length projection on the vertical axis.

In the example below:

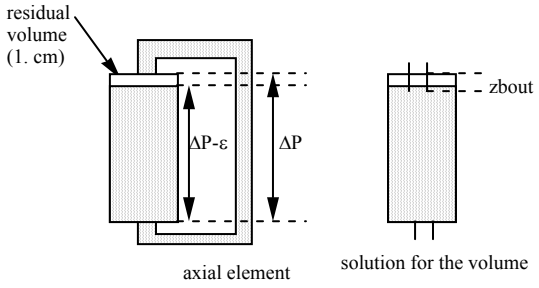
- J1 is a horizontal junction without length of penetration
- J2 is a vertical bottom junction with a length of penetration Zbout2
- J3 is a vertical top junction with a length of penetration Zbout3

The parts of the pipes inside the vessel must be represented in their geometry and mesh arrangement.



76.3 Recommendation about the residual level effect

Consider a circuit with a single-phase liquid and natural circulation. The upper sub-volume is reduced to the residual value (height = 1mm). The code will introduce a ΔP due to this lack of water in the circuit. In a configuration such as that which follows, this ΔP may induce flow circulation in the circuit and the user will not be able to reach the quiescent state with zero velocity. Solution: for the junction the user gives a length of penetration which leads the junction inside the lower sub-volume. In practice this error is often very small, except when the dimensions of the circuit are not very large compared to 1mm.



76.4 Length of influence of a junction

For a horizontal junction on the side of a volume, the influence zone calculated and used in CATHARE ($1.2*D_{junction}$) for this junction should be out of the residual sub-volume (1 cm for the upper sub-volume and 1 mm for the lower sub-volume).

For a vertical leg on the top or the bottom, this limitation is imposed automatically.

76.5 Specific phase flow distribution at the junction

If the volume is a **PWR upper plenum**: the **UPPLEN** option should be used (derived from SEROPS experiment). For an **upper plenum without internal structure** (BETHSY) the **UPPLENWS** option should be used. Both options modify the phase flow distributions at the junctions.

For an **annular downcomer-lower plenum junction** the **ANNULSPA** option can be used. It more accurately predicts the formation of voids in the lower plenum at the end of blowdown during a LBLOCA. A specific law, derived from the analysis of the PIERO experiment, is used.

For **a PWR steam generator separator-dryer** junction, the **SEPDRYER** option must be used. It puts all the liquid in the lower sub-volume and generally all the vapour in the upper sub-volume (only if the level is above the junction, a fraction of the vapour may be put in the lower sub-volume). The result is that the fluid entering this junction provides only dry steam to the upper sub-volume. It plays the role of the separator but not of the dryer. In any case, at the junction between the steam dome and the steam line, there will only be steam except in two cases: first there can be some condensation in the upper sub-volume by depressurisation or by wall cooling. Secondly, if the water level rises near the steam line, there may be some liquid of the lower sub-volume pulled through the steam line.

For a **BWR vessel separator-dryer junction**, specific options must be used (**SEPINLET, SEPSIDE**, **SEPOUT, MINDRYER and DRYER,** see *HOW TO MODEL a SEPARATOR and a DRYER?***)** to impose the carry over, the carry under of the separator and the liquid quality of the dryer. In this case, the phase flow distribution at the junction takes into account the separator and dryer behaviour.

76.6 Specific mass and energy transfer model

76.6.1 PRESSURISER model

Specific modelling of mass and energy transfer between both sub-volumes and wall and of flow distribution at the junction must be used in case **of PWR PRESSURISER** by using the **PWRPRZ** option (see *HOW TO MODEL a PRESSURISER?*). This option is used to take into account the behaviour of the spray (no liquid breakdown at the wall), the heaters and the enlarged energy transfer at the interface of both sub-volumes.

The **PRSRIZER** option is used to increase the interface exchange in the **PRESSURISER** during steady-state calculations, in order to reduce the time constants associated with pressuriser dynamics (speed-up of the steady-state calculation).

76.6.2 Film condensation model

A specific heat transfer from walls to the film in case of film condensation mode can be used, in particular when the reactor containment is modelled with VOLUME modules. In this case and using the VFILM directive, users can choose different correlations from the film condensation with or without non-condensable gases. Types of available correlations are NUSSELT, USHIDA or COPAIN laws.

77. HOW TO DEFINE THE GEOMETRY OF A 3D ELEMENT?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019** **-CATHARE 2 V2.5 - Description of the 3D module: SSTH/LDAS/EM/2004-048**
KEY WORD	**THREED**
See also	***WHAT is a 3D ELEMENT?***

The GEOM directive is used to describe the 3D module in terms of fluid volumes and surfaces. Either these data are directly calculated and introduced in the input deck by the user, or the notions of porosity, both volumic and surface, are used.

The scalar equations (mass and energy balances) are integrated on the scalar mesh volumes, according to the finite volume approach. Therefore, they directly depend on the fluid volumes and fluid surfaces, contrary to the momentum equations which are discretised according to the finite difference approach.

Indeed, the main application is the modelling of a PWR vessel which contains a large number of solid structures such as rod bundles, spacer grids, etc. An approach similar to the porous media modelling has been retained. Therefore, some volumetric and surface porosities can be defined, giving respectively the relative volumes occupied by the fluid and by the structures in each mesh, and the available areas for the fluid flowing from one mesh to neighbouring ones. These porosities are calculated according to the chosen nodalisation network and are imposed in the input deck by the user.

It is highly recommended to use these two notions of volumic and surface porosities for the following reasons:

- to minimise the amount of data in the input deck
- to think in terms of average data in the description of the 3D module. For most of the applications (PWR vessel modelling) large meshes are used and the average flow is described. Therefore, there is no point in describing the geometry precisely.

The vessel representation is thus based on zones such as the lower plenum, the core, the downcomer, etc., and each zone is schematised. All the plates (core upper plate, vessel lower plate, etc.) do not have to be exactly described in terms of fluid surface but singular pressure drops must be imposed. The point is not to introduce excessively big singularities in the flow description because they may introduce numerical problems directly related to the modelling process: a strong reduction in fluid section will very likely result in a considerable increase in velocity, unfortunately non-physical. In such case, it is recommended to impose an average fluid section and the corresponding singular pressure drop.

Volumic porosity

The volumic porosity ε_v is defined as follows:

$$\varepsilon_v = \frac{V_{fluid}}{V_{mesh}} = 1 - \frac{V_{structures}}{V_{mesh}}$$

with V_{fluid}: the fluid volume of the mesh cell

V_{mesh}: the geometrical volume of the mesh

$V_{structures}$: the volume of the structures located in the mesh cell

Surface porosity

The surface porosity ε_s is defined as follows:

$$\varepsilon_s = \frac{S_{fluid}}{S_{mesh\_face}} = 1 - \frac{S_{structures}}{S_{mesh}}$$

with S_{fluid}: the fluid surface of the mesh cell face

S_{mesh_face}: the mesh cell face geometrical area

$S_{structures}$: the area of the structures located in the mesh cell face

78. HOW TO DEFINE THE CONNECTIONS OF A 3D ELEMENT?

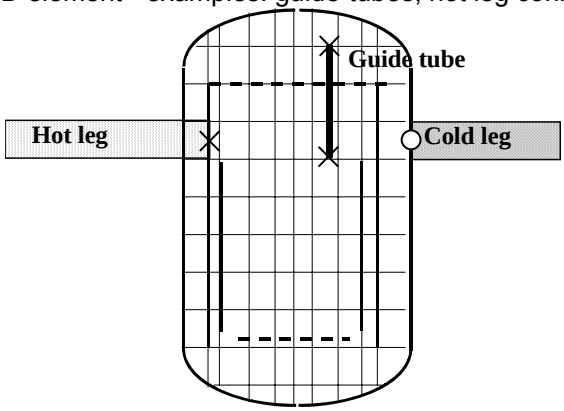
REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019** **-CATHARE 2 V2.5 - Description of the 3D module: SSTH/LDAS/EM/2004-048**
KEY WORD	**THREED**
See also	***WHAT is a 3D ELEMENT?***

The CONNECT operator defines the connections between the 3D hydraulic mesh of the element and the other elements of the circuit. It allows the type of mesh face (key word EDGETYPE): closed or open to be introduced. All the junctions must be defined before using EDGETYPE.

78.1 Definition of junctions:

The 3D element can be connected to the other hydraulic modules of CATHARE (0D, 1D, BC). Algorithmically, 3D-3D junctions can be used, but this is not recommended (it has not been thoroughly tested, and besides, in most cases, it is not likely to be worthwhile in terms of CPU time). The junctions are based on the standard 10 principle variables map (p, hl, hg, α; vl, vg; p, hl, hg, α). Remember that the junctions are strictly one-dimensional, i.e., the 3D information is lost in the connection. For instance, a single 3D element is not equivalent to the same element divided into two 3D elements connected with junctions. One junction can only be connected to one cell face (i.e., one junction cannot be distributed over several 3D adjacent meshes, which implies that the 3D nodalisation in the vicinity of a junction cannot be refined below the size of the junction). Two kinds of junctions are available:

- **external (standard) junctions**: used when the adjacent element is connected to a mesh edge located at the boundary of the 3D element
  example: connection of the cold leg
- **internal junctions**: used when the adjacent element is connected to a mesh edge located inside the 3D element - examples: guide-tubes, hot leg connection



x: internal junction
o: external junction

Two options are available for internal junctions:

- **open**: means that the mesh edge where the junction is located is open for fluid flow. Example: guide tube junctions

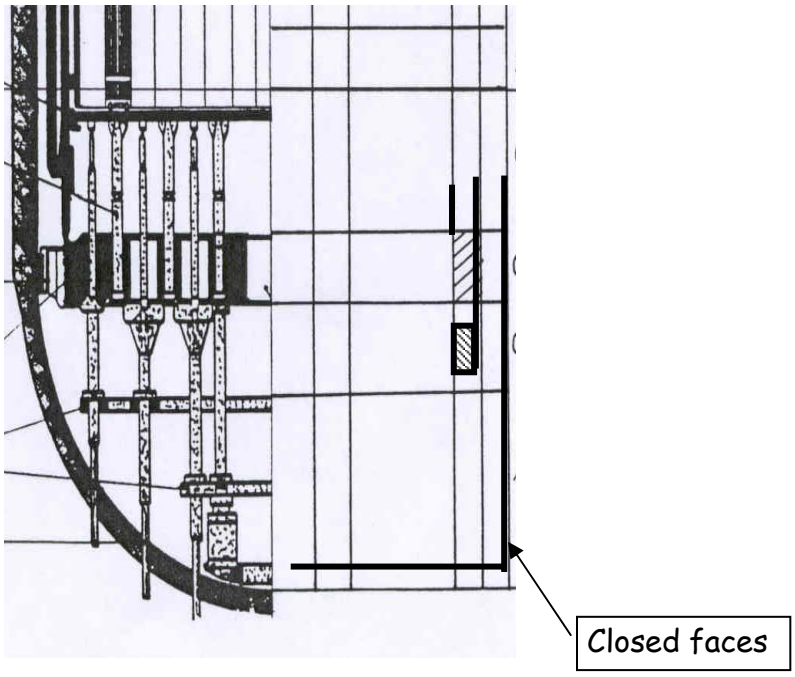
- **closed**: means that the mesh edge where the junction is located is closed, no fluid flow can occur through this mesh face except for the junction. Example: hot leg junctions

78.2 Definition of mesh type:

The key word EDGETYPE is used to define the contours of the 3D flow domain and the internal obstructions by means of open/closed faces:

- OPEN faces: fluid can flow through
- CLOSED faces: faces are declared as an impermeable wall in terms of fluid flow

Remember that the use of a closed face is not equivalent to a face with zero-porosity (i.e., fluid surface close to zero). In the first case, all the mass and energy transfers are equal to zero and the velocities are not calculated but forced to zero. In the second case, the mass and energy transfers are equal to zero too, but the velocities are still calculated and their values can be non-physical, due to the very small fluid section.



79. HOW TO CHOOSE A NODALISATION FOR A 3D ELEMENT?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019** **-CATHARE 2 V2.5 - Description of the 3D module: SSTH/LDAS/EM/2004-048**
KEY WORDS	**THREED**
See also	***WHAT is a 3D ELEMENT?***

The MESH directive is used to define the nodalisation of a 3D element. Therefore, the system of coordinates (**COORDINA**) must be chosen, then the number of meshes (**NODALISATION**) in each direction must be defined, the orientation of the system (**GRAVITY**) must be specified and finally the size of the mesh cells must be listed for each axis.

COORDINA: Two kinds of mesh are available: Cartesian (x,y,z) and cylindrical (θ,r,z). When the geometry to model is typically annular, then the key word ANNULAR can be used. This refers to a cylindrical system of coordinates and the first internal radius (rint) must be specified. The **COORDINA** adapted to the modelled domain can be chosen. For instance, for the vessel, a cylindrical **COORDINA** can be used. For an annular downcomer, an annular **COORDINA** can be used.

GRAVITY: In the current version, the z-axis is assumed to be vertical; the only degree of freedom is the orientation: upward (GRAVITY Z –9.81) or downward (GRAVITY Z +9.81).

MESH DEFINITION: The 3D module has been developed to fill one main objective, the modelling of a PWR vessel for LBLOCA calculations. Most of the meshes are considered to contain a large number of solid structures and no diffusive terms are taken into account. The closure laws are qualified from the analysis of full scale experiments and the quality of predictions may rely on compensating errors which may be mesh size dependent. In such case, there is no possible mesh convergence and the module must be used with a reference mesh (See How to model the 3D vessel?).

A non-exhaustive list of recommendations for mesh definition is proposed. It is mainly deduced from experience:

- The mesh size should be progressive (large mesh size gradients should be avoided as much as possible), a ratio around 1.5 between two adjacent connected meshes (not separate by a closed face) is a good base but very often, because of geometrical constraints, this ratio is very difficult to respect.
- The porosity should not vary too drastically from one cell to the next. There is little point attempting to describe the exact geometry and all the internal structures but, on the other hand, zones with average porosities should be considered.
- When there is no geometrical constraint to the mesh size definition in one direction, it might be advantageous to look at the walls to be defined. If the walls are homogeneously distributed over that direction, the mesh size should be defined in order to get iso-volume meshes. Then the calculations of wall characteristics (HSURF) will be faster.
- A junction can only be connected to one 3D mesh cell.
- It is recommended to define in the MESH directive all the zones that will be used to describe the 3D element.

If a given number of radial meshes is desired in one particular region (for instance the core), this implies specifying the same number of radial meshes in the entire vessel.

⇒ **this may lead to an excessively large nodalisation**

The 3D module – 0D/1D module junctions are highly CPU time consuming and it is advisable to minimise their number as much as possible. For instance, if the physical phenomena to calculate are typically 2D, it is recommended to use the 3D module in terms of a 2D module with only one mesh cell in the width.

80. HOW TO CHOOSE A NODALISATION FOR AN AXIAL ELEMENT?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019**
KEY WORDS	**BEND, BREAK, REFLOODING**
See also	

Main recommendations: For AXIAL modules, the nodalisation must converge in size.

A nodalisation converges in size when the result of the prediction does not change - or does not change too much - when the mesh size is reduced (or when the number of meshes is increased).

The tolerance interval for mesh convergence (residual quantitative modification of the prediction when reducing mesh size) may depend on the physical situation to simulate. For example, the prediction of a break flowrate can be considered as converged when the flowrate is within a range of + or - 1% of the asymptotic value obtained for very small meshes. But a higher accuracy may be required if the transient events are very sensitive to the break flowrate. There is not any general rule to apply for ensuring mesh convergence. **Mesh convergence is the responsibility of the code user** who should perform the appropriate sensitivity tests for each physical situation in order to check mesh convergence.

The required mesh size to obtain convergence also depends on the physical situation to simulate. Some indications can be found in the qualification reports which cover a large domain of flow conditions.

A non-exhaustive **list of recommendations** for the nodalisation deduced from the experience of the code is proposed hereafter:

- The length ratio between two successive meshes should not exceed 1.2 (1.5 may be accepted)
- A bend in a vertical plane (such as the bends of an intermediate leg) should be modelled using at **least 3 mesh cells** with varying inclination angles. This rule was obtained for two-phase situations in the ECTHOR Loop seal tests. However, if the flow remains liquid single phase during a transient, a rougher nodalisation may be adequate.
- At a volume-pipe junction, the first mesh of the pipe should be approximately:
  - 2% of the volume height for a vertical junction
  - 10% of the volume hydraulic diameter for a horizontal junction
- At a tee-pipe junction, the first scalar mesh of the pipe (distance from the junction to the 1st scalar point) should be approximately 100% of the distance from the junction and the edge between the main pipe and the branch of the tee.
- For a core reflooding calculation using the reflooding sub-module, the mesh size should not exceed 0.1 metre. (See *HOW TO MODEL CORE THERMAL-HYDRAULICS (1D MODELLING?*)).
- For a break pipe, the presumed critical section should use a fine mesh (not more than 1 mm). The critical section is generally the last downstream section of the pipe segment with the smallest cross-sectional area. (See *HOW TO MODEL a BREAK?*).
- If choked flow is expected to occur somewhere within a pipe, the presumed critical section should use a fine mesh (not more than 1mm).
- Heat exchangers with a two-phase level on one side should use fine meshes close to the two-phase level. A rough mesh in this zone would lead to an overestimation of the heat exchange area with the liquid phase.

81. HOW TO CHOOSE THE HYDRAULICS OF AN AXIAL ELEMENT?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019**
KEY WORDS	**HYDRAULICS, ROD BUNDLE, ANNULAR**
See also	***HOW to model CORE THERMAL-HYDRAULICS?*** ***HOW TO MODEL a DOWNCOMER?***

The type of hydraulics is defined for an axial element.

- **PIPE** type: the fluid is flowing inside a simple pipe or a duct without internal structures (default value)
- **RODBUNDLE** type: the fluid is flowing around rods or tubes: electrical or fuel rods in a core, tube bundles in the secondary side of a steam generator.
- **ANNULAR** type: the fluid is flowing in an annulus: downcomer

An axial element can have different “hydraulics” according to the location.

This choice has an effect on:

- the interfacial friction: it is generally lower in a rod bundle or an annulus than in a pipe with the same hydraulic diameter.
- certain heat transfer laws between walls and fluids. In particular, the critical heat flux is higher in pipes than in rod bundles.

82. HOW TO CHOOSE THE HYDRAULICS OF A 3D ELEMENT?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019** **-CATHARE 2 V2.5 - Description of the 3D module: SSTH/LDAS/EM/2004-048**
KEY WORDS	**HYDRAULICS, ROD BUNDLE, ANNULAR**
See also	***WHAT is a 3D ELEMENT?***

Two types of hydraulics are available, for both scalar (SCAGRID) and vector (VECGRID) correlations:

- ***standard flow***, for cells with some internal structures, also called nearly open domain (for instance lower plenum, upper head, downcomer)
- ***rod bundle*** *flow*, for cells with lots of internal structures, also called porous domain (for instance the core).

This choice has an effect on:

- the interfacial friction: it is generally lower in a rod bundle than in a nearly open domain with the same hydraulic diameter;
- certain heat transfer laws between walls and fluids. In particular, the critical heat flux is higher in nearly open domain than in rod bundles.

For an annular downcomer, a specific model (DCOGRID) has been developed based on a UPTF downcomer refill test assessment. This choice has an effect on:

- the void fraction discretisation for the interfacial friction term,
- the entrainment rate: the entrainment is delayed in terms of onset of entrainment V_0
- the liquid to interface heat transfer: condensation rate is higher.

The user must check the consistency of the declared hydraulics. Each zone of a 3D module should be defined with one and only one type of hydraulics. For instance, the core should not be declared as ***standard*** for the scalar nodes and as ***rod bundle*** for the vector nodes.

Therefore, considering the modelling of a PWR vessel with the 3D module, it is recommended to declare:

- core active part: rod bundle flow
- annular downcomer: specific model (DCOGRID)
- lower plenum, bypass, upper head: standard flow
- upper plenum: rod bundle flow

83. HOW TO CHOOSE THE PHYSICAL LENGTH SCALES OF A 3D?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019** **-CATHARE 2 V2.5 - Description of the 3D module: SSTH/LDAS/EM/2004-048**
KEY WORD	**THREED**
See also	***WHAT is a 3D ELEMENT?***

The physical correlations used for the 3D module are directly extrapolated from the axial correlations. For the momentum equations, they have been extended to the three directions. Similar physical correlations are used to describe an open space with nearly no internal structure or, on the contrary, a zone very much filled with internal structures like the core. Therefore, the choice of the physical scales used in the physical correlations is very important and depends on the nature of the flow domain and the type of hydraulics defined in the input deck.

1. without any internal structure (open domain)
2. filled with lots of internal structures like the core (porous domain)
3. filled with some internal structures like the lower plenum, the upper head (nearly open domain)
4. annular space like an annular downcomer

Some recommendations can be listed depending on the flow domain.

83.1 Open domain, i.e., no or nearly no solid structures

The volume hydraulic diameter is used in the scalar correlations, mainly q_{le}, q_{ve}, q_{pl}, q_{pi}, q_{pv} and both the hydraulic diameter and the droplet diameter are of importance, depending on the flow. When the flow is assumed to be mainly dispersed, the driving parameter is the droplet diameter. It is then advisable to impose a large volume hydraulic diameter (for instance $D_h \approx 1000$).

The edge hydraulic diameter is used in the vector correlations, mainly the wall friction correlation and the interfacial friction correlation. For an open domain, the wall friction term calculated by CATHARE is not well adapted especially for the internal meshes with no wall contact. In the absence of any diffusion term (molecular or turbulent), the wall friction term helps to stabilise the calculations, but its value is small. Therefore it is recommended to impose an arbitrarily large edge hydraulic diameter to minimise the wall friction term (for instance $D_h \approx 1000$). It is important to underline that this does not affect the consistency of the interfacial friction. In that case, what limits the bubble maximum size is the LAPLACE scale.

For an open domain with dispersed flow, it is recommended to impose a high hydraulic diameter, for both the surface and the volume hydraulic diameter (for instance $D_h \approx 1000$).

83.2 Porous domain, i.e., numerous internal structures: the core.

In this case, the use of 1D correlations for the scalar correlations as well as for the vector correlations seems quite legitimate. Therefore, the hydraulic diameter, volume and edge, are calculated for each mesh following the same logic as for a 1D element. But attention must be paid to the fact that both the volume and the edge hydraulic diameters should be determined at the level of a homogeneous zone, rather than distinctly for any individual cell, in order not to induce numerical singularities (see ref. [2]).

The *volume hydraulic diameter* is computed as

$$D_{hv}(i) = 4\,\frac{V_{fluid}(i)}{S_{friction}(i)}$$

where $V_{fluid}(i)$ is the scalar mesh fluid volume and $S_{friction}(i)$ is the friction area inside the scalar mesh (or average values of V_{fluid} and $S_{friction}$ within a homogeneous zone).

The *edge hydraulic diameter* is computed for each face of the cell as follows, whenever possible:

$$D_{hs}(i) = 4\,\frac{S_{fluid}(i)}{P_{friction}(i)}$$

where $S_{fluid}(i)$ is the flow section and $P_{friction}(i)$ the friction perimeter (or average values within a homogeneous zone). When it is not possible, the volume hydraulic diameter is imposed.

83.3 Nearly open domain, i.e., some internal structures: the lower plenum, upper head

The main idea is to consider the domain as totally homogeneous without any empty mesh (without internal structure). The objective is to calculate the global flow characteristics of such a domain and not to be mesh-dependent. Therefore, both hydraulic lengths, volume and edge lengths, are computed considering global quantities: total fluid volume $V_{fluid,tot}$, total friction area $S_{friction,tot}$,

$$D_{hv}(i) = D_{hs}(i) = 4\,\frac{V_{fluid,tot}}{S_{friction,tot}}$$

83.4 Specific case of the annular downcomer

Both volume and surface hydraulic lengths are computed equal to twice the annular downcomer width e:

$$D_{hv}(i) = D_{hs}(i) = 2e$$

84. HOW TO CHOOSE AND TO USE BOUNDARY CONDITIONS?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019**
KEY WORDS	**NONCONDENSABLE GAS, BLIND, BOUNDARY CONDITION, GUILLOTINE RUPTURE, SAFETY VALVE**
See also	***WHAT is a BOUNDARY CONDITION?*** ***HOW TO USE the BCMOD command?*** ***HOW TO CHOOSE and to use a BOUNDARY CONDITION?***

The **MODEL** directive defines the type of boundary condition element.

The **BCMOD** directive modifies the type of boundary condition (see *HOW TO USE the BCMOD command?*).

A boundary condition element imposes some of the principal variables (P,HL,HG,α,X1,X2,X3,X4,VL,VV) at the inlet or outlet of an adjacent element. These values can be:

- values calculated by CATHARE: SAFETYVA,
- user-managed values as a function of physical time: BC3X, BC3EX, BC4X, BC5X, BC5XX, BC5YY, BC5ZZ, BC5EX, BC5HO, BLIND.

84.1 Inlet boundary condition

For an **inlet boundary condition** (the flow is entering the adjacent element), use BC3X or BC3EX or BC5XX or BC5YY.

- **BC3X (BC3A, BC3B,..., BC3G)** will impose time-dependent functions for HL, HG, α, (X1,X2),VL, VV.
- For **BC3EX**, the variables are not time-dependent, but they are imposed by CCV.
- **BC5XX, BC5YY** and **BC5ZZ** can be used for inlet or outlet conditions (see below).

Recommendations:

- Never use BC3X, if one velocity (or both velocities) must change direction, since only incoming velocities are allowed.
- For single-phase flows, impose the saturation condition for the residual phase and equal velocities for both phases.
- Be careful with BC3F. As no velocity nor mass flowrate is imposed, you must be sure that the velocity will not change direction. If a flow reversal could possibly occur, choose a BC5XX, BC5YY or BC5ZZ.
- Avoid conditions with subcooled steam or superheated liquid.

84.2 Outlet boundary condition

For an **outlet boundary condition** (the flow is going out of the adjacent element), use RUPTURE or BC4X or BC5X or BX5XX or BC5YY or BC5ZZ or BC5EX, BC5HO or BLIND or SAFETYVA

- **RUPTURE** simulates a double-ended guillotine break. Before the break opening, it behaves as a pipe-pipe junction. After the break opening it behaves as two BC4 conditions and the outside conditions, pressure, temperature and non-condensable mass fractions are given as a function of time (see *HOW TO MODEL a BREAK?*).
- **BC4X** (resp **BC5X)** will impose the pressure as a function of time.
- **BC4X, BC5X** allow flow reversal of one or both phases.
  For **BC4A, BC4B and BC5C** and in case of flow reversal of one or both phases, the incoming

values of HL, HG, α, X1, X2 are taken equal to the values calculated at the last time step with outgoing velocity.
For **BC4C and BC5B**, only incoming gas flow is possible (α=0.999999 on the last scalar node).

- **BC5XX** or **BC5YY** do the same as BC5X but the values of all parameters can be given as function of time. The value of HL and α will be imposed only in case of reversal of the liquid velocity. The values of HG, X1, X2, X3 and X4 will be imposed only in case of reversal of the gas velocity.
- **BC5ZZ** is similar to the BC5XX but also allows the counter current flows when the liquid flow comes in the boundary condition and the gas flow goes out.
- **BLIND** will impose the values of VL and VV for outgoing velocities. It keeps constant the values of the velocities calculated by the PERMINIT operator but these values can be changed during a transient by applying a BCMOD directive.
- **SAFETYVA** (*HOW TO MODEL a SAFETY VALVE?*) models a safety valve and imposes an outgoing flowrate according to predefined characteristics of the valve (opening pressure, pressure for complete opening, steam capacity of the valve, water capacity of the valve, etc.) and according to upstream conditions (void fraction, pressure).

Recommendations

- Use BC4X for a break if the boundary condition is located at the possible choked flow section (downstream end of the smallest cross-section area of the nozzle). Do not forget to use the required fine mesh just upstream of a BC4(X) boundary condition '(see *HOW TO CHOOSE a NODALISATION FOR AN AXIAL?*).
- Use BC4A or BC4C for a break opening at time zero.
- Use BC4B for a break opening at time different from zero.
- Use BC5A, BC5B, BC5XX, BC5YY or BC5ZZ to impose the pressure at the downstream end of a pipe or in a circuit.
- BC5X is used to regulate the pressure of a pipe or a circuit and to absorb the expansion of the fluid in a circuit due to temperature changes. Then a BC5 boundary condition located somewhere in a closed loop may represent in a simple manner the functions of the pressuriser and the volumetric control circuit.
- BC5XX, BC5YY and BC5ZZ are the most general and most powerful boundary conditions:
  - They can be used as inlet or outlet conditions.
  - They allow flow reversal of one or both phases.
  - All imposed parameters can be given as a function of time.
- Use a BLIND condition for a dead end of a pipe. In this case, the velocities calculated during the steady state must be very small (a NOFLOW condition is required upstream of the BLIND element in the PERMINIT directive).
- A BLIND condition may be used to limit in a simple manner a water level in a tank modelled by a volume module. Connect a BLIND element at a horizontal junction in the VOLUME module at the elevation of the desired water level. Impose a high exiting liquid velocity and a very small gas velocity. If the level in the volume is below the BLIND leg, the gas flow will be zero (as VV=0) and the liquid flow will be zero (as α = 1). If the water level reaches the BLIND leg, the gas flow will be zero but the liquid flow will be high as long as the level crosses the junction. This is not exactly a level regulation. If the level is below the junction, the BLIND will not increase the level.

85. HOW TO CALCULATE A GAS SINGLE PHASE FLOW?

REFERENCE:	
KEY WORDS	**AIR, NONCONDENSABLE GAS, HYDROGEN, NITROGEN**
See also	

A gas single phase flow calculation can be performed using the **MONOPHASE** directive.

In this case, the liquid residual phase is set in thermal and dynamic equilibrium with the gas phase, i.e. saturation temperature at vapour pressure, an liquid velocity equal to the gas velocity. The void fraction is set to the maximum value ($1.\text{-}10^{-6}$) and the OD (VOLUME) levels are set to a residual value 10^{-3} m.

No more liquid mass, energy and momentum balances are computed. No water mass and/or energy and impulsion injections are allowed by SOURCE, PIQREV, PIQVANNE, PIQSOUP, PIQSEB, ACCU, BC3x, BC4y or BC5yy or CANDLEs. The use of REFLOOD or CCFL models are then not allowed.

The **MONOPHASE** directive can be used several times for a same element or for different elements during a computation. This, for instance, allows the user to first initialize a circuit CIRC1 under gas condition (MONOPHASE circ1 GAZ), then perform a stabilized transient under the same conditions, then re-activate the two phase flow computation (MONOPHASE circ1 OFF) in order to enable water injection again.

86. HOW TO CALCULATE USING NON-CONDENSABLE GASES?

REFERENCE:	**-CATHARE 2 V2.5_1 : Description of the base revision 6.1 physical laws used in the 1D, 0D and 3D modules : SMTH/LMDS/EM/2005-038** **-CATHARE 2 V2.5 - Fluid and material constitutive relationships: SSTH/LDAS/EM/2004-032**
KEY WORDS	**AIR, NONCONDENSABLE GAS, HYDROGEN, NITROGEN**
See also	***HOW TO CALCULATE a GAS SINGLE PHASE FLOW?***

CATHARE is able to simulate liquid water and gas mixture two-phase situations. The gas mixture is composed of steam plus up to four non-condensable gases.

For a computation with non-condensable gases, you have to define (for each circuit) the list of non-condensable gases used with the operator NONCOND (the list is then specified in the CIRCUIT operator). Six non-condensable gases have their physical properties predefined in CATHARE (helium, nitrogen, oxygen, hydrogen, argon, air), but others can be user-defined.

The model of the gas mixture transport properties can be chosen by the user (see below). The specific heat at constant pressure may be input by the user for each non condensable gas either as a constant value or as a function of the gas temperature (POLYNCPT option for the NOCOND operator).

As soon as you have chosen to make the calculation of the circuit with n non-condensable gases, all the constituents are always present, even in residual form. In particular, steam is always present in the gas phase. A gas single phase flow (without liquid) calculation may be performed (see *HOW TO CALCULATE a GAS SINGLE PHASE FLOW?*).

To initialise the circuit or when you have fluid flowing in the circuit, you have to give data concerning the gas mixture.

Initialisation is performed with the PERMINIT and REALC directives.

Inlet flow conditions are encountered in certain boundary conditions (BC3.., BC5XX, BC5YY, BC5ZZ, BC5HO type), in SOURCE or certain types of SINK.

When providing data concerning the gas mixture, caution has to be taken for the non-condensable gas mass fractions and the gas mixture enthalpy (or temperature).

86.1 MASS FRACTIONS:

If you declare **n** non-condensable gases ($1 \leq n \leq 4$), you have to determine the mass fraction of each of the i^{th} non-condensable gas X_i. The total mass fraction of non-condensable gas is: $X_{NC} = \sum_{i=1}^{n} X_i$, and the mass fraction of steam is: $1 - X_{NC}$

Each non-condensable gas is considered as a perfect gas with: $\rho_i = \frac{P_i}{R_i.T}$

Steam is always present in the gas mixture, even as a residual constituent. So, do not use $X_{NC} = 1$ (which would imply that the vapour pressure is zero). The saturation temperature of the vapour T_s should be chosen such that: $0°C < T_L \leq T_s \leq T_G$. The partial vapour pressure is: $P_v = P_{sat}(Ts)$. The value of X_{NC} is then computed as:

$$X_{NC} = \frac{P - P_v}{P - P_v + RT_G\rho_V} \quad \text{with:} \quad R = \frac{1}{X_{NC}} \sum_{i=1}^{n} X_i.R_i$$

If there is no liquid (in fact only as a residual phase), the liquid temperature tends to be equal to the saturation temperature ($T_L = T_s$). As the liquid temperature (the liquid enthalpy, in fact) is limited to a value close to 0°C, avoid imposing T_s close to this limit. If you want to simulate saturated (or 100 % humid) non-condensable gas, $T_s = T_G$ should be chosen.

Example 1: air-water tests: declare air (user-defined) as N.C. gas and choose $T_s = T_L = T_G = T_{exp}$. At atmospheric pressure and 20°C, this corresponds to **$X_1 = 0.987$**.

Example 2: gas (helium) cooled reactor: declare **HELIUM** as non-condensable gas, choose T_S such that: 0°C << Ts << TG. For instance for P = 70 bar, T_G = 600°C, if you choose Ts = 50°C you find : **$X_1 = 0.992$**

86.2 GAS ENTHALPY:

The enthalpy of the gas mixture is computed as:

$$H_G = (1 - X_{NC}).H_V(P_V, T_G) + \sum_{i=1}^{n} X_i.H_i$$

where H_i is the specific enthalpy of the i^{th} non-condensable gas. The origin of the non-condensable gas enthalpy is redefined by CATHARE (and may be different to what is found in handbooks of gas properties) in the following way:

$$H_i = H_o + Cp_i.(T_G - T_0)$$

with the constants $H_o = H_{Vsat}$(7 bar) = 2766432 J/kg and $T_0 = T_{sat}$(7 bar) = 164.93°C. The heat capacity Cp_i of each non-condensable gas is taken constant.

Considering the complexity in the computation of H_G in the previous equation, it is (strongly) advised to give the gas temperature T_G instead of H_G. This is always possible.

86.3 GAS TRANSPORT PROPERTIES

The value of transport properties used during calculation depends on the mixture model chosen by user. By default the following standard models are used for the gas mixture viscosity and conductivity computation:

$$\mu_G = \sum_i y_i \mu_i \quad \text{and} \quad \lambda_G = \sum_i y_i \lambda_i$$

where y_i and μ_i (respectively λ_i) stands for the molar fraction and for the pure gas viscosity (respectively pure gas conductivity) of each species.

Specific models are also available:

- Wilke's model for gas mixture viscosity computation :
- Mason et Saxena's model for gas mixture conductivity computation :
- An average model, based on a weighting coefficient Q, for gas mixture conductivity computation : $\lambda_G = (1-Q)\left[\sum_i y_i \lambda_i\right] + Q\frac{1}{\sum_j \frac{y_j}{\lambda_i}}$

87. HOW TO INITIALISE A DEADZONE?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019**
KEY WORDS	**DEAD ZONE, VALVE**
See also	

87.1 What is a DEADZONE?

A DEADZONE is generally a part of a circuit isolated by closed CONTROL VALVES in order to have no flow zones. A DEADZONE can contain several CATHARE elements or can be a part of an AXIAL element.

A DEADZONE is used to initialise, with the PERMINIT module, certain isolated parts of a circuit such as a developed ACCUMULATOR (modelled with an AXIAL element and a VOLUME element) or the RRA sub-circuit.

A DEADZONE is defined in the PERMINIT directive.

87.2 DEADZONE initialisation

A DEADZONE is initialised automatically by the steady state calculation (directive GOPERM) as well as the rest of the circuit: the non-isolated zone.

In order to initialise a DEADZONE and calculate the steady state, it is necessary to impose the thermal-hydraulic conditions at one point only in the isolated zone. This can be done:

- in a VOLUME element of the DEADZONE by the directive REALVO;
- at a specified point of an AXIAL element contained in the DEADZONE by the directive REALAX.

As usual the user must use the REALC directive at certain circuit junctions to initialise the non-isolated zone (“flowing zone”).

87.3 Recommendations for the use of DEADZONES

- For the calculation of the steady state, a DEADZONE must imperatively be bound by closed CONTROL VALVES. These CONTROL VALVES may be opened for the TRANSIENT. They can also be disabled and certain CHECK VALVES defined at the same points can be enabled to model an accumulator for example.
- NOFLOW conditions must be defined at the junctions which separate the DEADZONE and the flowing zone.
- The two phase velocities imposed by the REALAX directive must be very small (example 10^{-6} or 10^{-10} m/s).
- The thermal-hydraulic conditions imposed in a VOLUME element must be as close as possible to an equilibrium state between liquid and gas sub-volumes.

88. HOW TO INITIALISE COLD/HOT FUEL DATA?

REFERENCE:	
KEY WORDS	**EXCHANGER, FUEL, HEAT LOSS,HEAT SOURCE, NEUTRONICS, WALL**
See also	***HOW TO MODEL CORE NEUTRONICS?*** ***HOW TO MODEL FUEL RODS?*** ***WHY/WHEN TO USE THE STAND ALONE FUEL?***

The cold fuel data – clad radii, gap pressure - are generally known from experimental reports for instance. But in fact, many CATHARE transients start in hot temperature conditions.

Two methods are then possible to reach the initial hot fuel steady state:

- The cold data are entered in the input deck (FUEL and FUELCHAR operators) and a transient calculation from a cold state to a hot state is simulated. This method is recommended for simple test cases like separate effect tests.
- For a reactor-like calculation, it is recommended to compute the core or vessel part from a cold state to a hot state first, and then, to feed the data deck of the reactor calculation with the previous initialisation results of the core or vessel.

89. HOW TO PROGRAM REGULATIONS?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019** **-CATHARE 2 V2.5_1 - User Manual: SSTH/LDAS//EM/2005-035**
KEY WORDS	**REGULATIONS, INPUT DECK**
See also	

Regulations may be programmed in the input deck (see the user manual for input deck examples) for the following purposes:

- to represent actual regulations of a reactor or an experiment,
- to obtain a stabilised initial state (or true steady state).

They need:

- knowledge of certain parameters or thermal-hydraulic quantities (pressure, temperature, etc.); these values may be obtained by the CCVs
- knowledge of the set point or operating point to reach
- a way of acting on the system in order to reach the set point. This requires the use of trips, interventions, modifications of the boundary conditions, etc. This may be achieved by using the standard directives of the code.

A general expression for PDI (proportional-differential-integral) regulations is the following:

$$\tau \, dA/dt + A = G \, (Fop - F) + T \, d(Fop - F)/dt$$

A is the action or modification of the controlling parameter

F is the controlled value

Fop is the operating value of F (or set point) to be reached

G is the gain of the proportional action

τ is the integration time constant

T is the differential time constant

Most of the time, T is taken equal to zero to avoid problems such as high frequency oscillations of F and A.

Example: level regulation in a Steam Generator by a source or a sink of saturated water.

❖ **How to choose the gain G?**

- Estimate the sensitivity of F to A: S = dF/dA
- Take G < A G= S/3 seems to be OK

❖ **How to choose the time constant τ ?**

- Estimate the physical time constant of the system τ_ϕ: time necessary to obtain the result of the action.
- Take $\tau > \tau_\phi$ $\tau = 1.5 \, \tau_\phi$ seems to be OK

If you use several regulations together, use the largest time constant for all the regulations.

❖ **How to choose the time step dt?**

- The time step should not be too large.
- Take dtmax $< \tau_\phi$ dtmax $= 0.2\,\tau_\phi$ seems to be OK

Last recommendation: If you are using a regulation to reach a steady state, when the operating point is reached, check that it is a steady state and that the action is zero: stop the regulation, continue the calculation and check that all variables are constant and that the time step is increasing.

90. HOW TO PROGRAM WITHIN CATHARE (UTILX)?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019**
KEY WORDS	**INPUT DECK**
See also	***HOW TO USE MASKS?***

It may be interesting for the user to program CATHARE to perform auxiliary calculations with CCVs, to print out certain results in other files for special post-processing.

To perform these operations, the user should use UTIL1 … UTIL20 directives. Each of these directives is equivalent to calling the corresponding UTIL1.f, UTIL2.f,... UTIL20.f subroutines.

Remarks on UTILx.f subroutines:

- No argument can be put in their definition.
- The language used in CATHARE is FORTRAN 77.
- Use an IMPLICIT NONE statement before your variable declaration to avoid using wrong symbols or non-initialised variables.

If you want to use other files (or Fortran units), take care not to open a file used by CATHARE (the list of Fortran units used by CATHARE is given in the User Manual).

In the Dictionary of operators and directives, you can find for each useful directive the corresponding Fortran subroutine with explanations about its arguments.

Example: (extract from the VALUE directive in CATHARE 2 V2.5)

VALUE for DOUBLE PRECISION type variables:

- Zval = VALUE (KEYWOR, CNAME, IMESH, IRAD, IVSTAT)

IVALUE for INTEGER type variables:

- Ival = IVALUE (KEYWOR, CNAME, IMESH, IRAD, IVSTAT)

KEY WORD	**CHARACTER*8 name of the variable to be read**
CNAME	**CHARACTER*8 name of the element**
IMESH	**INTEGER mesh number – 0 if no mesh number, 1 for lower sub-volume, 2 for upper sub-volume**
IRAD	**INTEGER 0 if no meaning, else radial mesh number for walls, or radio-chemical number for hydraulic components**
IVSTAT	**INTEGER error code**

Thus the equivalent of the following command in an input deck:

P2 = VALUE PRESSURE pipe1 2 ;

is the following line in a Fortran subroutine:

P2 = VALUE (‘PRESSURE’, ‘pipe1 ‘, 2, 0, IERROR)

91. HOW TO PROGRAM SIGNALS, TRIPS AND ACTIONS?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019**
KEY WORDS	**INPUT DECK, CCV, OPERATOR ACTION, INTERPOLATION**
See also	***HOW TO USE MASKS?***

Signals may be programmed using the CATHARE Calculation Variables (CCVs) and manual or automatic actions or trips may be programmed with directives.

For example if the High Pressure Safety Injection (HPSI) is modelled with a source and the HPSI flowrate depends on the primary pressure, the actuation of the HPSI is programmed as follows:

- The VALUE operator is used at each time step to get the value of the primary pressure. This value is stored in a variable declared before in the input deck
- A test is performed to determine if the HPSI has to be opened or not. Generally, this test is composed of two parts:
  - a physical part: the pressure is compared to the set point for the HPSI actuation specified in the input deck
  - a logical part: an index representing the state (open/closed) of the HPSI is compared to a value required for the action. In this example, the HPSI may be opened exclusively if it is closed.

If the two parts of the test are true, the following actions are performed:

- the HPSI source is opened with the directive OPEN
- the index value is changed so that it represents an open state.

At each time step and when the HPSI is open, the mass flowrate is calculated using the pressure and imposed on the LIQFLOW parameter of the source with the WRITE directive. Note that the action may be delayed as the pressure is read only at the end of the time step. If the pressure changes rapidly, it is necessary to use small time steps.

An interpolation tool (operator INTERP) may be used in the command block to impose parameter in boundary condition module, source or sink to impose either experimental or user data. The interpolation may be linear or logarithmic.

Specific signals can be also used in case of fuel kinetics for the axial profile of the residual power and the distribution coefficient of the neutronic power (XNEULISX and PRESHAPX).

92. HOW TO CHECK THE ENERGY AND MASS BALANCE?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019**
KEY WORDS	**MASS BALANCE, ENERGY BALANCE, PRINT**
See also	***HOW TO CONTROL PRINTING?***

The CATHARE code is designed to minimise energy and mass balance errors:

- Energy and mass balance equations use the conservative form and use a time and space discretisation which strictly complies with the energy and mass conservation condition. Minor errors come from the tolerance of convergence tests.
- Junctions between elements respect the equality between the incoming and outgoing energy and mass fluxes.
- Thermal structure calculations ensure energy conservation through the calculation of the heat exchange between the walls and the hydraulic.

The use of the 3D module may give slightly higher mass errors because of semi implicit discretisation (see What is an implicit code?).

The printing of mass errors is no longer valid after the use of a WRITE directive which reinitialises a variable in the circuit (other than source flowrate and source enthalpy).

The INIBIL directive allows the user to consider the variation in balances from a new starting time in an entire circuit or a reactor (re-initialisation of balance reports). The time of the INIBIL directive actuation becomes the new initial state and reference state for the cumulated balances.

Information on the mass and energy balances can be found in the output listing (using PERIOD directive), using CCV (see WRITE and VALUE directives) and in result file (see dictionary of post-processing).

A special ZONE element has been designed to make the balance checks easier to perform. It allows the user to assemble elements and get all mass and energy information available as if it was a single element (see “What is a zone?”)

Using any of these methods, information is also available for the circuit and for any of its modules (AXIAL, VOLUME, THREED or ZONE) (and gadgets) and also walls (for energy balances).

In the output listing, for each module printout (see *HOW TO CONTROL PRINTING?*”), you will find:

- the instantaneous mass and energy balances,
- the instantaneous energy balances of each wall and the wall overall balance.

In the output listing, for each circuit printout, you will find:

- the instantaneous mass and energy balances,
- the instantaneous energy balances of the walls
- the cumulated mass and energy balances of each element and of the overall circuit (since the last initialisation),
- the cumulated energy balances of each wall and the wall overall balance (since the last initialisation).

The description of all this information is given in the User Manual

Information about radio-chemical balances is also available for each module, zone or circuit. This consists of a mass balance for the boron and an activity balance for radio-active components and gives the same information: instantaneous balance for each module, instantaneous and cumulated balances for the zones or the circuits.

In calculation using a 3D module, a specific directive “DTMASS3D” can be used to limit the time step increasing when the global mass error is over a value imposed by the user.

93. HOW TO CONTROL CCFL AND FLOODING LIMIT

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019**
KEY WORDS	**CCFL,FLOODING LIMIT,SINGULARITY**
See also	**HOW TO USE THE CCFL SUB-MODULE?**

CATHARE has the capability of predicting the CCFL (Counter-Current Flow Limitation) phenomenon and a flooding limit. This is the result of the different forces modelled in the two momentum equations (gravity terms, interfacial friction, acceleration terms, wall friction terms, singular pressure drop terms, etc.). The result depends mainly on the validity of the interfacial friction model which is used. As interfacial friction correlations are generally obtained from quasi-established flows, they are no longer valid in complex geometries such as abrupt area changes, diaphragms, bends, etc.

It has been observed that a frequent source of mispredictions is associated with CCFL at the core upper tie plate, at the hot leg bend and at the inlet of steam generator tubes:

- CCFL at the core upper tie plate is generally too severe when it corresponds to a PIPE-VOLUME junction (even if the flow restriction is not represented).
- CCFL at the hot leg bend is found also to be slightly too severe (MHYRESA tests).
- CCFL at the inlet of steam generator tubes is not severe enough (BETHSY tests)

In any case, the predicted flooding limit depends on the nodalisation:

- ❖ mesh size (smaller meshes generally give more severe limitations)
- ❖ a large mesh above a small mesh results in a more severe flooding limit at the vector node between the two meshes than with two equal size meshes.
- ❖ a small mesh above a large mesh results in a less severe flooding limit at the vector node between the two meshes than with two equal size meshes.
- ❖ representation of flow restrictions or not in the geometry (representation of flow restrictions generally give more severe limitations)
- ❖ presence of a VOLUME-PIPE junction. If the volume is above the pipe, the limitation may be too severe.
- ❖ presence of a singular pressure drop (gives more severe limitations).

A specific local model is required to control CCFL and the flooding limit in complex geometries. Such a model is available in CATHARE V2.5 for the VOLUME-PIPE junctions (with the volume above the pipe, for instance, at the core upper tie plate) and for any mesh of a pipe (for instance, when SG channel heads are modelled with a pipe).

94. HOW TO CONTROL PRINTING?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019**
KEY WORD	**PRINT**
See also	

The printing of results in an output listing is controlled by several directives.

- PERIOD directive is used to control the printout of the physical state of elements. The printing frequency of calculated variables can be defined, separately for each element and for junctions either every n time steps or every n seconds of physical time. There is a hierarchy notion: for example, all elements of a circuit inherit its period of printing.

For example, if a period T1 is input for element E1 belonging to circuit C1 and a period TC1 for circuit C1, the smaller period between T1 and TC1 will be used for printouts of E1.

There is no possibility of printing results every n meshes of an element.

For each element, information on its mass and energy balance verification can be found at the end of its personal printout,.

- IMPRIME directive does the same as PERIOD but once, and for all circuits and their elements.
- PRIN3D directive is used in a chosen plan to print out once chosen variables for a 3D element
- LIST directive is used to print out the definition (topology, nodalisation, geometry) of any modules and any sub-modules. There is no period of printing associated with this directive.
- MESSAGE directive can be used by the user to print out any character string of his convenience (to spot an event in the listing for example) and any variable value accessible from the input deck. There is no period of printing associated with this directive.
- VERBOSE directive can be used to print out information about element convergence or THREED element geometrical variables.

It can be used to analyse the origin of certain numerical problems, such as a time step decrease.

Use the VERBOSE directive to print the Jacobean matrix and/or the internal variable increments of a module.

Remark: Some messages or information are automatically printed and cannot be deleted.

95. HOW TO CONTROL THE TIME STEP?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019**
KEY WORDS	**TIME STEP, CONVERGENCE, OPTION**
See also	***HOW to explain a TIME STEP DECREASE?***

The time step is controlled by the code, but can be controlled and changed by the user from the input deck. After each time step, CATHARE proposes a value for the next time step DT. If the convergence has been difficult to reach, the next DT will be smaller and if the convergence has been easy, the next DT will be larger. The systematic use of the time step proposed by the code does not guarantee that the calculation converges with time.

The maximum and minimum values are imposed with the OPTION directive:

- OPTION DTMIN dtmin; (default value 1.d-5 s)
- OPTION DTMAX dtmax; (default value 1000.s)

The code will use and propose values of dt between dtmin and dtmax. If it cannot converge with dtmin it fails and stops.

For any transient time step the user may choose to calculate with his own value DT. If the circuit calculation does not converge, the code will decrease DT as many times as necessary; 10 successive time step decreases are allowed by default; this number may be modified by the OPTION MAXREP directive.

In the case of several circuits, the calculation of all circuits is performed with the same DT. If a secondary circuit needs a smaller time step than the time step proposed by the primary circuit, two or more sub-time steps are done in order to reach the primary time step. If the secondary circuit needs too low a time step, the code starts a new calculation of the primary with a smaller DT (automatic restoration of the values of the previous time step).

If there were time step reductions, the clock at the end of the time step takes into account the change of time which is actually reached.

Recommendations for DTMAX

For a good accuracy and time convergence of a transient calculation, it is recommended to limit the time step.

DTMAX must be small for rapid transients:

--> use DTMAX=0.01 second for a LARGE BREAK LOCA.

DTMAX can be larger for a slow transient. It cannot be very large when there are secondary circuits in case of explicit coupling. There may also be some trips actuated explicitly during the transient: a DTMAX of 1 or 2 seconds may be recommended for SMALL BREAK LOCA.

As there cannot be a unique recommendation valid for all transients, it is the user's responsibility to check the time convergence of the calculation. This requires time step sensitivity tests.

The maximum time step can be managed by the user with :

- The FUELDTMX directive, in case of fuel calculation, in order to control the time step before and after the clad rupture.
- The DTMASS3D directive in order to maintain the time step to it’s preceding value when the global mass relative error in a 3D module is over the user’s imposed value

96. HOW TO EXPLAIN A TIME STEP DECREASE?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019**
KEY WORDS	**TIME STEP, CONVERGENCE, OPTION**
See also	***HOW TO CONTROL TIME STEP?***

A time step decrease is the result of a convergence difficulty of the iterative solution method. It may be due to:

- a rapid change in the physical state somewhere in the system:
  - ❖ passage from two-phase to single phase
  - ❖ mass quality of steam or of a non-condensable gas reaching an extreme value
  - ❖ passage of a criterion for changing the flow regime,
  - ❖ passage of a criterion for changing the wall heat transfer regime (critical heat flux, rewet temperature, etc.)
  - ❖ any situation creating strong condensation
  - ❖ actuation of a source, an accumulator, opening a valve, etc.
- a code error:
  - ❖ derivative error
  - ❖ discontinuous equations
  - ❖ discontinuous term in an equation
  - ❖ discontinuous derivative
  - ❖ bad numerical conditioning, etc.
  - ❖ error in a mask used for a sensitivity test
- a specific directive activated as:
  - ❖ FUELDTMX, with its criteria to control the time step before and after the clad rupture
  - ❖ DTMASS3D to maintain the time step to it's preceding value when the global mass relative error in a 3D module is over the user's imposed value

  Try first to identify the origin of the problem before calling the maintenance team.
- allow very small time steps (with the OPTION DTMIN xxx, with xxx<1.10-6) and a large number of successive time step reductions (OPTION MAXREP 30) and see if it solves the problem.
- if the code fails to converge with a very small time step, locate the problem:
  - ❖ determine the element which has the most difficult convergence problem. Automatic failure messages can be found in the printouts or created using the VERBOSE directive. Using the VERBOSE directive in particular, you can access the expected next time step for each element (VERBOSE reactor GENERAL 1), and the number of iterations used by each element for each time step (VERBOSE reactor STAT 1). The element which had the largest number of iterations (before satisfying convergence criteria) and which requires the smallest time step is the element where the problem lies.
  - ❖ Perform a restart calculation with additional printed information. Use VERBOSE ELEM1 CONWR 1 or 2 to get information on the increments of all variables at each iteration. Use VERBOSE ELEM1 JACOWR 1 or 2 to get information on the Jacobean matrix of the element. These items of information can be used to identify the variable experiencing convergence difficulties and the equation experiencing convergence difficulties.

  Give this information to the maintenance team for further analysis.

97. HOW TO REDUCE THE COST OF A CALCULATION

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019**
KEY WORDS	**TIME STEP, CONVERGENCE, OPTION**
See also	***HOW to explain a TIME STEP DECREASE?***

The cost of a calculation (CPU time) depends on the elementary time, the number of junctions, the number of nodes, the number of iterations per time step and the value of the time step.

The elementary time is the average CPU time for one iteration and one node. It depends only on the numerical method, the code structure and the optimisation of the coding.

The number of junctions defines the size of the assembling matrix to be solved for each iteration and each circuit. The larger the number of junctions, the greater the time required to solve this assembling matrix.

The number of nodes must satisfy the recommendations for the nodalisation (see *HOW TO CHOOSE a NODALISATION FOR A PIPE?* Or *HOW TO CHOOSE a NODALISATION FOR A 3D ELEMENT?*). This defines the size of the elimination matrix of an element. The larger the number of nodes, the greater the time required to solve this elimination matrix.

The number of iterations per time step is controlled by the code and depends on the convergence of the iterative Newton method. An average value of about 5 or 6 may be expected when the time step is controlled by the code itself. It may be lower if the user gives a low maximum time step.

The value of the time step is controlled by the code and depends on the convergence of the iterative Newton method. The maximum time step allowed may be reduced by the user according to certain requirements (*HOW TO CONTROL TIME STEP?*).

The user does not have many possibilities for reducing the cost of a calculation. Only a few recommendations may be given:

- Do not define unnecessary junctions or nodes
- Do not use OPTION ITERMIN or at least, use it with a smaller minimum number of iterations
- Use OPTION ROCP during steady state to accelerate the computation of a stabilised state
- Do not use the THREED module when it is not necessary. This module is rather expensive.

In case of a parametric study concerning fuel rods, the CATAFUEL directive (Why/When to use the stand alone fuel?) is the most efficient way to reduce the cost of the calculation.

Remark: Significant decreases in time step or an increasing number of iteration are signs of a convergence difficulty (see How to explain a time step decrease)

98. HOW TO PERFORM SENSITIVITY STUDIES?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019**
KEY WORDS	**TIME STEP, CONVERGENCE, OPTION,SENSITIVITY STUDY, NODALISATION**
See also	

Sensitivity studies can be **useful** in different cases:

- Time convergence study
  - to assume that a stabilised state is reached
  - to find the right limitation for the time step.
- Nodalisation convergence study
  - there is not any clear criterion to choose between several possible schematisations or nodalisations; the different possibilities must be tested
  - the nodalisation convergence must be verified.
- Physical study
  - one closure law is not well qualified in a certain domain of parameters; the uncertainty of the code response to this closure law must be quantified
  - to see the effect of certain parameters of the calculation which are not very well known
  - to better understand the physical processes and to see what is responsible for what in a calculation.

98.1 How to perform a time sensitivity study?

Facilities are given using the OPTION directive to control the minimum number of iterations for each time step (ITERMIN) or the maximum value of the time step (DTMAX).

Time convergence is reached when a reduction in the maximum time step (DTMAX) does not change - or does not change too much - the results of the prediction. (*HOW TO CONTROL TIME STEP?*

There is an easy way to determine whether a **steady state** has been reached, taking advantage of the fact that CATHARE is an implicit code. Just apply constant boundary conditions and let the time step increase by itself. When a large time step is reached (100 sec or better 1000 sec) the steady state is reached.

98.2 How to perform a study of sensitivity to nodalisation or schematisation?

Modify the input data and start a new calculation from the steady state. All tested schematisations must respect the User Guide recommendations.

98.3 How to perform a closure law sensitivity study?

- The main physical law parameters are equipped with sensitivity coefficient CCVs. The list of available sensitivity coefficient CCVs is given in the Dictionary (see WRITE directive).

  For these parameters, the easiest way to change their value is to use the WRITE directive. Moreover, this allows the user to modify a parameter not necessarily for the whole circuit but only for selected elements and not necessarily for the whole duration of the calculation but only for selected time steps.

It is also possible to change the value of these parameters in the initialisation Fortran subroutines, using masks. In this case, the parameters are modified for the whole circuit and for the whole duration of the calculation. The initialisation subroutines are:

HPRSPRI.f for hydraulic scalar laws,

HPRVPRI.f for hydraulic vector laws,

FECPRI.f for wall to fluid heat transfer laws.

❖ For the physical law parameters which are not equipped with sensitivity coefficients, their values can be changed by modifying the Fortran subroutine defining the physical laws (with masks). The derivatives of the closure law with respect to the principal variables must also be modified accordingly in the Fortran subroutine. The most important closure laws are in the subroutines listed below (with 'r6' or 'r61' extensions when using revisions 6 or 6.1 of physical laws - revision 6.1 is default value). The 'te' extensions may also be found and mean “extended water table”.

…/hydcom/griscalar for hydraulic scalar laws,

…/hydcom/grivector for hydraulic vector laws,

…/wallcom/fecpar for wall to fluid heat transfer laws,

…/volume/grilles for special closure laws for a Volume element,

…/pipe/grilles for special closure laws for an Axial element.

Caution:

In any case, you must be aware that certain parameters interfere with each other, and that modifying them improperly may lead to inconsistencies and/or discontinuities in the physical laws. For example, modifying the natural convection wall to gas heat transfer may lead to unexpected switching from the natural to the forced convection heat transfer zone, if the forced convection term has not been modified accordingly.

99. HOW TO PERFORM A RESTART?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019**
KEY WORDS	**INPUT DECK**
See also	

The restart is performed by the RESTORE directive.

99.1 The save step

To perform a restart calculation, the SAVE directive should have been used in a first calculation to save the values of:

- all state variables,
- all state flags of reactor elements,
- and all local variables of the input deck.

All values are saved in the same file named “V25.RESTART” (see How to save results).

Note that user input deck variables are automatically saved by the SAVE directive and recovered at restart (as well as time and time step values)

The user simply has to ensure that the local user TIME and DT values where updated with the corresponding CATHARE internal variable before the SAVE action: (see the following right and wrong way to perform the SAVE action

Right way:	**Wrong way:**

...
REPEAT BLOCK1 NSTEPS;
TIME = TIME + DT;
TRANSIENT CIRCANON TIME DT;
DT = NEWDT;
TIME = NEWTIME;
IF (TIME > TMAX) THEN;
SAVE 3;
QUIT BLOCK1;
ENDIF;
END BLOCK1;
……
	...
	REPEAT BLOCK1 NSTEPS;
	TIME = TIME + DT;
	TRANSIENT CIRCANON TIME DT;
	IF (TIME > TMAX) THEN;
	SAVE 3;
	QUIT BLOCK1;
	ENDIF;
	DT = NEWDT;
	TIME = NEWTIME;
	END BLOCK1;
	……

	Here the calculation has chosen a new value for the next time step and the time reached may be different from the one first desired BUT user corresponding variables TIME and DT are not yet updated with CATHARE - calculated (for time) / proposed (for time step) - values and will be saved with the wrong values)

99.2 The restore step

In order to proceed with a calculation restart, the user should create a restart input deck following these recommendations:

- It is recommended to start from the first calculation input deck to build the restart input deck. You may need the regulation loops or some operations made during the transient. This depends on the time at which you want to perform the restart.
- The block data can be removed from the restart input deck except if you are using a 3D element.
- The declaration of all DOUBLE and INTEGER local variables of the first calculation which are still used in the restart calculation must be kept. You can add the declaration of new local variables.
- Use the RESTORE directive with the right number for the save.
- Only the directives INTEGER, DOUBLE, CHAR8 and TABLE can be used before the RESTORE directive in the execution block. Any other directive will stop the reader execution with an error message.
- Restore the arguments (TIME, DT) of the transient by using NEWTIME and NEWDT operators.

The scenario should then continue.

100. HOW TO REACH THE INITIAL OR STEADY STATE?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019** **-CATHARE 2 V2.5_1 - User Manual: SSTH/LDAS//EM/2005-035**
KEY WORDS	**STEADY STATE, STATIONARY STATE, INITIAL STATE**
See also	

100.1 What is a CATHARE steady state calculation?

Any CATHARE transient calculations requires an **initial state**. This initial state should be a well stabilised state and may be a **reference state** defined by the knowledge of certain variables (loop flowrates, temperatures, pressure in the pressuriser, pressure differences, pump rotation speed, etc.).

Obtaining **a CATHARE steady state** calculation generally corresponds to the work necessary to obtain the **reference initial state.**

A well stabilised state is obtained when all the thermal-hydraulic variables are a solution to the equations of the code with the time-dependent terms ($\delta/\delta t$) equal to zero. A reference state is obtained when these values are consistent with our knowledge of the system.

100.2 What are the CATHARE tools for calculating a CATHARE steady state?

The CATHARE steady state can be obtained in two steps: an initialisation step and if necessary a user managed regulation step in order to obtain the reference state.

100.2.1 The initialisation of the thermal-hydraulic variables

This initialisation includes two steps both performed by the PERMINIT directive.

- **The initialisation phase**: first CATHARE performs, successively for each element of the reactor, the **initialisation of variables** and calculates false transient time steps. It solves the same equations as for the transient (with $\delta/\delta t$ terms, non-condensable gases, sources, sinks etc.) applied only to a single element. It requires only the knowledge of some upstream values.
- **Then, ten time steps of a CATHARE stabilised transient** are calculated (a transient where the time is fixed to zero) in order to reach a globally coherent state.

100.2.2 User managed regulations

These user managed regulations are sometimes necessary to obtain a reference state.

The regulations use the values of CCVs and modify certain parameters until the thermal and mechanical equilibrium is reached:

- Boundary conditions (directive BCMOD);
- Feedwater or power released by steam generators (directive SGFEED for point SG)
- Flowrate and temperature of water injected by the SOURCE (WRITE directive);
- Valve position (WRITE directive);
- Pump rotation speed (WRITE directive).

100.3 Recommendations for the steady state with transient regulations:

Except in cases with simple circuits and one-phase conditions, a calculation transient with user managed regulations is recommended.

- For the pure initialisation phase, choose conditions easy to calculate: avoid two-phase situations for the primary circuit.
- If the initialisation phase is in a single-phase steam flow, a small steam superheated temperature may help convergence of the stationary algorithm.
- If the steady state is far from the reference state, define a transient to reach flow conditions closer to the reference state (for example inject steam or air and drain water if a two-phase situation must be simulated starting from a single-phase liquid initial state). The write directive can also be used but may be dangerous (See What are the CCVs?)
- Program the regulations necessary to reach the reference state. They can be used for:
  - ❖ changing certain parameters of the circuit: modification of singular pressure drop coefficients to obtain good flowrates, changing the fouling to obtain the good temperature difference between primary and secondary circuits, etc.
  - ❖ changing certain boundary conditions to reach the reference state: outlet pressure for the secondary circuit, feedwater flowrate, etc.

Regulations must be chosen so that the response time is as short as possible:

Do not try to programme necessarily the regulations of the reactor. Some virtual regulations may have a better response time. For example, in reactors the SG level is regulated by changing the cold feedwater flowrate which has a delayed effect. Injecting or draining saturated water into or from the SG cavity will have a faster effect.

Reduce the wall response time to temperature changes by reducing the wall inertia (OPTION ROCP). Do not forget to restore the physical wall inertia at the end of the regulation phase.

Last recommendations:

Check that the regulations are no longer necessary to keep the system in a well stabilised state: stop them and let the system calculate a few time steps.

101. HOW TO READ A LISTING?

REFERENCE:	**-CATHARE 2 V2.5_1 - User Manual: SSTH/LDAS//EM/2005-035**
KEY WORDS	
See also	**LISTING**

Printouts are described in the User Manual

102. HOW TO READ ERROR MESSAGES?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019**
KEY WORD	**ERROR**
See also	***WHAT are the common SYNTAX ERRORS?*** ***HOW to explain a TIME STEP DECREASE?***

Error messages can be generated at three levels in a calculation:

102.1 Error messages issued during the reading of the input deck.

They are often difficult to understand because the directive or operator has not been able to analyse the input deck.

- check the spelling of the operator or directive
- make sure that a "; " ends the command
- make sure that no tabulation character has been created

If these checks show no error, a better understanding of the problem can be obtained by splitting the input deck into small parts to find out which command causes the trouble (See *WHAT are the common SYNTAX ERRORS?**)***

102.2 Error messages issued during creation of an object.

These are self-explanatory.

102.3 Error messages issued during a calculation

Some of these messages can be accompanied by short comments:

- **UTION: UNDETERMINED SYSTEM; ZERO PIVOT AT LINE...**: The assembling Jacobian matrix is undetermined. This may be due to inappropriate boundary conditions.
- **ELIM: ZERO PIVOT DURING INVERSION OF BLOCK NB.....:** The Jacobian matrix for the elimination of internal variables of a 1-D module is undetermined. This is probably due to an error in the system of equations or its programming. Call maintenance.
- **ELEMENT... ALFA OUT OF RANGE** or **ELEMENT... X1 OUT OF RANGE** or **ELEMENT... X2 OUT OF RANGE**: These messages do not mean that the solution of the equations is non-physical (ex. alfa<0) but that such non-physical values were obtained during iterations before convergence. A smaller time step is necessary. The code reduces the time step by itself.
- **ELEM... VALUES OUT OF RANGE IN FPEAU... THE PRESSURE IS**...or **ELEM... VALUES OUT OF RANGE IN FPEAU... THE LIQUID ENTHALPY IS**...or **ELEM... VALUES OUT OF RANGE IN FPEAU... THE GAS ENTHALPY IS...**or **NEGATIVE WALL TEMPERATURE IN...:** These messages do not necessarily mean that the solution of the equations is out of range (ex. P=250 bars). It might be that such values were obtained during iterations before convergence. A smaller time step is necessary. The code reduces the time step by itself.
- **YOU TRIED TO RESTART 10 TIMES WITHOUT SUCCESS**: 10 is the default value of the number of successive time step reductions allowed. If after these reductions, the time step is very low, say dt<10-5, there is a difficult convergence problem: (See *HOW to explain a TIME STEP DECREASE?**).*** If after these reductions, the time step is not very low, say dt>10-5, try to allow more successive time step reductions (OPTION MAXREP 30).

103. HOW TO RESTORE CCV?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019**
KEY WORD	**CCV**
See also	

The CATHARE code provides tools to get the values of certain thermal-hydraulic properties in an element from the input deck, during the calculation. These tools are especially useful when the user wants to make regulations or more generally when he wants to perform conditional operations that will modify the behaviour of the calculation.

Example: In a LOCA calculation, the safety injections must start when a reference pressure in the primary circuit reaches a threshold value. This means that the pressure must be known (in the pressuriser for example) at each time step and that the value must be compared to the threshold value.

103.1 Using VALUE and VALBA operators

There are two different operators for obtaining the values:

- **The VALUE operator** can be used to get the value of a specified CCV in a specific element. It returns an INTEGER or a DOUBLE value depending on the variable the user wants to obtain. Consequently, a local variable must have been declared with the appropriate type, in the input deck, at the beginning of the command block.

  ```
  PPRESSU =       VALUE PRESSURE PRESSU   INF;
  ```

- The VALUE operator can be applied on elements such as AXIALs, VOLUMEs, THREEDs, WALLs, SINKs, SOURCEs etc. It can also be used with CIRCUITs and ZONEs for energy and mass balance checking. The lists of CCVs allowed for each kind of CATHARE object are presented in the Dictionary of operators and directives.
- **The VALBA operator** is similar to VALUE but only deals with CCVs concerning radio-chemical components:
  ```
  BORECL  =       VALBA  LIQFRA        FROIDE1       10       BORON;
  ```

103.2 Using the VALUFEAU operator

The **VALUFEAU** operator provides direct access to the **water and steam properties** from the *command block* of the input deck. The user provides the steam/water state variables as input data; the directive returns all the corresponding properties (see the dictionary of directives and operators).

104. HOW TO SAVE THE RESULTS?

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019**
KEY WORDS	**INPUT DECK**
See also	

Two ways are provided to save the results

- automatic saving
- user-managed saving

104.1 Automatic saving:

Saving is done automatically on logical unit 999 after a code stop for a physical reason (DT < DTMIN, number of restarts > MAXREP). The 999 logical unit file can be used to restart the calculation in some situations.

WARNING: if the stop occurs because of a computational error (in the PILOT for example or obviously because of your system) no saving will be done by the code.

104.2 User-managed saving

The user may save the results of the computation at any time during the steady state or the transient. The results are saved on logical unit 55 (file V15.RESTART).

The user has to call the SAVE directive (SAVE n;). The label n is associated with each save so that it is possible to restart from any save with the RESTORE directive (RESTORE n;).

105. HOW TO SIMULATE FACILITY MEASUREMENTS

REFERENCE:	**-CATHARE 2 V2.5_1 - Dictionary of operators and directives: SMTH/LMDS/EM/2005-019**
KEY WORD	
See also	

Measurements on a power plant, an integral test or separate test facilities may be simulated in CATHARE by using the SENSOR operator. A sensor is a passive element. It is used in the command block to provide access to physical variables of the plant using CCV (VALUE).

Fluid temperature, wall temperature, pressure, liquid and gas mass and volume mass flowrates, liquid and gas velocities, and activity measurements (in liquid, gas and radio-chemical components) can be simulated using sensors.

A level or a DP measurement must be calculated by an external system (in a simulator), in the execution block or in a user subroutine (*HOW TO PROGRAM within CATHARE (UTILX)?*) from the values of the pressure given by two pressure sensors.

Special case of SWLLEVEL SENSOR :

This sensor is used to evaluate the swollen level of a vertical axial or threed element. The following variables are available :

SWLLEVEL(**) : Swell level according to the bottom of the studied zone (first defined in the sensor definition) (m)

QGMNIV(*) : Gas mass flow rate on the vector point below the swell level according to the reference zone area (kg/s)

QGPNIV(*) : Gas mass flow rate on the vector point over the swell level according to the reference zone area (kg/s)

HLIN(**) : Liquid enthalpy at the bottom of the studied zone (J/kg)

HGIN(**) : Gas enthalpy at the bottom of the studied zone (J/kg)

ALIN(**) : Void fraction at the bottom of the studied zone

LIQFIN(*) : Liquid mass flow rate at the bottom of the studied zone according to the reference zone area (kg/s)

GASFIN(*) : Gas mass flow rate at the bottom of the studied zone according to the reference zone area (kg/s)

HLOUT(**) : Liquid enthalpy at the top of the studied zone (J/kg)

HGOUT(**) : Gas enthalpy at the top of the studied zone (J/kg)

ALOUT(**) : Void fraction at the top of the studied zone

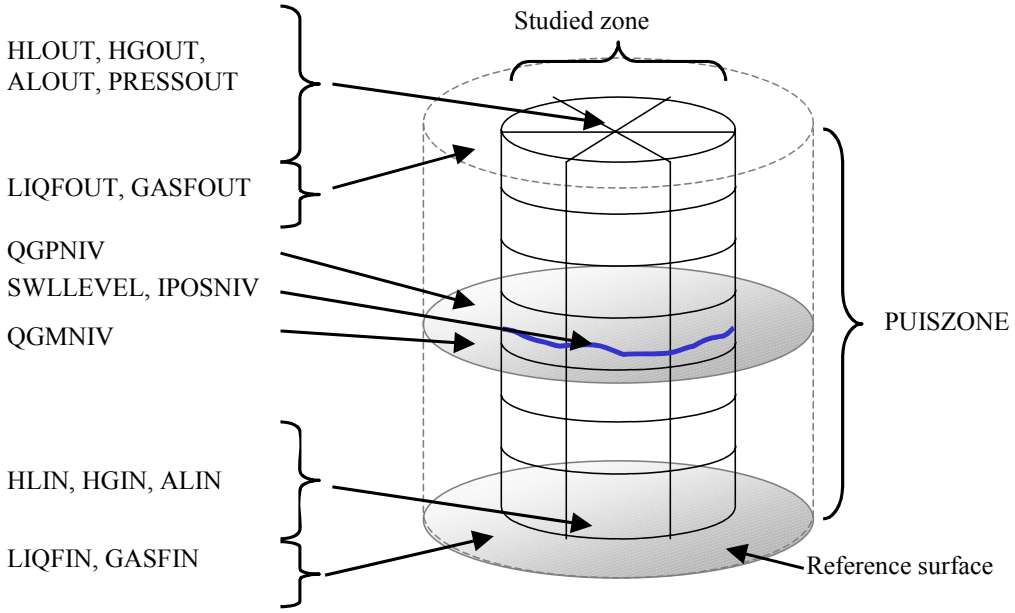
LIQFOUT(*) : Liquid mass flow rate at the top of the studied zone according to the reference zone area (kg/s)

GASFOUT(*) : Gas mass flow rate at the top of the studied zone according to the reference zone area (kg/s)

PRESSOUT(**) : Pressure in the fluid at the top of the studied zone (Pa)

PUISZONE : Total power exchanged between walls and fluid in the reference zone (W)

IPOSNIV : Scalar node number corresponding to the swell level



(*) in case of a threed element, all the flow rates are evaluated according to a reference surface given in the definition of the sensor. That is to say that the final flow rate is as following :

$$Q_{final} = \frac{Q_{considered\ zone}}{S_{considered\ zone}} \times S_{reference\ zone}$$

(**) in case of a threed element, all the enthalpies, the void fractions and the pressures are averaged according to the number of (θ*R) meshes of the studied zone.

106. WHY-WHEN TO USE A VOLUME?

REFERENCE:	**-CATHARE 2 V2.5_1 - Description of the Volume module: SSTH/LDAS/EM/2005-037**
KEY WORDS	**VOLUME, PRESSURISER, UPPER PLENUM, LOWER PLENUM, SG CHANNEL HEAD, PWR (pressurised water reactor), BWR (boiling water reactor), CONTAINMENT**
See also	***HOW TO MODEL a LOWER PLENUM?*** ***HOW TO MODEL a PRESSURISER?*** ***HOW TO MODEL a STEAM GENERATOR CHANNEL HEAD?*** ***HOW TO MODEL an UPPER PLENUM?***

106.1 Advantages and limitations of the VOLUME module

The **VOLUME** module of the CATHARE code has the following advantages:

- It is a simple model. Its behaviour is easy to control by the user. Simplicity is often the solution for a very complex problem. When a very complex situation (3-D flow in a complex geometry) has to be described, this two-node model is sometimes preferable to any attempt to imagine an assembly of several 1-D elements which are supposed to represent the multi-D behaviour.

- It allows multiple connections. There can be several junctions connected at the bottom, at the top, on the sides, or penetrating anywhere into the vessel. This gives a high degree of flexibility but remember that the gain in generality is lost in accuracy.

- It describes the vertical stratification. The axial module is well adapted to describing stratified flows in horizontal or inclined pipes but may have difficulties in case of void fraction stratification in vertical pipes: there is the water-packing problem when the level passes a vector node, there are perturbations of the void fraction profile under the swell level (see qualification reports), and many other problems associated with the discretisation of the momentum equation. The volume module describes stratification in a vertical element without these problems.

- It describes phase separation at junctions. For a side branch it takes account of the position of the interface level with respect to the branch. It also describes vapour or liquid pull-through phenomena (models are not very sophisticated as the module must keep its generality). The tee sub-module also has phase separation models but only for the case of a stratified flow in a horizontal pipe. In the case of a horizontal branch from a vertical pipe, the volume module is more suitable. For this reason it is recommended for the UPPER PLENUM.

The volume module has limitations:

- It assumes low internal velocities. But it is often used in situations where the velocities are not so negligible. In this case, the internal thermal-hydraulic model may be out of range and constitutive laws are just thought to give qualitatively good trends and to avoid non-physical situations (inverse stratification). The inertia of the fluid is not taken into account (for example, inertia of the water in the lower plenum when there are oscillations between core and downcomer).

- It is a simple and flexible model. But, on the other hand, it is never very accurate.

106.2 Recommendations

- Use the volume module for stagnant or quasi-stagnant stratified fluid.
- Also use the volume in other situations when there is no better choice but be fully aware of the assumptions of the model and of the capabilities and limitations of the module:
    - Cases with multiple connections (top of downcomer, upper plenum, etc.)
        - *HOW TO MODEL an UPPER PLENUM?*
    - An inlet channel head for large break LOCA or any situations with high quality and high velocities:
        - *HOW TO MODEL a STEAM GENERATOR CHANNEL HEAD?*
    - A lower plenum, pressuriser and upper head
        - *HOW TO MODEL a LOWER PLENUM?*
        - *HOW TO MODEL a PRESSURISER?*
    - Containment pressurisation calculation (using single or multiple volume modelling) during a LOCA accident involving a break in the containment.
        - *HOW TO MODEL a CONTAINMENT?*

107. WHY-WHEN TO USE A 3D MODULE?

REFERENCE:	**-CATHARE 2 V2.5 - Description of the 3D module: SSTH/LDAS/EM/2004-048**
KEY WORD	**THREED**
See also	**WHAT is a 3D ELEMENT?**

The general objective of the 3D module is to represent large-scale 3D effects in nuclear plants. It can be used from simple 3D geometries or separate effects tests up to the complete PWR vessel 3D description.

The 3D module is recommended for calculations when multi-dimensional effects are of importance in the prediction of experimental phenomena. These effects include, among others:

- re-circulation phenomena
- multi-dimensional counter-current flow such as in an annular downcomer
- for a nuclear power plant, radial power profile in the core which induces strongly non-homogeneous flow distribution
- for a nuclear power plant, non-symmetrical behaviour of the different loops
- voiding of the lower plenum (blowdown phase of a LBLOCA)
- reflooding with 3D effects on the quench front progression.

The 3D module has been validated for the modelling of PWR 3D vessel. It can be use for other applications but users have to validate the model according to theirs uses.

108. WHY-WHEN TO USE A TEE?

REFERENCE:	**-CATHARE 2 V2.5 - Description of the PIPE module and TEE sub-module: SSTH/LDAS//EM/2004-065**
KEY WORDS	**TEE, BREAK**
See also	***WHAT is a TEE?***

The tee sub-module is designed for connecting a small pipe with a larger pipe. It enables the phase separation phenomena to be taken into account. It is suitable for the following components:

- Connection between the hot leg and the surge line of the pressuriser.
- Connection between a hot leg, a cold leg or an intermediate leg to a break pipe. See *HOW TO MODEL a BREAK?*)
- It can be used to inject feedwater in a steam generator (with a BC3 boundary condition)
- Connection between legs and accumulator (see *HOW TO MODEL an ACCUMULATOR?*)

Using a combination of several tees is not recommended for:

- representing cross flows between parallel channels
- representing complex components with multiple connections.

This could lead to numerical problems as it is not a normal use of the sub-module. Moreover this use of the Tee is not qualified.

109. WHY-WHEN TO USE THE STAND ALONE FUEL?

REFERENCE:	**-CATHARE 2 V2.5_1 - Description of the fuel sub-module SSTH/LDAS/EM/2005-043** **-CATHARE 2 V2.5_1 - User Manual: SSTH/LDAS//EM/2005-035**
KEY WORDS	**FUEL, NEUTRONICS, WALL**
See also	***HOW TO MODEL CORE NEUTRONICS?*** ***HOW TO MODEL FUEL RODS?*** ***HOW TO INITIALISE COLD/HOT FUEL DATA?***

The stand alone fuel module (also called CATAFUEL) has been developed to allow fast calculation of fuel rods. For this purpose, the hydraulics of the 1D or 3D elements associated with the fuel rods is computed in a first step and then imposed on the studied fuel rod in a second step. There is no backward influence from the fuel rod computation on the previous hydraulic calculation.

The stand alone fuel module is mainly used in two cases:

- To perform sensitivity studies on fuel rods (influence of the initial gap width, behaviour of fuel rods with different nominal powers, etc.)
- To assess the fuel module. The behaviour of the fuel rod is computed without having to deal with hydraulics.

When a stand alone fuel module is used during a reflooding calculation, the quench front position and velocity are deduced and imposed from the previous hydraulic calculation.

110. WHY-WHEN TO USE A REFLOODING SUB-MODULE?

REFERENCE:	**-CATHARE 2 V2.5_1 : Physical laws used in reflooding sub-module: SMTH/LMDS/EM/2005-036** **- CATHARE-2 V2.5 : Description of the reflooding sub-module**
KEY WORDS	**QUENCH, REFLOODING, REWETTING,**
See also	***WHAT is the REFLOODING SUB-MODULE?*** ***HOW TO MODEL CORE THERMAL-HYDRAULICS?***

The heated structure is attached to a vertical 1D or 3D element.

The rewetting is slow enough so that the axial heat conduction is expected to play a role in the quenching (moreover, it is required for the convergence of the 2d-conduction calculation with a moving mesh). This condition requires a limited coolant capability (flowrate of water entering the element).

The initial wall temperatures must be high enough (Tw > 450°C), and at least higher than the "rewetting temperature" T_{mfs}. If they are too low, standard heat transfers are able to calculate rapid quenching without using the reflooding sub-module. In this case, do not use the reflooding sub-module.

The pressure must be low enough (P<0.6 MPa). Above this pressure, the models are not qualified for rod bundles. Moreover for high pressures, heat transfers by film boiling or steam convection can be efficient enough to result in rapid quenching without using the reflooding sub-module.

END OF DOCUMENT

END