Università di Pisa

Facoltà di Ingegneria

Corso di Laurea Magistrale in Ingegneria Nucleare

Tesi di Laurea

FEM Structural Analysis Of Critical Heat Flux Test Facility

Relatore

Prof. Francesco D'Auria
Prof. Leonardo Bertini
PhD. Fulvio Terzuoli

Candidato

Roberto Martini

Anno Accademico 2015/2016

FEM Analisys of Critical Heat Flux Test Facility

0.1 Introduction

The present work has been performed in the context of the research program between Gruppo di Ricerca Nucleare San Piero a Grado of the University of Pisa and Nuclear and Indusrtial Engineering srl. This research aims at the design and construction of a test facility for the experimental investigation of Critical Heat Flux phenomena at the same conditions as the reference PWR. The present thesis is related to the structural analysis of the test facility, having the purpose of optimizing the facility layout and design the support structure in compliance with the applicable regulations. Reference was made ASME VIII Boiler and Pressure Vessel Code, adopted both for pipilines and for vessels. The analysis were performed with the aid of Finite Element Method using ANSYS APDL MECHANICAL release 15.0 and ANSYS WORKBENCH release 15.0 codes.

0.2 CHF Phenomena in NPP

In a nuclear reactor system the critical heat flux (CHF) is the heat flux at which a boiling crisis occurs causing an abrupt rise of the fuel rod surface temperature and, subsequently, a failure of the cladding material. It is agreed that experimental investigations on CHF have to be performed for each specific design of nuclear reactors. Nowadays a huge number of experimental data are available which were obtained in different flow channel geometries and in different fluids. In addition, a large number of empirical correlations have been developed for the CHF data base obtained from particular flow channel geometries and particular parameter ranges. The purpose of the development of the correlations and of the studies on test facilities is to have less uncertainty and reduce the value of MCHFR, defined as the minimum ratio between the critical heat flux and the operative heat flux, increasing the performances of the nuclear fuel and decreasing the cost.

0.3 Existing CHF Test Facilities

Because knowledge of the phenomena that governs the CHF is critical to the core project, other similar facilities were built around the world:

1. KATHY(Karlstein thermal hydraulic test loop), AREVA, Germany
2. HANARO (High-flux Advanced Neutron Application Reactor), KAE, South Korea
3. ODEN, Westinghouse, Sweden
4. STAF, ENEA, Italy
5. HPCHF(High Pressure Critical Heat Flux), Oregon State University, USA
6. HTRF (Heat Transfer Research Facility), Columbia University, USA

0.4 CHF-TF Layout

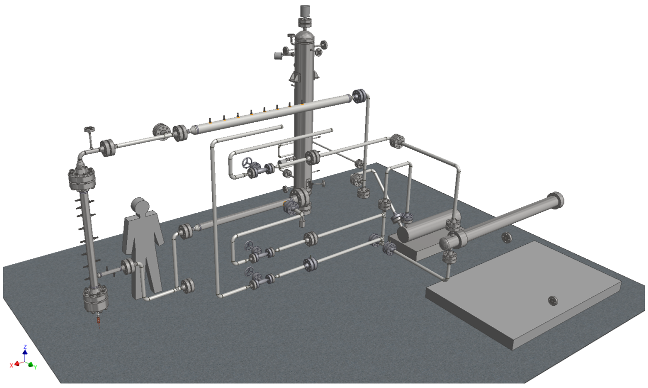


Figure 1: CHF-TS

The Critical Heat Flux Test Facility Project has been built to make a series of experiments about the influence of a particular type of spacer grids on the CHF phenomena, on a facility that simulate the conditions occurring in a real pressurized nuclear power reactor. The radioactive fuel bundles are simulated by nine electrical heating rods, positioned into a test section in a square array. Before arriving in the test section the water pass through the Pre-Heaters, used for the fine regulation of the fluid temperature just before the Test Section inlet. The steam condenser is placed at the exit of the test section, in order to condense the vapor produced in test section. After the condenser fluid goes into primary circulation pump and restarts the loop. Pump, pre-heaters, test section and steam condenser composed the test section loop, but the primary circuit is composed by other two loop:

- Heat-exchanger loop

- heat-exchanger by pass loop.

This two loops have the purpose to feed the injection system of the steam condenser. The pressure in the primary circuit is controlled by the pressurizer, designed in the same way of those that equip a standard Pressurized Water Reactor.
All primary circuit's piping is composed by 2" pipe all made in ASTM SA 312 TP 304L stainless steel, while the surge line is 1" stainless steel pipe. The vessel of the Test Section and the pre-heater are made starting from a 4" pipe, while the steam condenser has nominal dimension of 5", and all are made in ASTM SA 312 TP 304L. All fittings are projected according to specifications given by ASME B16.5 and so do not need verification.
The Maximum Allowable Stress Value for the material is taken from ASME BPVC sec. II sub. D, Table 1A. Stainless steel used for pipes is isotropic and it has linear elastic behaviour. It has the following material properties:

Poisson's Modulus: 0.3
Density: 8030 $\frac{kg}{m^3}$
Young's Modulus and the thermal expansion coefficient aren't take as a constant, but they change in a linear way in function of temperature.

0.5 Finite Method Analisys

The finite element method (FEM) is a numerical technique for finding approximate solutions to boundary value problems for partial differential equations. FEM subdivides a large problem into smaller, simpler, parts, called finite elements. The simple equations that model these finite elements are then assembled into a larger system of equations that models the entire problem.
The analysis is run with the Piping Commands of ANSYS, a special group of commands that enable you to model piping systems and their loads in terms of conventional piping input data, instead of in terms of standard ANSYS direct-generation modeling operations. As you input piping commands, the program internally converts your piping data to direct-generation model data, and stores the converted information in the database. Piping command uses PIPE16 and PIPE 18 elements.

- **PIPE 16**

  PIPE 16 is a uniaxial element with tension-compression, torsion, and bending capabilities. The element has six degrees of freedom at two nodes: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z axes. This element is based on the three-dimensional beam element, and includes simplifications due to its symmetry and standard pipe geometry.

- **PIPE 18**

  PIPE18 is a circularly uniaxial element with tension, compression, torsion, and bending capabilities. The element has six degrees of freedom at each node: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z axis.

In this analysis is considered the mass load, considering steel's density equal to 8030 kg/m3 and a fluid's density equal to 727 kg/m3, at 350 C and 15.5 MPa, and an operative pressure of 15.5 MPa.

Pipes are thermally isolated with a material of density 126 kg/m3, considered in the analysis.

For the temperature distribution, from the experiment's matrix we have chosen the situation that gives saturation conditions at the exit of the test section, because in this condition greater thermal gradients arise, higher temperature in the plant is reached, and condensation occurs more tumultuously.

The fluid exits from the pump at 310 C, it arrives inside the pre-heater at 315 C and at this temperature enters in the test section, where it reaches the saturation's temperature at 15.5 MPa of 345 C.

At this point vapor is condensated and exits from the steam-condenser at 310C. The first step injects at 310 C from by-pass line, while the second step coming from the heat exchanger injects at 77 C.

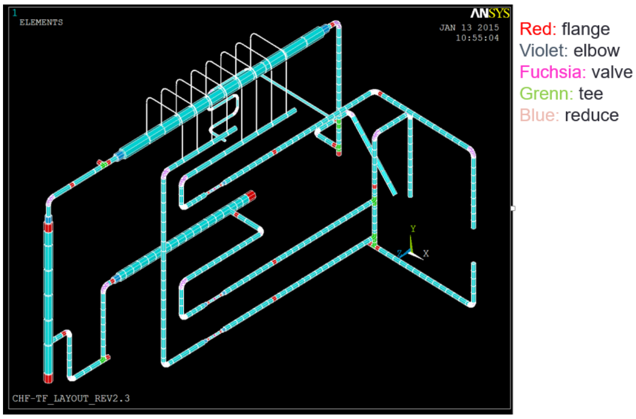


Figure 2:

0.6 Analysis of Results and Design Optimization

The goal of my work is to project a constraining system to design the skid on which the pipes are fixed.

The research of the definite constraining system was done by an iterative pipe stress analysis, starting by a preliminary hypothetical system, based on simple considerations. After each analysis, the critical issues was analysed and implemented in the new one.

The main purpose of the process is to limit the stress on pipes due to internal and external loads due to pressure and to thermal gradient under 94 MPa (conservative assumption), the maximum allowable stress value for the material on the design temperature. To do this it is necessary to minimize the reaction values applied by constraints on the structures, paying attention that deformations are not so big as to create an intensification of the stress or compromise the functionality of the facility.

- **Preliminary System**
  First I have bonded only the pump, the pressurizer and the heat-exchanger as perfect joints (because their rigidity is much greater than the rest of the structure) and test section, to see where the inflection is greater and so where the supports need to be placed. This analysis is performed at ambient temperature and ambient pressure, considering only the weight of steel and water.

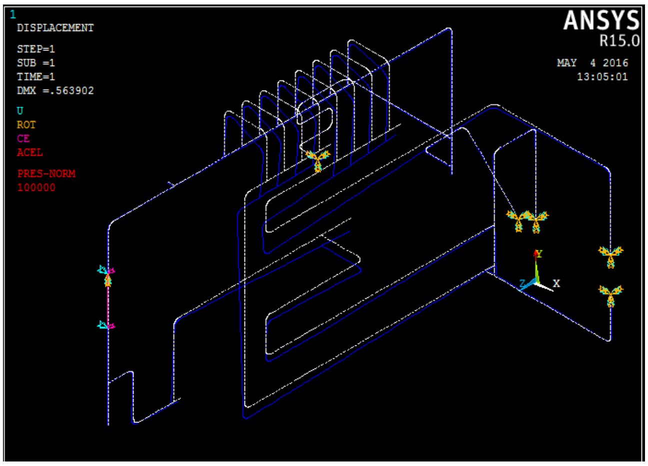


Figure 3:

- **Supports Layout**

  To prevent the vertical displacement of the pipes, we place a series of vertical supports, all bilaterals. The supports are placed under the connection pipe, the pre-heater and the condenser, plus one after the pump inlet to download the load on the pump's joint.

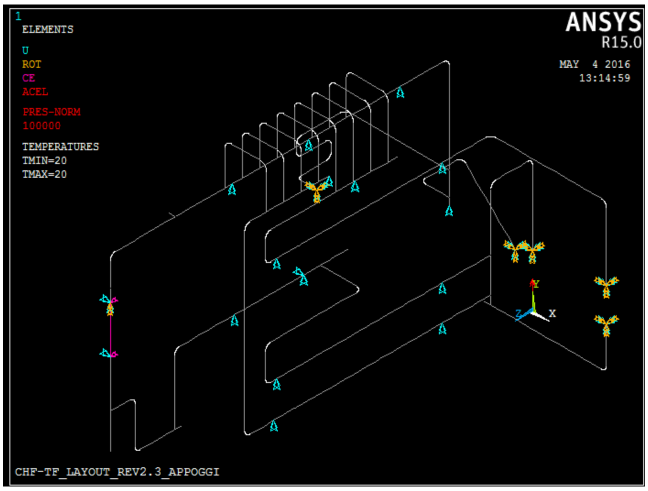


Figure 4:

- **Working Conditions**

  From this layout, we analyze how the system responds to applied loads of pressure and temperature. We can see an important displacement in the direction of the pump axis. This deformation is not dangerous for the integrity and the functionality of the system, but moreover it allows to decrease the stress favoring the thermal dilatations.

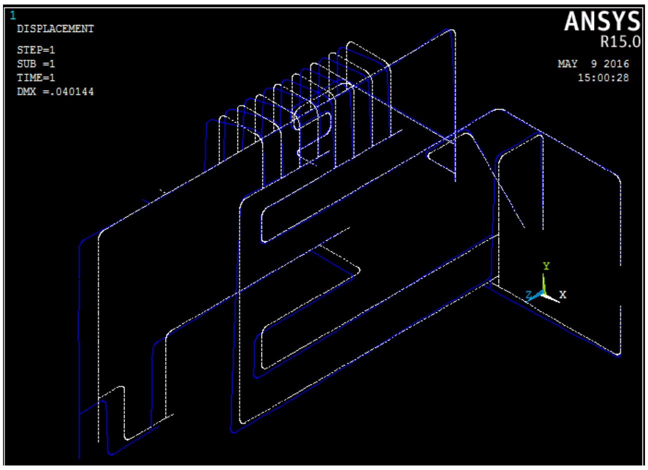


Figure 5:

  The immages show that with a rigid system of constraint we have two main problems: we can see that at the end of condenser there's an intensifications of the stress over the limit, becouse the vertical pipe have a behavior as a beam with double joint. Furthermore some supports are loaded with a very high stress, that could break themselves.

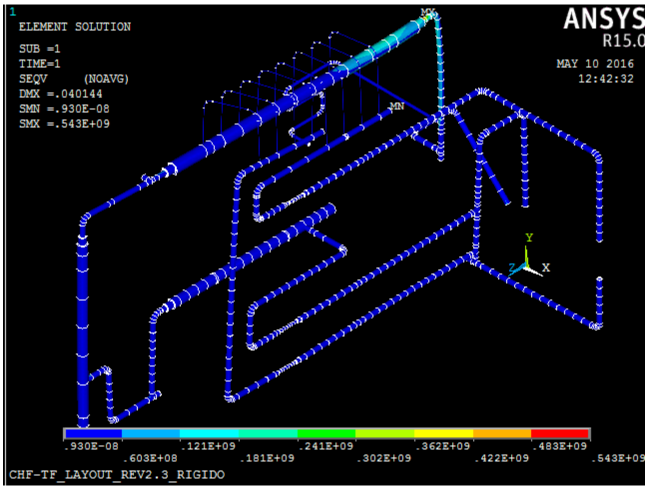


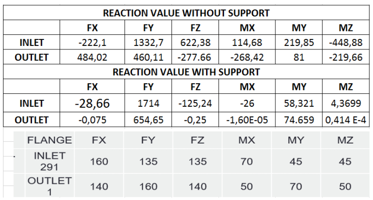
Figure 6: Equivalent Stress

- **Elastic Supports**

  It's necessary to change the supports more solicited with elastic supports made with springs, both to decrease their load, both to give greater flexibility to the structure. We replaced the perfect supports on the condenser with elastic supports able to absorb the vertical loads caused by the thermal expansion. This supports are made using LINK 11 elements. In this way we reduce the reactions on those supports by two order of magnitude. With an initial value of the load of 2559.8 N, we choose spring MEFA FH1-3000, with an elastic constant of 117.9 N/mm.

Auxiliary Support System for the Pump

A critical point is the stress over the pump junctures, because acooording to the manual the values of the pipework load have to be very strict to assure the perfect operation without damage for the pump. We chose to create an auxiliary system that downloads the load over the principal skid, and is given only a normal reaction along the pipe axis over the flange, absorbed by an external structure.



REACTION VALUE WITHOUT SUPPORT						
	FX	FY	FZ	MX	MY	MZ
INLET	-222,1	1332,7	622,38	114,68	219,85	-448,88
OUTLET	484,02	460,11	-277.66	-268,42	81	-219,66
REACTION VALUE WITH SUPPORT						
	FX	FY	FZ	MX	MY	MZ
INLET	-28,66	1714	-125,24	-26	58,321	4,3699
OUTLET	-0,075	654,65	-0,25	-1,60E-05	74.659	0,414 E-4
FLANGE	FX	FY	FZ	MX	MY	MZ
INLET 291	160	135	135	70	45	45
OUTLET 1	140	160	140	50	70	50

Figure 7:

Some loads exceed the reference values, but for FY this does not represent a problem because we suppose to absorbe this forces along the pipe axis with an external structure able to download the loads on the earth. MY values is greater than the maximum value, but we suppose that dowloading FY, also reduces MY under the limit.

Test Section

It's important to prevent rotation and torsion to guarantee the integrity of the heaters and to prevent their rupture. Test section could only be dilated axially, and to absorb the thermal dilatation of the structure test section is whipped on a slide that permits the movement along the pump axis direction.

Buckling Analysis

Buckling analysis gives a safety factor among instability of one order of magnitude.

Modal Analysis

In modal Analysis we chose the pump's frequency of 50Hz as exciters frequencies, the frequency coused by movement of fluid between 0 and 0.5 Hz, and a value for condenstation phenomena of the order of kHz. The own frequencies of the structure are in the order of Hz, but they are very close, and so at high frequencies resonance problems are possible. In this case it's necessary to insert a dumping system in the most solicited parts.

0.7 3D FEM Analysis of Sealing Flange of Test Section

The most difficult and delicate part of the plant is the design of the sealing flange that close below the test section. Through the flange pass the heating rods, and it must support itself and, at the same time, it's important to ensure the system is properly sealed from the outside. Wills Rings are metal O-Rings for static face-sealing applications that give reliable performance over a large temperature range for gases and liquids. Extreme high pressures and vacuums can be sealed with Wills Rings. We start from a standardized flange 4" ASME B16.5 cl.2500, on which we are going to perform mechanical process to generate rods supports, and the cavity for wills rings and for the blockage system. Rods pass through nine hole of two different diameters, to make a support for the rods, and this parts are sealed by a second flange maintained in position by four captive bolts, which deform the Wills Rings.

In this situation it's important to find a compromise between the fluid-dynamics and structural requirements. From a fluid-dynamics point of view the goal is to obtain a high heat exchange, because this part of the plant isn't cooled by the fluid and the high temperatures could damage the rods. It's necessary to minimize the thickness of the flange to obtain this, and eventually make a central hole to increase the capability of cooled systems. An external cooling system could be necessary.

We have performed a 3D FEM Analysis using Ansys Workbench, to compare various solutions differentiated in dimensions and processing. The results show that the stress caused by the pressure on the rod's supports is incompatible with a normal stainless steel use for mechanical construction: more attention needs to be placed on high resistance alloy based on Vanadium instead, which is able to resist up to 100MPa.

0.8 Conclusions

This process has been done to project and design a skid able to support the pipes and the vessels of the system, to guarantee the dimensional and structural integrity of the plant. Furthermore the analysis allow to design correctly the supports themselves, so that they don't yield under the loads caused by the reactions. The results combly with the ASME Boiler and Pressure Vessel Code. The main purpose was to keep all the stress under 94 MPa, that rapresent the limit for the primary strees. This value represent a conservative limit, because we haven't done a strict Pipe Strees Analysis, but we have considered the protection against primary stress caused by mechanical loads, that represent lower stress limit. If S is the Maximum Allowable Stress Value, the limit against bending stress, secondary stress, thermal stress and peak stress are equal to αS with $\alpha > 1.5$. Furthermore the structure does not undergo at fatigue, because the number of cycles of heating are limited, and after the first heating the variations of temperature are low.

We have a margin of 6% compared to the limit, but the maximum value on the structure is limited in a small region, where the greatest contributions are given by bending loads, so it is possible to compare this value of 89.4 Mpa with 1.5S=142,5 MPa, and so the margin increases to 40%. The rest of the structure has a margin of 60%, and is therefore in safety conditions. The download of the pump is not completed, so the necessity to keep attention at the load arises and in case improve the isolation. In modal analysis we have considered only the lower own frequencies, because at high frequencies damping phenomena are present.

Currently the criticality of the project is represented by the sealing system of test section. I have analized one of the possible solutions, and concluded that in order to keep the Wills Rings configuration, further attention should be focused on high performances steel containing vanadium, which is able to resist up to 1000MPa, but unfortunately reference codes are not more detailed for this material.

After the construction of the plant this project could be enriched with a monitoring system able to obtain real forces and real reactions on the pipes and on the supports, and also capable of measuring the real vibrations, to study the response of the structure to the induce loads, and the accuracy of the model.