



UNIVERSITA' DI PADOVA
FACOLTA' DI INGEGNERIA

SCUOLA DI DOTTORATO IN
INGEGNERIA INDUSTRIALE

INDIRIZZO IN
INGEGNERIA ENERGETICA

CICLO: XX

METHODS FOR THE THERMO-MECHANICAL ANALYSIS AND DESIGN OF HIGH POWER ION SOURCES

DIRETTORE: Ch.mo Prof. Paolo F. Bariani

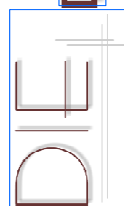
COORDINATORE: Ch.mo Prof. Giorgio Rostagni

SUPERVISORI: Ch.mo Prof. Giuseppe Zollino

Dott. Ing. Pierluigi Zaccaria

DOTTORANDO: Piero Agostinetti

DIPARTIMENTO DI INGEGNERIA ELETTRICA



UNIVERSITÀ DEGLI STUDI DI PADOVA

Dipartimento di Ingegneria Elettrica

Scuola di Dottorato in Ingegneria Industriale

Indirizzo in Ingegneria Energetica

Ciclo XX

**METHODS FOR THE
THERMO-MECHANICAL ANALYSIS AND DESIGN
OF HIGH POWER ION SOURCES**

Direttore: Ch.mo Prof. Paolo F. Bariani

Coordinatore: Ch.mo Prof. Giorgio Rostagni

Supervisori: Ch.mo Prof. Giuseppe Zollino

Dott. Ing. Pierluigi Zaccaria

Dottorando: Piero Agostinetti

31 gennaio 2008

Contents

Acknowledgements	vii
Summary	ix
Riassunto	xi
Introduction	1
1 Neutral beam heating overview	7
1.1 Introduction	7
1.2 Neutral beam generation	9
1.3 Positive and negative ions	11
1.4 Negative ions production	11
1.4.1 Surface production	12
1.4.1.1 Ion sources based on surface production	12
1.4.2 Volume production	13
1.4.2.1 Ion sources based on volume production	13
1.4.3 Two concepts for the ITER NBI Ion Source	14
1.4.4 Cesium seeding	16
1.5 Beam optics calculations	17
1.5.1 Perveance	18
1.5.2 Divergence	18
1.5.3 Beamlets Steering	18
1.5.4 Negative ion extraction and acceleration	19
1.5.5 A typical negative ion extractor and pre-accelerator	20
1.5.6 Overview of negative ion extraction and acceleration physics .	21
1.5.7 Stray electron suppression	23
1.5.8 Stripping losses	24
1.6 Two concepts for the MeV acceleration	24
1.7 Negative ion neutralization	26

1.8	The ELISE test facility at IPP	27
1.9	The Neutral Beam Injector for ITER	28
	Bibliography	32
2	Experimental measurements on ion sources and evaluations of data	33
2.1	Introduction	33
2.2	The RADI ion source	35
2.2.1	Former estimations on the power loads on RADI ion source . .	35
2.3	A new method for the post-processing of calorimetric measurements .	36
2.3.1	Calorimetric measurements on ideal and non-ideal pulses . . .	40
2.4	Calorimetric measurements on the RADI source	41
2.4.1	Description of the cooling system of the RADI ion source . . .	41
2.4.2	Description of the diagnostic system for calorimetric measures	42
2.4.3	Application of the new method for the post-processing of the calorimetric measurements on the Faraday shields	43
2.4.4	Application of the new method for the post-processing of the calorimetric measurements on the Radio Frequency coils . . .	45
2.5	Integration with numerical analyses	46
2.5.1	Sensitivity analysis and robust design philosophy	50
2.5.2	Indications for the design optimization	50
	Bibliography	51
3	Methods for the design optimization of an ion source	53
3.1	Introduction	53
3.1.1	Back-streaming ions at the Large Helical Device	55
3.2	Power load estimations	56
3.3	Back plate cooling system	58
3.3.1	Solution 1	59
3.3.2	Solution 2	59
3.3.3	Solution 3	59
3.3.4	Solution 4	61
3.4	Sputtering analysis	61
3.4.1	Copper surface	64
3.4.2	Tungsten surface	64
3.4.3	Molybdenum surface	64
3.4.4	Comparison on sputtering yields	64
3.4.5	Estimation of the sputtered layers thickness	65
3.4.6	Comments	67

3.4.6.1	Estimation of layer removed by sputtering during the whole ITER life	68
3.4.7	Discussion	69
	Bibliography	72
4	Decision making methods for thermo-mechanical design	73
4.1	Introduction	73
4.2	Description of the Pugh Matrix method	74
4.2.1	Generalities	74
4.2.2	How to build up and use a Pugh Matrix	75
4.2.3	When to use a Decision Matrix	76
4.2.4	Review the concepts	77
4.2.5	Example	77
4.3	Description of the Analytic Hierarchy Process	78
4.3.1	Generalities	78
4.3.2	Criticisms	80
4.3.3	Example	80
	Bibliography	84
5	Decision making methods applied to the accelerator design	85
5.1	Overview of the accelerator design	85
5.2	FEM models description	89
5.2.1	Material properties	89
5.2.2	Convection heat transfer calculation	90
5.2.3	Pressure drop calculation	91
5.2.4	Segment models	92
5.3	Plasma Grid analyses on segment models	93
5.3.1	Heat loads and boundary conditions	94
5.3.1.1	Heat loads	94
5.3.1.2	Structural boundary conditions	95
5.3.2	Hydraulic results	95
5.3.3	Thermal results	97
5.3.4	Structural results	97
5.3.4.1	Equivalent (Von Mises) stress	97
5.3.4.2	Grid deformation	99
5.3.5	Alignment analysis	99
5.3.5.1	Analytical calculations to estimate in-plane deformations	99
5.3.5.2	Alignment optimization	102

5.3.6	Overall comparison between the two schemes	104
5.3.6.1	Pugh Method	104
5.3.6.2	Analytic Hierarchy Process	104
	Bibliography	109
6	Numerical methods for the design of high performance cooling systems	111
6.1	Introduction	111
6.2	Modeling the turbulent flow	112
6.2.1	The Standard k- ϵ Model	114
6.2.2	The Shear-Stress Transport (SST) k- ω Model	115
6.2.3	The Reynolds Stress Model (RSM)	117
6.3	Comparison between the CFD models	119
6.4	Description of a novel approach for the FEM analysis	121
	Bibliography	125
7	Numerical methods applied to the accelerator design	127
7.1	Design optimization	127
7.1.1	First optimization cycle	127
7.1.2	Second optimization cycle	130
7.2	Description of the analyses	133
7.2.1	Computational Fluid Dynamic analysis	133
7.2.2	Thermal analysis	134
7.2.3	Structural analysis	134
7.2.4	Material properties	135
7.2.4.1	Thermal and structural elastic properties	135
7.2.4.2	Plastic properties	135
7.2.5	Heat loads	137
7.2.6	Boundary conditions	139
7.3	Fluid dynamic results	139
7.3.1	Water velocity distribution and limit layer	140
7.3.2	Water pressure drop calculation and comparison with analytical approach	141
7.3.3	Convection heat transfer coefficient calculation and comparison with analytical approach	142
7.4	Thermal and structural results	144
7.5	Sensitivity analyses	146
7.6	Surface cooling efficiency evaluation	147
7.7	Fatigue life evaluation	151

7.8	Discussion	155
	Bibliography	156
8	Methods for the analysis and verification of vacuum vessels	157
8.1	Design criteria	157
8.1.1	Generalities	157
8.1.2	Stress linearization	159
8.1.3	Stress categorization	160
8.1.3.1	Primary stress	160
8.1.3.2	Secondary stress	160
8.1.4	Stress classification according to ASME	160
8.1.5	Static verifications for vacuum vessels	161
8.1.6	Ratcheting verification for vacuum vessels	164
8.1.7	Fatigue verification for vacuum vessels	164
8.1.8	Buckling verification for vacuum vessels	164
8.1.8.1	Load controlled and strain controlled buckling	165
8.1.8.2	ASME rules for buckling prevention	165
8.2	Comparison between analytical and numerical models for buckling analysis	166
8.2.1	Analytical calculations for buckling	167
8.2.2	Numerical calculations for buckling with regular mesh	169
8.2.2.1	Observations	169
8.2.2.2	Sample of analysis with shell elements in Ansys Classic	172
8.2.2.3	Sample of analysis with solid elements in Ansys Classic	173
8.2.2.4	Sample of analysis with solid elements in Ansys Workbench	174
8.3	Numerical calculations with irregular mesh	174
8.3.1	Comparison between buckling models	175
8.3.1.1	Comparison on Equivalent (Von Mises) stress	177
8.3.1.2	Comparison on deformations	177
8.3.1.3	Comparison on Buckling Load Factor	178
8.4	Application: Design analyses and verifications of the ELISE vacuum vessel	179
8.4.1	General description	179
8.4.2	Material	179
8.4.3	Model description	179
8.4.4	Loads and boundary conditions	181
8.4.5	Static results and verifications	182
8.4.6	Ratcheting results and verifications	183

8.4.7	Fatigue results and verifications	185
8.4.8	Buckling results and verifications	187
8.4.9	Deformation results and verifications	188
8.5	Conclusions	188
	Bibliography	189
Conclusions		191
Index		193

Acknowledgements

My PhD work would not have been possible without the collaboration of my colleagues in Consorzio RFX, particularly the “Ingegneria del Plasma” group. I would like to thank everybody who have helped and encouraged me during these three years.

I am also grateful to the engineers and physicists in Max-Planck-Institut für Plasmaphysik (IPP) of Garching. The five months passed there were full of collaborative work and fruitful discussions.

Department of Electrical Engineering (University of Padua) website:

<http://www.die.unipd.it>

Consorzio RFX website:

<http://www.igi.cnr.it>

Max-Planck-Institut für Plasmaphysik website:

<http://www.ipp.mpg.de/eng>

Author’s personal website:

<http://www.pieroagostinetti.eu>

Summary

The present doctorate thesis regards the major issues related to the thermo-mechanical analysis and design of high power ion sources. The activities were carried out in the following three areas:

1. Experimental activities on the existing ion sources.
2. Evaluation of the main operating parameters (plasma density, heat loads etc.) of the ITER Neutral Beam Injector (NBI) ion source.
3. Optimization of the thermo-mechanical design of the ITER NBI ion source, considering both the point of views of a good operating behaviour (in terms of ion current density and electrons-to-ions ratio) and a good reliability of the machine (in terms of structural verifications, low deformations and ability to control the operating parameters).

The main original contributions in the three fields have been respectively:

1. A new method for the post-processing of the calorimetric measurement on an ion source, able to precisely evaluate the heating power loads deposited on the various components of the ion source.
2. A new method for the evaluation of the damages given by sputtering inside an ion source.
3. A new Finite Element Method (FEM) technique especially developed for high performance cooling systems, that integrates non-linear CFD, thermal and structural analysis.

Moreover, the following themes, related to the analysis and design of a generic mechanical component, have been investigated and applied to the ion sources:

- The Decision Making methods, with a comparison between the main approaches and an application on the accelerator cooling system design.

- The Computational Fluid Dynamics analyses, with a comparison between the most sophisticated approaches, analytical formulas and experimental data, and an application to the ITER Neutral Beam Injector design.
- The Buckling analyses, with a comparison between different analytical and numerical approaches for the calculation of the buckling modes of a structure subjected to external pressure, and an application to the vacuum vessel of the ELISE facility.

The methods described on this doctorate thesis were developed during the design activities for the ITER Neutral Beam Test Facility at Consorzio RFX (Padova, Italy) and for the ELISE Facility at the Max-Planck-Institut für Plasmaphysik (Garching, Germany), supported by the European Communities under contracts of Association between EURATOM and ENEA, and carried out within the framework of the European Fusion Development Agreement.

Most of these methods can be used also for the analysis and design of a generic high heat flux component. In particular, they are suitable for the analysis and design of many components for the ITER experiment, for the demonstrative reactor DEMO and for the future commercial fusion reactors.

Riassunto

La presente tesi di dottorato riguarda le principali questioni legate all'analisi e al progetto termomeccanico di sorgenti di ioni ad alta potenza. Le attività sono state svolte nelle seguenti tre aree:

1. Attività sperimentali sulle sorgenti di ioni esistenti.
2. Valutazione dei principali parametri operativi (densità del plasma, carichi termici ecc.) della sorgente di ioni dell'iniettore di neutri di ITER.
3. Ottimizzazione del progetto termomeccanico della sorgente di ioni dell'iniettore di neutri di ITER, considerando entrambi i punti di vista di un buon comportamento operativo (in termini di densità di corrente di ioni e rapporto tra ioni ed elettroni estratti) e una buona affidabilità della macchina (in termini di verifiche strutturali, basse deformazioni e capacità di controllare i parametri operativi).

I principali contributi originali nei tre campi sono stati rispettivamente:

1. Un nuovo metodo per il post-processing delle misure calorimetriche in una sorgente di ioni, capace di valutare precisamente i carichi di potenza termica depositati sui vari componenti della sorgente di ioni.
2. Un nuovo metodo per la valutazione dei danni dati dallo sputtering all'interno di una sorgente di ioni.
3. Una nuova tecnica agli elementi finiti sviluppata specialmente per i sistemi di raffreddamento ad alte prestazioni, che integra analisi non-lineari fluidodinamiche, termiche e strutturali.

Inoltre, i seguenti temi, connessi all'analisi e al progetto di un generico componente meccanico, sono stati studiati e applicati alle sorgenti di ioni:

- I metodi di Decision Making, con il confronto tra gli approcci principali e l'applicazione al progetto del sistema di raffreddamento dell'acceleratore.

- Le analisi di fluidodinamica computazionale, con il confronto tra gli approcci più sofisticati, le formule analitiche e i dati sperimentali, e l'applicazione al progetto dell'iniettore di neutri per ITER.
- Le analisi di stabilità meccanica (buckling), con il confronto tra diversi approcci analitici e numerici per il calcolo dei modi di instabilità di una struttura soggetta a pressione esterna, e un'applicazione alla camera da vuoto dell'esperimento ELISE.

I metodi descritti in questa tesi di dottorato sono stati sviluppati durante le attività di progetto per l'impianto di prova per l'iniettore di neutri di ITER al Consorzio RFX (Padova, Italia) e per l'esperimento ELISE al Max-Planck-Institut für Plasma-physik (Garching, Germania), supportate dall'Unione Europea mediante contratti d'Associazione tra EURATOM, ENEA e IPP, e svolte sotto l'egida di EFDA (European Fusion Development Agreement). Molti di questi metodi possono essere altresì utilizzati per le analisi e il progetto di un generico componente meccanico sottoposto ad elevati flussi termici. In particolare, essi sono adatti per l'analisi e il progetto di parecchi componenti per l'esperimento ITER, per il reattore dimostrativo DEMO e per i futuri reattori a fusione commerciali.

Introduction

*“Everything should be made
as simple as possible,
but not simpler.”*
- A. Einstein -

Nowadays, more than in every other moment of our history, secure and sustainable energy sources are required for the development of the human kind. As a consequence, researchers from all over the world are working to develop a range of environmentally acceptable, safe and sustainable energy technologies. Nuclear fusion is one of them.

Nuclear fusion is the process that powers the sun and other stars - providing heat and light to sustain life on Earth. While gravity confines the hot plasma in the stars, on earth strong magnetic fields can be used to hold the plasma inside a chamber (magnetic confinement approach). To harness fusion power, light nuclei

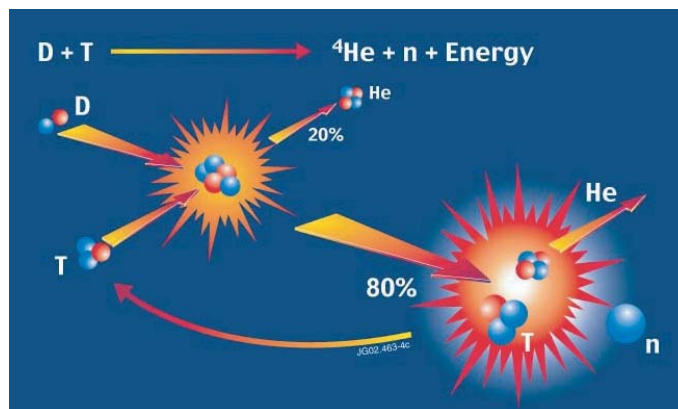


Figure 1: The nuclear fusion reaction

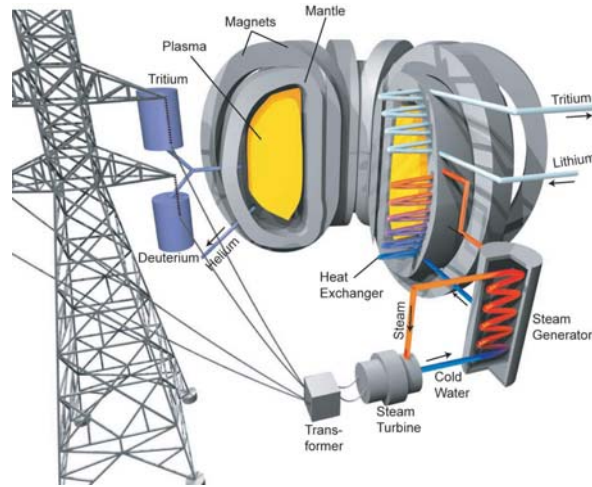


Figure 2: Schematic of a future fusion reactor

are forced together, undergoing reactions that produce a net energy gain. In order to make fusion happen, the fuel (a hot gas called plasma) has to be heated up to very high temperatures (in the order of 100 million degrees).

The “tokamak” has been the most successful concept to confine fusion plasmas up to now. Alternative concepts, like Reversed Field Pinch and Stellarator, are also developed to prepare for fusion power plants.

The primary fuels for nuclear fusion are deuterium and lithium. Deuterium can be extracted from sea water and lithium is abundant in the earth’s upper crust. In a power plant lithium is used to breed the tritium which fuses together with deuterium. Only 150 Kg deuterium and 2÷3 tonnes of lithium are needed for a full year of electricity supply for one million persons.

Fusion power plants will be particularly suited for base load energy generation to serve the needs of densely populated areas and industrial zones. They can also produce hydrogen for a “hydrogen economy” .

ITER is an international tokamak research project which is intended to be an experimental step between today’s studies of plasma physics and future electricity-producing fusion power plants. It will build upon research conducted with devices such as DIII-D, EAST, TFTR, JET, JT-60, and T-15, and will be considerably larger than any of them.

On November 21, 2006, the seven participants formally agreed to fund the project. The program is anticipated to last for 30 years – 10 years for construction, and 20 years of operation – and cost approximately 10 billion Euro, which would make it one of the most expensive modern technoscientific megaprojects. It will be based in Cadarache, France. It is technically ready to start construction and the first

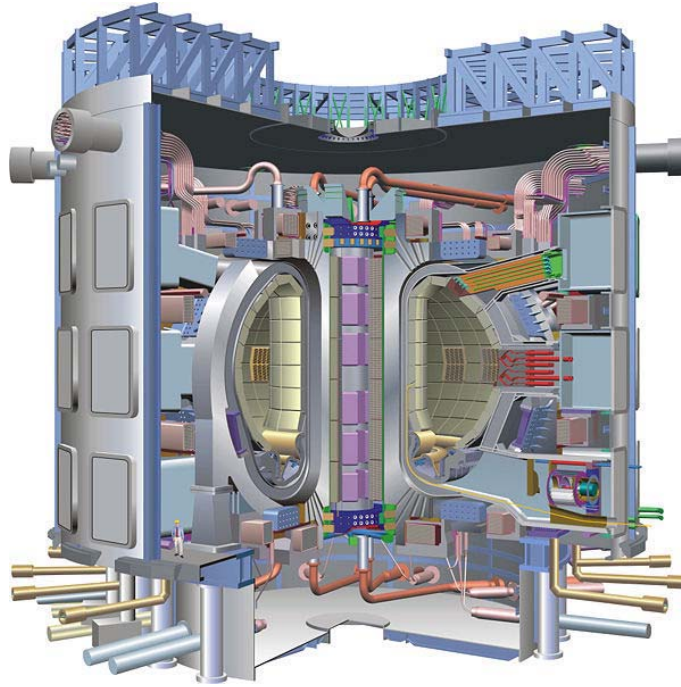


Figure 3: Technical Cutaway of the ITER Tokamak Torus encasing. Note the human figure for size comparison

plasma operation is expected in 2016. The main goal is to produce approximately 500 MW of fusion power sustained for up to 400 seconds (compared to JET's peak of 16 MW for less than a second) by the fusion of about 0.5 g of deuterium/tritium mixture in its approximately 840 m³ reactor chamber.

ITER is the acronym for International Thermonuclear Experimental Reactor, but also means “trail” or “path” in Latin, and this double meaning reflects ITER's role in harnessing nuclear fusion as a peaceful power source.

The Heating and Current Drive (H&CD) systems for ITER are an appropriate combination of Neutral Beam Injectors (NBI) and Radio Frequency (RF) H&CD antennas operating at the electron cyclotron (EC), ion cyclotron (IC) and lower hybrid (LH) frequency. They aim at providing an active control on all the key phases of the operating scenarios:

- the plasma temperature rise, while density is increased to allow the requisite fusion power,
- the achievement of a steady burn,
- the control of the excursions about the operating point,

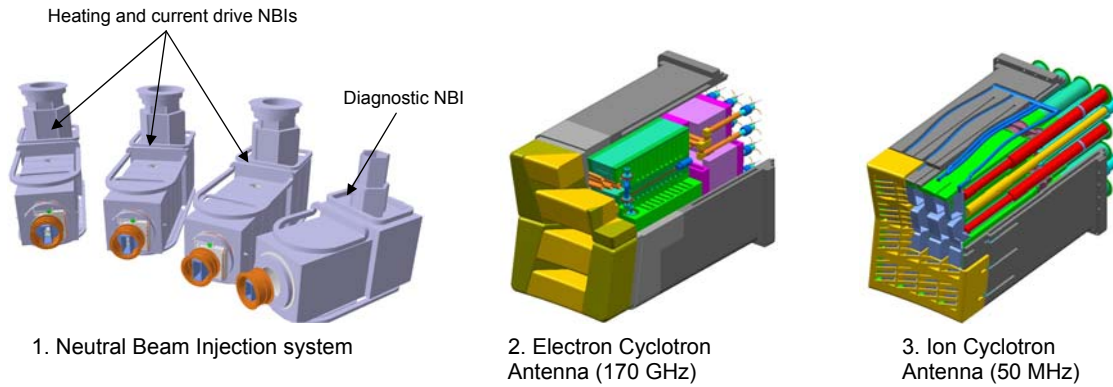


Figure 4: Overview of the Heating & Current Drive systems

- the suppression of instabilities, and
- the achievement of a soft termination

The neutral beam system design consists at present of two heating and current drive (H&CD) injectors and one diagnostic neutral beam (DNB) injector. Each H&CD injector will deliver a deuterium beam of 16.5 MW (total 33 MW), with energy of 1MeV, and will be able to operate for long pulses (up to 3600 s for steady state operation).

In order to obtain an effective and reliable Neutral Beam Injector (NBI) for ITER, it is required a large amount of research & development, both on physics and engineering sides. For this reason, a Neutral Beam Test Facility is foreseen to be built in Padova. The facility has the goal to address and solve the main physics and engineering issues related to this system.

The present Doctorate Thesis regards some of this issues.

An overview on the physics theories on the operations of the existing ion source is given in Chapter 1. These theories, coming from the experimental and theoretical activities carried out in many laboratories around the world, must be always kept in mind by anyone who wants to develop a new facility in this field.

Experimental measures on an existing ion source are described in Chapter 2. These measures are post-processed according to a new model, that permits to obtain precise information on the power deposited on the various components of the source, driving the design of the ITER NBI ion source. These evaluations are to be considered also for the design of the new facilities, with proper scaling laws. A new method for the post-processing of the calorimetric measurement on an ion source, in order to precisely evaluate the power loads on the various components of the ion source, is described.

The optimization of the ion source is considered in Chapter 3, introducing a new method for the evaluation of the layer removed by sputtering during operation of an ion source.

Chapter 4 deals with the decision making methods, which are general optimization techniques applicable to a large amount of subjects, ranging from politics to engineering. The application of these methods to the design of the Plasma Grid of the NBI (one of the most critical components of the whole machine) is explained in Chapter 5.

Chapter 6 introduces a new FEM technique especially developed for high performance cooling systems, that integrates non-linear CFD, thermal and structural analysis. The application of this to the design of the Extraction Grid is illustrated in Chapter 7.

Chapter 8 deals with the structural analyses and verifications of vacuum vessels. As an ion source works inside a vacuum environment, all the components must operate inside a proper vacuum vessel. After a discussion on the design techniques, an application on the vacuum vessel design of the ELISE (Extraction from a Large Ion Source Experiment) facility is carried out.

Chapter 1

Neutral beam heating overview

SUMMARY - Neutral beam heating is at the moment the most important method for plasma heating in many fusion experiments around the world. The concept is straightforward: neutral atoms can penetrate through the confining magnetic field and are ionized in the plasma via collisions with electron and ions. The fast ions generated in this way are then confined by the magnetic field as well. If their kinetic energy is large compared with the plasma temperature, they deliver their energy to plasma ions and electrons by collisions, thereby heating up the plasma. Fast neutral atom beams are generated via charge exchange neutralization of ion beams. This chapter summarizes the physical basis of generation and neutralization of an ion beam, that must be always kept in mind during the design of the related facilities.

1.1 Introduction

High power neutral beams for heating high temperature plasma are required for realization of controlled thermonuclear fusion as an energy source. The ion sources used for such Neutral Beam Injection (NBI) systems are characterized by high energy (several hundred keV) and high current (several tens of A), and consequently by high power (several tens of MW).

The term ion source in this thesis indicates the assembly of:

1. a plasma source, where the plasma is generated;
2. an extractor, where the ions in the plasma are extracted by means of an electric potential, forming an ion beam;
3. an accelerator, where the ion beam is accelerated to the maximum potential.

This terminology is in agreement with most of the literature on this topic, since the early times of the research activities on this topic. Nevertheless, in some documents the assembly here introduced is referred as “beam source” while the term “ion source” indicates only the plasma source. This happens for example in some ITER official documents.

The injected beam energy is determined by the expected penetration depth in the target plasma. Beam energy has increased from about 20 keV for the first experiments on injection into magnetically confined plasmas in 1974 up to about 400 keV at present, as the target plasma size has become larger. In a future fusion reactor the beam energy will need to be up to 2 MeV. The neutral is created by charge-exchange conversion from an ion beam, and at higher energies, above 100 keV nucleon⁻¹, the neutralization efficiency for positive ions decreases drastically while maintaining at around 60% for negative ions. Thus large scale, positive ion sources were developed in the 70s and 80s when the required beam energy was below 80 keV nucleon⁻¹, followed by negative ion source development for the high energy NBI system aimed at reactor-sized fusion machines [1, 2].

There are several circumstances that make the principle of neutral injection work [3]:

- the size of the charge exchange cross sections are just adequate for an efficient neutralization (at least for energies of several tens of keV);
- the size of the ionization cross sections in the plasma are just adequate to trap the neutral beam efficiently (at least for medium size plasmas);
- the collision rates in the plasma are just adequate to slow the fast ions down and distribute the resulting heat between plasma ions and electrons.

However, the characteristics of neutralization and trapping also define the limitations of neutral beam heating. In the following, short summaries of the theory of neutral beam heating and the technical background of neutral beam generation are given.

The interaction of fast neutral atoms with a plasma comprises the following physical processes [4], the last three of which occur simultaneously:

- ionization of the fast neutral atoms by collisions with plasma electrons and ions;
- the drift motion of the fast ions in the magnetic field;
- the collisions of the fast ions with plasma ions and electrons, giving rise to slowing-down and scattering;

- the charge exchange collisions of the fast ions with background neutral atoms.

All of these processes together not only determine the local input rates of particles and energy (power and particle deposition profiles), but also of toroidal momentum and toroidal current.

Furthermore, these processes are related to certain losses. Due to the neutral beam not entirely ionized (shine-through), or the fast ion orbits intersecting the walls before the complete thermalization of the ions (orbit losses).

1.2 Neutral beam generation

The characteristics of an ion beam are determined by the plasma and the accelerator. Thus, for example, the ion beam current is determined by the plasma density, the plasma electron temperature, the extraction voltage and the extractor geometry. The beam emittance is determined by the plasma density distribution, the plasma ion temperature and the extractor geometry. The beam composition is clearly determined by the composition of the plasma. The physics of the ion source is thus largely plasma physics.

The generation of a neutral beam and its transport are best understood by subdividing the process into its successive steps, as shown in Fig. 1.1:

- generation of a powerful ion beam;
- neutralization of the ion beam;
- transport of the neutral beam to the plasma vessel.

The generation of a powerful ion beam is generally accomplished in a very well known way from accelerators in atomic and nuclear physics: positive or negative hydrogen ions are extracted electrostatically from a suitable plasma source and accelerated to energies of several tens of kV. The essential difference to accelerator beams is the much higher beam current (tens of ampere) necessary to achieve the required power (several MW) and consequently the much larger beam cross sections (several hundreds of cm^2). This extension from the level of milliamperes to tens of amperes, maintaining a reasonable beam quality, proceeded as a consequence of a number of distinct achievements:

- the generation of large-area, uniform, quiescent plasmas for ion extraction (arc and RF sources);

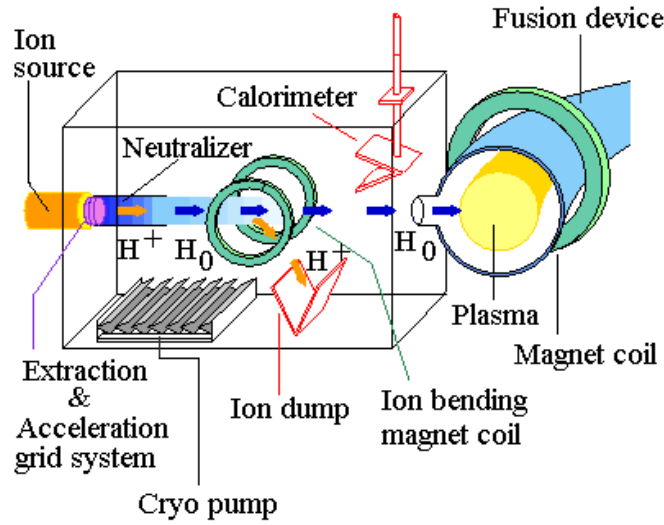


Figure 1.1: Neutral beam generation scheme.

- the idea of subdividing the beam into many beamlets by using electrodes with multiple apertures;
- the discovery of simple ways of controlling the direction of individual beamlets in order to yield the desired overall beam focal properties;
- the development of the technical means of cooling and protecting the delicate electrodes against high voltage breakdowns.

Charge neutralization of the ions occurs in charge-changing collisions of the fast ions with the hydrogen molecules present in the neutralizer. Cold gas is used to create a certain gas target density inside the neutralizer for the conversion of the ions into neutrals. One of the inherent limitations of neutral injection using positive ions is that the efficiency of the neutralization is less than 100% and drops with increasing beam energy. Therefore, for larger experiments as ITER, which need energies of up to 1 MeV, negative ion beams have to be developed. Here the neutralization efficiency stays at about 60%, even for these high energies [5].

The transport of the neutral beam into the plasma vessel faces the problems of

- limited acceptance due to the finite size of the apertures (portholes) in the plasma machine;
- removal of the residual fast ions in the beam which, depending on energy, may represent a significant fraction of the total beam power, and thermalization at suitable surfaces (ion dumps);

- removal of the cold gas streaming on the one hand out of the neutralizer and generated on the other hand by the residual ions.

1.3 Positive and negative ions

Positive ion beams, neutralized by charge-exchange on gas targets, have been used in many experiments to produce the required high power neutral beams. But this is limited to energies below 80-100 keV/nucleon, due to the rapid drop of the charge exchange cross section with energy, while beam energies in the 200-1000 keV/nucleon range are needed in larger fusion devices. For this reason, the next generation of neutral beam systems will be based on negative ions. The low binding energy of the additional electron (0.75 eV) enables a easily detaching of this electron, yielding to neutralization efficiencies of about 60%.

1.4 Negative ions production

The development of negative ion-based systems for fusion is progressing steadily, and year after year there is increasing confidence in this promising field: in 1992 the first complete negative ion-based beamline produced 100 kW of 100 keV D_0 beams in the framework of a collaboration between JEARI and CEA, 16 A of H^- were produced at Nagoya in 1994, and 40 mA H^- beams were accelerated to 700 keV at JEARI in 1995 [3]. After these successful small power injectors, large negative ion-based systems have been installed at JT60-U and LHD with achieved maximum parameters of 400 keV, 17.4 A and 180 keV, 34.5 A, respectively [6].

A negative ion-based system with two 1 MeV, 40 MW beamlines is proposed for ITER [7, 8, 9]. These projects use (and will use) large negative ion sources of 40 A with 20 mA/cm² D^- current densities, high performance electrostatic accelerators and efficient neutralizers to convert the D^- beam into D_0 .

It will be seen that the attractive feature of negative ions, its low electron affinity, constitutes a drawback when production or acceleration is considered: firstly it is difficult to attach this additional electron; secondly many negative ions are destroyed (“stripped”) by collisions with plasma or neutral particles before being extracted from the sources or accelerated to the final energy. This makes negative ion production and acceleration a difficult though fascinating subject.

The present high performances have been obtained in particular from large cesium-seeded sources. The fundamental processes which lead to this kind of negative ion production are well known: wall formation and dissociative attachment.

Nevertheless, their relative weight and combination in sources are still not completely understood.

1.4.1 Surface production

The probability of electron capture during backscattering of hydrogen atoms or ions

$$H, H_n^2 + wall \longrightarrow H^-, n = 1 \dots 3 \quad (1.1)$$

depends on the work function ϕ of the surface, on the affinity of the electron A (being 0.75 eV for negative hydrogen ions), and also on the perpendicular velocity $v_\perp$ of the escaping ion. The negative ion yield per H impact on a wall can be predicted by

$$Y(H^-/H) = \exp\left(-\frac{\phi - A}{C \cdot v_\perp}\right) \quad (1.2)$$

where C is a constant, and $\phi - A$ represents the difference of energy between the Fermi level of the surface and the affinity level; it should be as small as possible to enhance the electron capture probability.

The most widely used method to lower the surface work function is to cover a metal with cesium. The optimal amount of cesium was found to be about 0.6 monolayer ($3.3 \cdot 10^{14}$ Cs per cm^2). For tungsten, for example, the work function is reduced to from 5.25 eV to 1.45 eV. For the surface conditions which are found in most cesiated sources, the expected yield $Y(H^-/H)$ or $Y(H^-/H^+)$ is in the $10^{-2} \div 10^{-1}$ range, in agreement with experiments.

1.4.1.1 Ion sources based on surface production

This process has been very efficiently used to produce intense negative ion sources. Sources optimized for surface production (see for example Fig. 1.2) have the following characteristics:

- the atomic or ionic bombardment is as intense as possible;
- the positive ion energy is in the tens of eV range to maximize the negative ion yield, this is achieved for example with biased converters;
- the path of the negative ions in the source is minimized to avoid destruction.

These sources are therefore very compact, with high power densities, which generally limit the pulse duration or make DC operation delicate. Furthermore, the

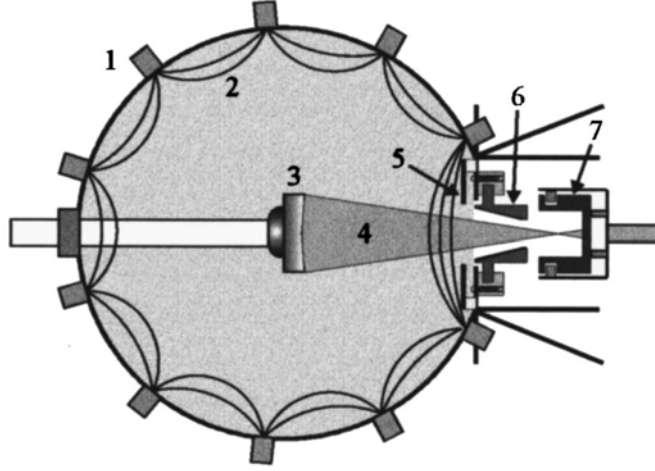


Figure 1.2: Schematic of a “surface plasma source”: (1) cusp magnet; (2) cusp field; (3) converter; (4) H_2 beam; (5) aperture plate; (6) repeller cone; (7) Faraday cup.

initial energy of the negative ion is rather high, due to the acceleration by a large plasma potential or by a converter bias: the resulting transverse energy distribution is hardly compatible with the obtention of high quality electrostatically accelerated beams. For these two reasons the application of such “surface plasma sources” to fusion has not been pushed although the concept is very attractive and might be further improved.

1.4.2 Volume production

The volume production is given by a dissociative attachment of electrons:



The cross-section of this reaction is enhanced when the hydrogen molecules are in high vibrational states.

1.4.2.1 Ion sources based on volume production

The most common type of ion sources based on this volume reaction are the “tandem sources”, as illustrated in Fig. 1.3. They comprise two regions separated by a magnetic filter, the “driver” from the “extraction” chamber. The driver contains the arc filaments (tungsten) which emit the primary electrons - the arc voltages being of the order of 100 V between the cathode emitter and the source walls - or the RF heating region. H_2 gas is fed into this region [10].

A number of permanent magnets are mounted on the source walls and create the multi-cusp configuration for plasma confinement. Another set of magnets is used to create the magnetic filter near the plasma grid. In the extraction region, both n_e and T_e are reduced. This concept optimizes the volume production of negative ions in a two-step process:

- excitation of H_2 molecules by “fast” electron collisions (T_e about $5 \div 10$ eV) to high vibrational levels in the driver region;
- drift of the excited H_2 molecules to the second chamber where the lower electronic temperature (T_e about $1 \div 2$ eV) maximizes the dissociative attachment reaction rates, whilst minimizing the destruction of H^- by electronic and ionic collisions.

In spite of intense development efforts, all the large sources of this type have shown severe limitations:

- H^- currents of at most 3 A, and D^- of below 1 A have been obtained, in spite of very high arc (and RF) power densities (1 W/cm^3);
- saturation of negative ion production with arc and RF power is always observed, explained by the rapid increase of negative ion quenching mechanisms (faster and more numerous electrons, ions, and atoms);
- the operating pressures are high, leading to early neutralization (stripping losses) in the acceleration system;
- high stray electron currents are extracted.

Some of these drawbacks have been recently compensated by adding argon to the hydrogen plasma in the source; H^- current densities of up to 8 mA/cm^2 have been obtained at pressures below 1 Pa with a 25% Ar mixture at the IPP negative ion RF testbed [11, 12]. (For comparison, the ITER NBI design of 1 MeV D_0 injection with 40 A extracted current and a grid area of 2000 cm^2 requires a current density of about 20 mA/cm^2).

1.4.3 Two concepts for the ITER NBI Ion Source

The generation of a powerful ion beam first requires a suitable hydrogen plasma source, which should possess the following characteristics:

- an ion flux density of typically a few tenths of amperes per cm^2 over an area of several hundreds of cm^2 in order to provide the required tens of amperes per source;

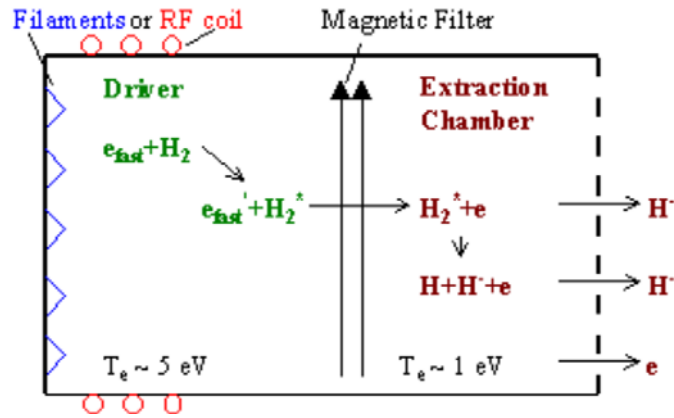


Figure 1.3: Schematic of a “volume plasma source”.

- sufficient uniformity in space ($< 10\%$);
- sufficient uniformity on long pulses (tens of minutes range);
- as high a content of atomic ions as possible (more than 80%, see this section);
- a sufficiently low content of impurity ions;
- a reasonable efficiency for the consumption of electrical energy and hydrogen gas

There are basically two different concepts that are used: Arc sources and Radio Frequency (RF) sources.

- **Arc Sources:** In an arc discharge source electrons are emitted from a hot cathode and accelerated by a DC voltage of typically 100 V into the source, where ionization of the gas molecules leads to the desired arc plasma. Numerous configurations, differing in the way of optimizing the ionization efficiency of the fast electrons (for example by utilizing magnetic fields, biasing the source walls etc.) have evolved over the last 25 years [13]. The development of these sources is based mostly on empirical methods, theoretical work being of a more accompanying nature and addressing singular topics only like ionization efficiency, composition of atomic and molecular species etc.
- **RF Sources:** The RF source utilizes the oscillating electric field, generally induced by a RF coil, for the necessary acceleration of electrons in order to create the source plasma. A typical frequency for RF sources is 1 MHz, and typical RF powers 100 kW [14]. RF sources have many advantages over the arc sources, such as:

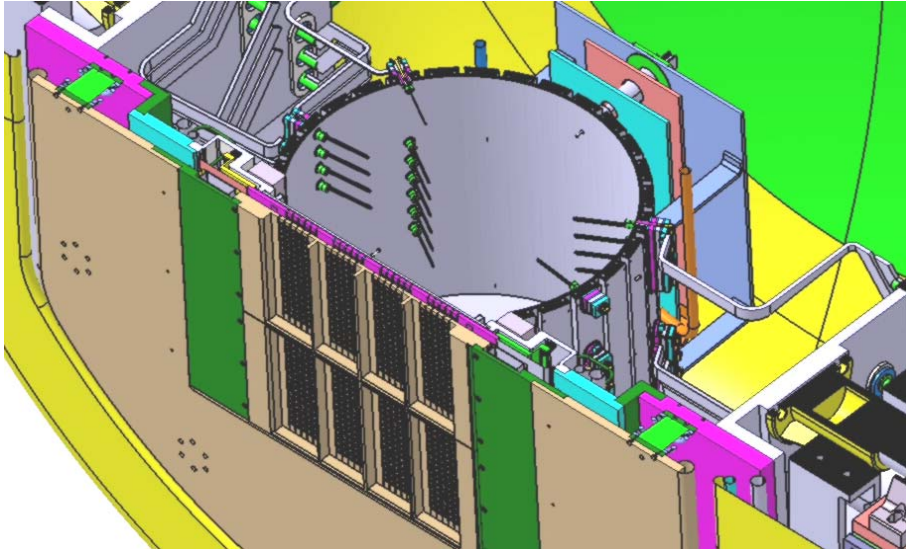


Figure 1.4: Arc driven source overview.

- long lifetime due to the absence of filaments having a limited thermal cycle time,
- low costs due to the possibility of disconnect the RF generators electrically from the high potential of the plasma source,
- better control, during beam modulation, of the extracted ion current due to the direct response of the ion density in the source to changes of the RF input power; this makes active beam current control possible;
- the simplicity of the source (less electrical connections compared to arc sources), making remote handling easier.

Due to these advantages, RF sources are a promising alternative to the Arc sources for the 1 MeV negative ion injectors of ITER. A picture of a RF ion source is shown in Fig. 1.5.

1.4.4 Cesium seeding

When cesium is seeded into the ion sources, the situation changes dramatically [15, 16]:

- the negative ion yield is increased by a factor of $3\div 5$;
- the operating pressure decreases;

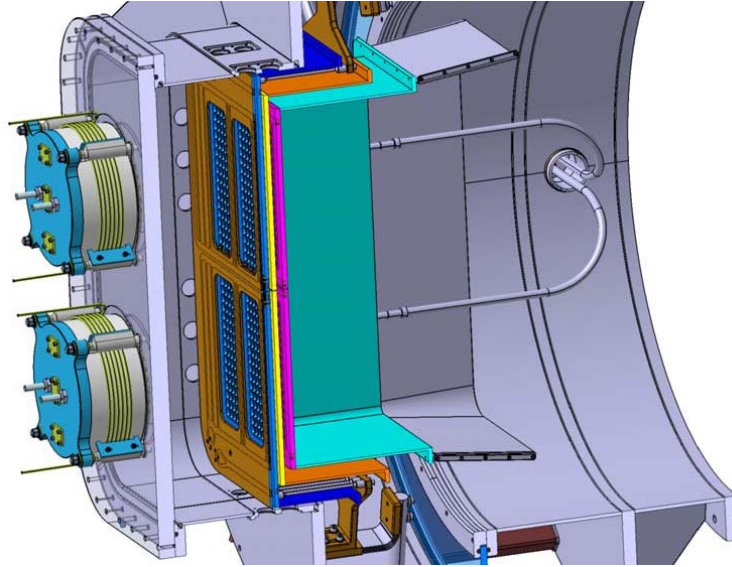


Figure 1.5: Radio Frequency ion source overview.

- the stray electron current ratio drops from $10 \div 100$ in pure sources to $1 \div 5$;
- the production of negative ions in Cs seeded ion sources increases linearly with increasing arc or RF power, and no experiment has yet revealed saturation.

It is not yet completely clear whether the improvement results predominantly from surface production or simultaneously benefits from an enhancement of volume production. In favour of the first explanation is a strong dependence of the negative ion yield on the temperature of the plasma grid in the extraction region, which could be experimentally related to a decrease of the work function [17, 18]. On the other hand, Cs lowers the electron temperature; this might be favourable to dissociative attachment, and, in any case, reduces the destruction rates.

1.5 Beam optics calculations

The actual formation of the ion beam occurs through electrostatic extraction and acceleration of the ions to the desired energy. The correct design of the extraction system plays an important role in the overall performance of the injection system, because there is no way of controlling the beam along his trajectory. Hence in contrast to atomic or nuclear physics accelerators the primary positive or negative ion beam must receive its final status at the level of the extraction system.

Although the physics of ion beam optics is better understood than the field of plasma sources, the situation is not quite satisfactory. There is a lot of analytical and

numerical description available but in varying agreement with experiments. Hence there is not yet a sufficiently solid basis for the designer. The main topics in this field are: perveance, divergence, and steering.

1.5.1 Perveance

The current that can be extracted from a single aperture is basically determined by space-charge-limited flow as known from electron guns, leading to the well-known Langmuir-Child law for non-relativistic charged particles:

$$I = C \cdot V^{\frac{3}{2}} \sqrt{\frac{Z}{M}} \left(\frac{a}{d+h} \right)^2 \quad (1.4)$$

where C is a constant, I the extracted ion current, V the applied electrostatic potential, Z and M the charge and the mass number of the extracted ion, a the aperture radius and d the gap between the plasma and extraction electrodes. $d+h$ is the length of the region in which the ions feel the electrostatic accelerating potential. The ratio

$$P = \frac{I}{V^{3/2}} \quad (1.5)$$

is called perveance.

The optimization of the current per aperture is limited by various constraints. The geometrical dimensions are limited by breakdown (for given voltage, there is a minimum gap) and by aberrations. This explains why a large beam cannot be extracted from a single aperture. Instead the electrodes must be subdivided into many apertures of smaller dimensions.

1.5.2 Divergence

The ions are emitted from a curved surface, whose radius is determined by the self-consistent space-charge of ions. For a given extraction geometry there is an optimum match between the ion flux density delivered by the plasma generator and the electrostatic potential V . Because the matching condition for optimum beam divergence has to be met in every single aperture, the plasma generator has to produce an ion flux density which is reasonable uniform in space and time. Non-uniformities larger than 10% usually lead to a degradation of beam divergence.

1.5.3 Beamlets Steering

The beam dimensions at some distance from the ion source (e.g. for example at the porthole of the plasma device) are determined on the one hand by the divergence of

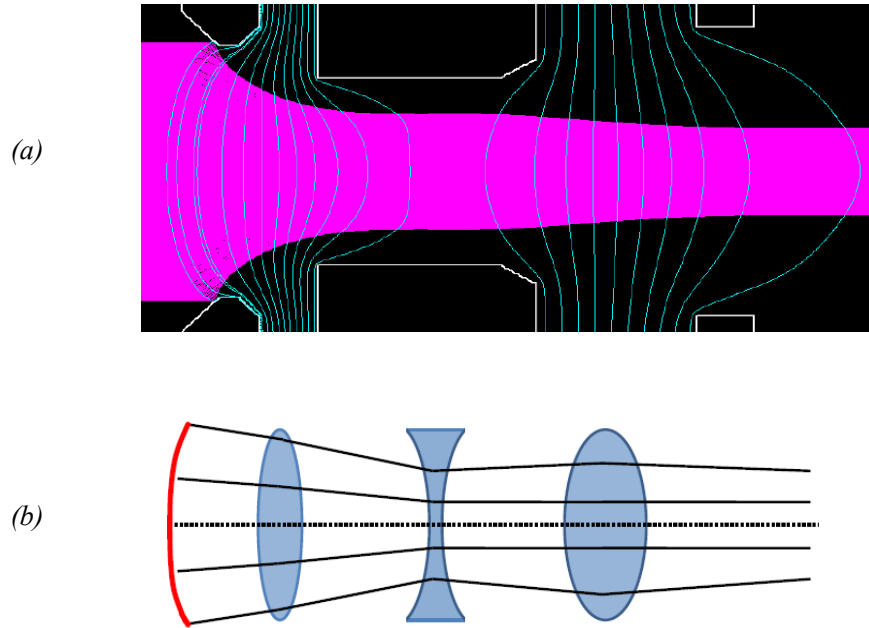


Figure 1.6: Divergence optimization: (a) Analysis of a beamlet with the code SLAC-CAD; (b) Equivalent optical scheme.

the single beamlets and on the other hand by the finite dimensions of the plasma grid. The latter effect can be made negligible small by aiming or steering the individual beamlets or groups of beamlets onto a common focal point or line.

1.5.4 Negative ion extraction and acceleration

Due to power efficiency the large current required for fusion can only be achieved via electrostatic acceleration. This technique has been developed at up to some keVs for large positive ion beams, but the 1 MeV domain has yet to be investigated, and proof-of-principle experiments are operating in Japan and Europe. Besides the difficulties expected with high voltages, in particular electrical breakdowns with high capacitive energy dissipation, the negative ion extraction and acceleration require many features which make the design much more complex than that of positive ions. This is due to two fundamental facts:

- the negatively charged ions are accompanied by co-accelerated stray electrons originating mainly from the source plasma
- the fragile negative ions may be “stripped”, i.e. neutralized or positively charged, during the acceleration. Three steps in acceleration can be distin-

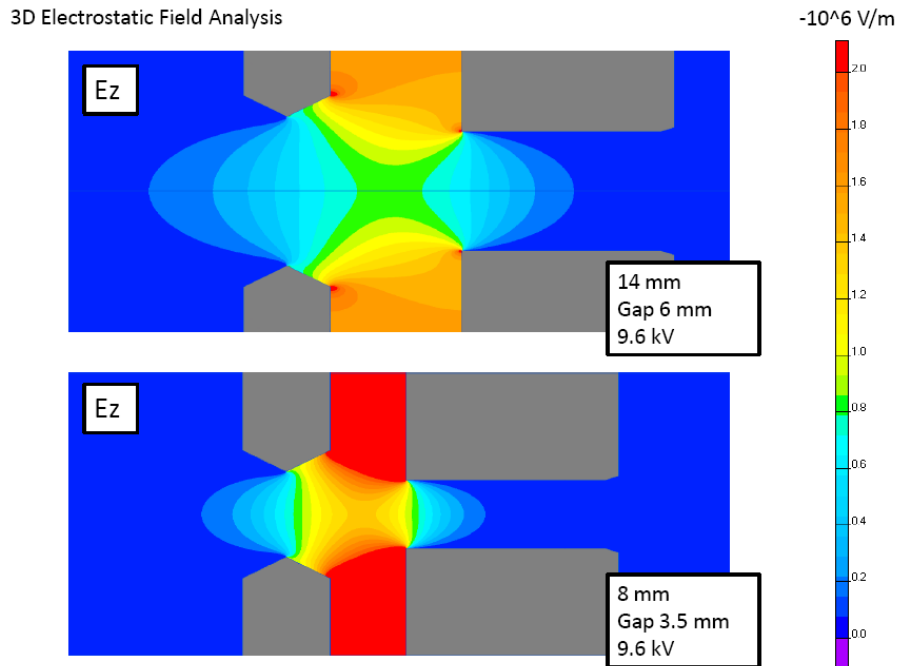


Figure 1.7: Simulation of the electrical field in the extraction area: comparison between two geometries.

guished:

- electrostatic extraction of the ions from the source,
- pre-acceleration, typically to $50 \div 100$ kV,
- full energy acceleration (up to 1 MeV as foreseen for ITER).

Numerical simulations were performed in order to optimize the electrical and magnetic fields in the extraction area (Figg. 1.7 and 1.8).

1.5.5 A typical negative ion extractor and pre-accelerator

An ion beam is typically composed of several beamlets, each of which is extracted from the ion source separately from the others. This is achieved by assembling a set of carefully aligned grids, in which cylindrical or conical apertures are drilled. These grids are polarized at various voltages, to achieve the desired beam energy. The geometry of the accelerating channels and the gap between the grids are carefully designed to control beam optics. The first grid separates the source plasma from the accelerator. In the second grid, often named “extraction grid”, permanent magnets

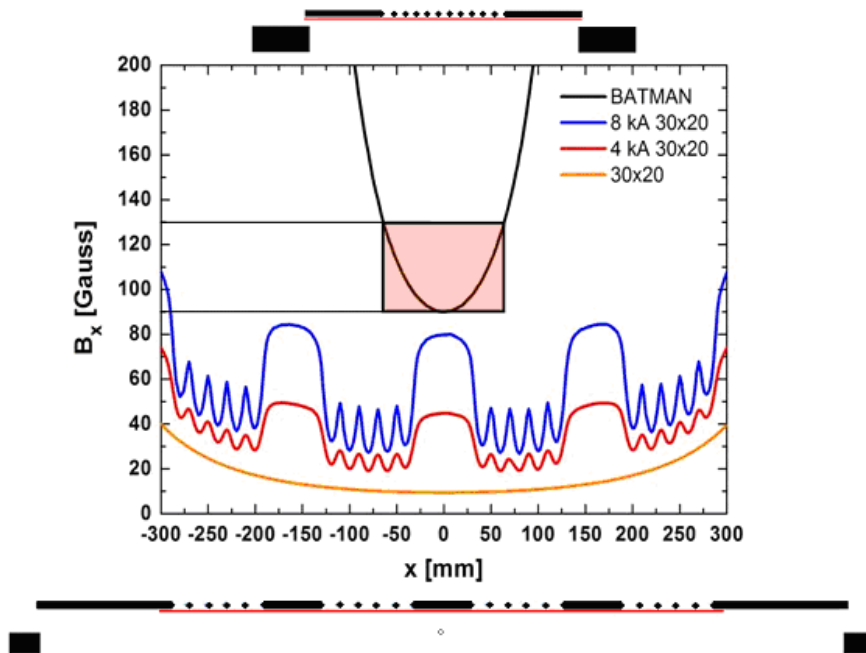


Figure 1.8: Simulation of the magnetic field in the extraction area: comparison between BATMAN and ELISE (with different values for the PG current).

are inserted in order to deflect the stray electrons and to prevent them from being further accelerated. This magnetic field breaks the axial symmetry of the acceleration channel. Therefore, exact calculations require 3-D codes, while 2-D codes were sufficient for calculating positive ions beam optics. Furthermore this magnetic field has to be compensated by opposite fields, as indicated in the figure; in order to prevent any deflection of the D^-/H^- beam, one must ensure that

$$\int B dx = 0$$

along the beam path.

An example of D^- beam and electrons trajectories calculation is given in Fig. 1.9.

1.5.6 Overview of negative ion extraction and acceleration physics

The codes which are used to simulate the extraction and acceleration of negative ions are generally derived from codes used for positive ions. These codes numerically

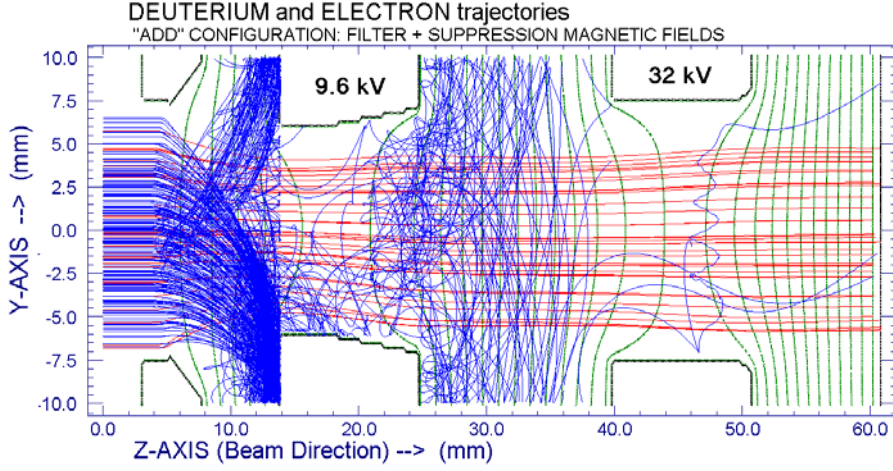


Figure 1.9: Examples of electrons (blue) and ions (red) trajectories for the ITER NBI beam source, calculated with TRACK code [19].

solve, in multi-step iterative process the Poisson and the dynamics equations

$$\Delta \vec{V} = -\frac{\rho}{\varepsilon_0} \quad (1.6)$$

$$m_i \frac{\partial^2 \vec{x}_i}{\partial t^2} = q_i \left(\nabla \vec{V} + \frac{\partial \vec{x}_i}{\partial t} \times \vec{B} \right) \quad (1.7)$$

where m_i and q_i are the mass and the charge state of the trajecting particle, $\vec{B}$ the magnetic deflection field, and x_i the particle spatial coordinates. The boundaries of the problem correspond:

1. to the voltages on the electrodes
2. to the plasma sheath

In the case of negative ions, the simulation of the plasma sheath is particularly complex because:

- Positive and negative ion beam optics are not merely mirror problems with respect to the electric charge; five species have now to be considered in the plasma: e, H⁻, H⁺, H₂⁺ and H₃⁺.
- The source plasma is magnetized. This led some authors to assume that the sheath was frozen and “quasi-planar”, although this assumption was never confirmed.
- The non-axial symmetry of the magnetic field makes it a 3-D problem.

- Large electron currents can be extracted; typically $\frac{I(e)}{I(H^-)}$ is in the $1 \div 20$ range.
- A significant fraction of the negative ions can be stripped in the accelerator, losing their second electron by collisions with the neutral gas. The cross section is large: $10 \div 15 \text{ cm}^2$ at $E(H^-) < 20 \text{ keV}$, while the gas density in the pre-accelerator is in the $10^{13} \div 10^{14} \text{ cm}^{-3}$ range.

A complete simulation of the plasma boundary requires a treatment based on kinetic theory. But proper beam optics simulation requires all the aforementioned physical effects to be taken into account, in particular the electronic contribution to the space charge in the extraction gap and the stripping which modifies the space charge during acceleration.

Today one can consider that the understanding of negative ion beam optics is good. Most codes include the dominant physical effects mentioned above, allowing the major accelerating parameters to be predicted confidently. This is particular important for the new generation of accelerators in the MeV range which are under design. However, progress is still required in the plasma boundary simulation. This could enable controlling secondary phenomena, such as beam halo or aberrated particle trajectories, which may have an impact on the detailed design of high power accelerators.

1.5.7 Stray electron suppression

The main source of stray electrons in a negative ion accelerator is the source plasma. Since large electron currents can be extracted, it is essential to stop these electrons as early as possible. This is the role of the transverse magnetic field. Experiments conducted at JAERI showed that a significant fraction of the electrons, in spite of being forced to impact onto the extraction grid surface, could leak out and be further accelerated. JAERI inserted a third grid just after the extraction grid, which could be negatively biased to $1 \div 2 \text{ kV}$ with respect to the extraction grid in order to repel stray electrons. This system was very efficient, but it increases the grid complexity and the decelerating potential of the electron suppression grid degrades the ion optics.

The electron leakage was explained after the development of a 3-D Monte-Carlo code, which takes into account the secondary emission and the backscattering of electrons. It was demonstrated that the stray electrons originate from backscattering; in the relevant energy range, $5 \div 10 \text{ keV}$, the probability of electron backscattering on a metallic surface can be as high as 30%. If the magnetic configuration of the suppression system is not appropriate, an electron can escape after one or two reflections and can be accelerated downstream. On the other hand, the low energy secondary

electrons (a few tens of eV), in spite of being numerous, have a small Larmor radius which prevents them from progressing far into the accelerator before being dumped on the electrodes.

1.5.8 Stripping losses

As mentioned above, the stripping losses in the acceleration system are due to the gas flowing out of the source. These losses can be as high as 20÷40%, and degrade the overall performance of the system. It is therefore important to reduce the source operating pressure as much as possible. This drives low pressure source (< 1 Pa) development.

1.6 Two concepts for the MeV acceleration

The development of the accelerator of the ion beam up to energies in the MeV magnitude has a mostly technological nature; it is certainly the major issue today. Two concepts are considered at the moment:

- The reference concept, the “Multi Aperture-MULTI Grid” (MAMuG) accelerator (see Fig. 1.10a), accelerates the ions to high voltage in several intermediate steps, each having the same aperture pattern as the plasma and extraction grids. This requires several multi aperture grids at intermediate potentials, the power supplies to feed them and water cooling to each grid. This system is extrapolated from positive ion accelerator concepts and features long and narrow acceleration channels from the source to the last acceleration stage [20, 21];
- The European alternative concept, the “SINGLE Aperture-SINGLE GAP ” (SINGAP) accelerator (see Fig. 1.10b), pre-accelerates the 1280 beamlets to 20-50 keV, then accelerates to the final energy in one single step through large apertures in the final (ground potential) electrode [22, 23, 24]. SINGAP would enable reducing the gas load in the accelerator and greatly simplifying the design of the accelerator, power supplies, and HV transmission lines, but some difficult issues will have to be faced, such as the acceleration of stray electrons to full energy.

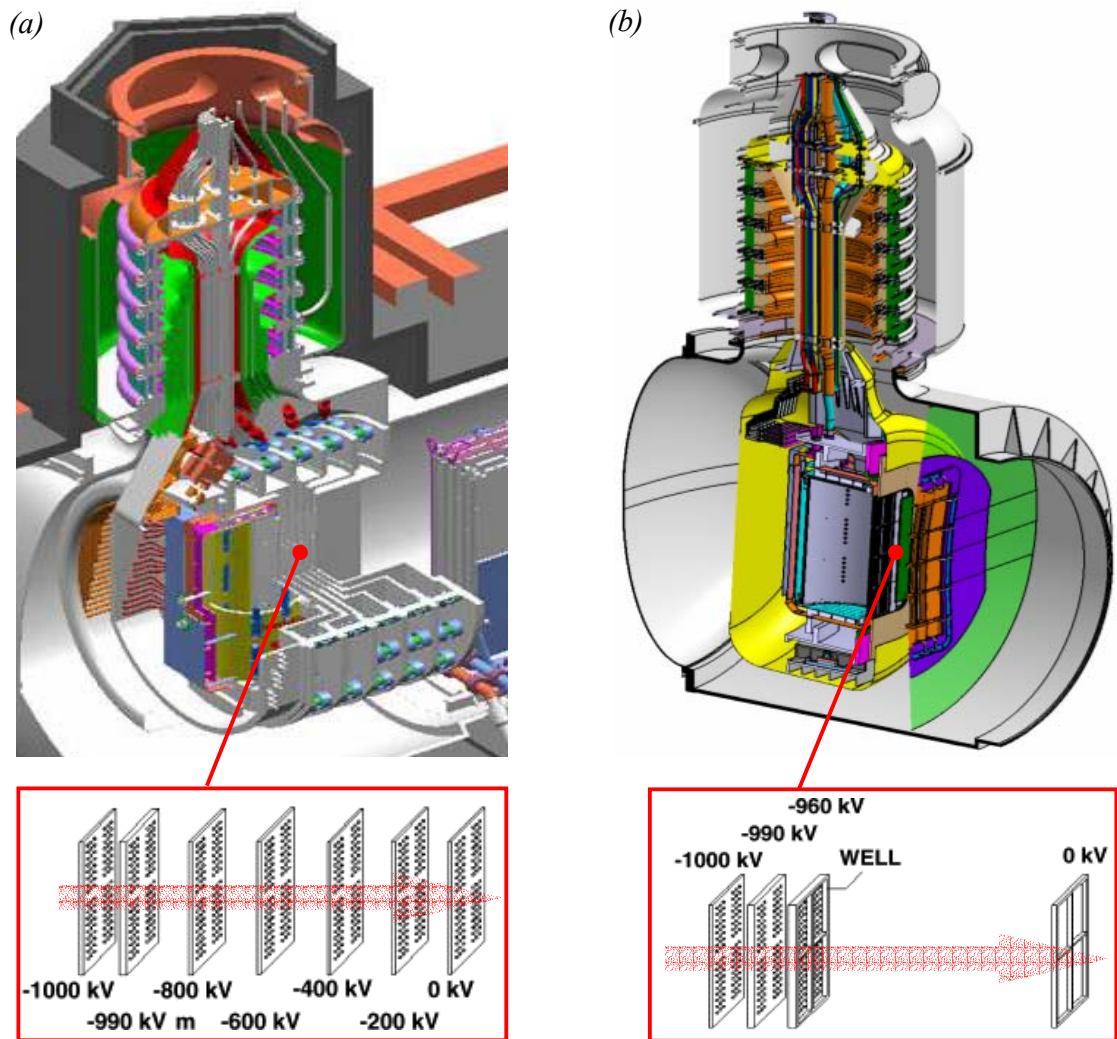


Figure 1.10: Overview of the two concepts for the ITER Neutral Beam Injector: (a) MAMuG; (b) SINGAP.

1.7 Negative ion neutralization

Negative ion neutralization is relatively easy due to the low affinity of the additional electron of only 0.75 eV. A neutralizer is required to:

- provide a high neutralization efficiency;
- operate with the lowest possible gas input, in order to limit the stripping losses of the negative ion beam in the accelerator as well as the reionization losses of the neutralized beam;
- prevent injection of additional impurities into the tokamak;
- require the lowest possible additional power;
- have a long service life and a high reliability.

All negative ion-based neutral beam projects are based at present on the use of gas neutralizers. Anyway, the always created positive ions in the neutralizers require a residual ion dump system in negative ion based neutral beam injectors. The gas neutralizer is based on collisional detachment:



(the fast particles are underlined). Unfortunately, several reactions compete with that. The main competitor is the ionization of the fast neutrals:



The optimum neutralization rate η_{MAX} is, in first approximation, a function of $r = \frac{\sigma_{-0}}{\sigma_{0+}}$, the ratio of the cross sections of the dominant reactions above:

$$\eta_{MAX} = \left(\frac{1}{r} \right)^{\frac{1}{r-1}} \quad (1.10)$$

Since the both dominant reactions correspond to the same physical phenomenon (electron detachment), r and η_{MAX} are practically independent of the beam energy for energies above several 10 keV (see Fig. 1.11).

A gas neutralizer is a technically simple and passive system with a reasonable high efficiency. It has a major drawback: the required high additional gas load which increases the pumping requirements in large beamlines by a factor of 2 or more.

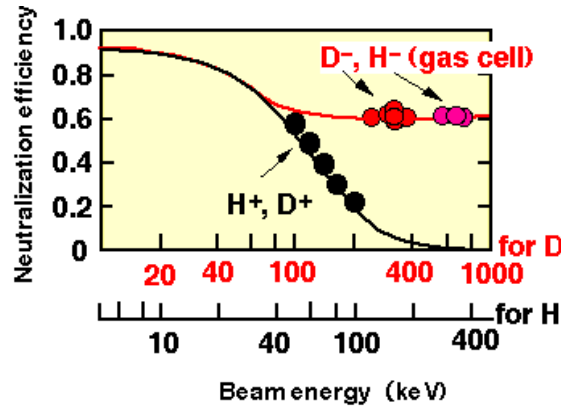


Figure 1.11: Optimum neutralization efficiency for positive and negative hydrogen ions against beam energy per nucleon.

1.8 The ELISE test facility at IPP

ELISE (Extraction from a Large Ion Source Experiment) is a new test facility to be built at the Max-Planck-Institut für Plasmaphysik (IPP) in order to test the extraction of a half-size RF-driven ion source.

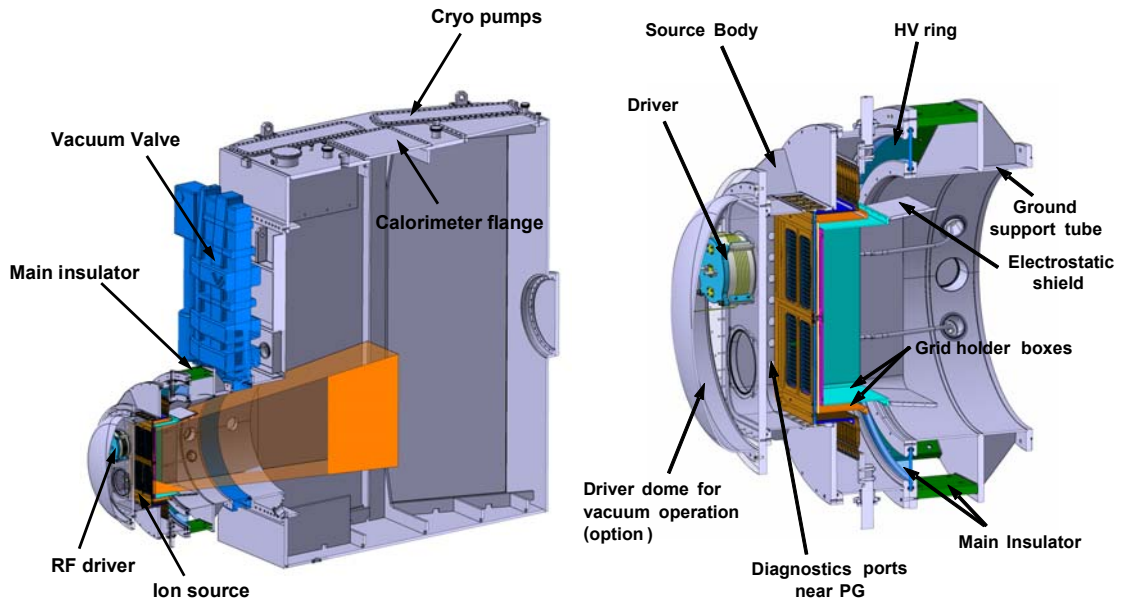


Figure 1.12: Overview of the ELISE facility.

This facility can be considered an intermediate step between the present PINI size RF sources at IPP (BATMAN and MANITU) and the full size ITER source

Isotope	H, D (limited)
Extraction Area	1000 cm ²
Source Size	1.0 x 0.86 m ²
Ion current	20 A
Total Voltage	60 kV
Extraction Voltage	12÷20 kV
RF Power	360 kW (4 x 90 kW)
Pulse Length:	
Plasma	3600 s
Extraction	10 s every 160 s

Table 1.1: Main parameters of ELISE.

- with a height of 1.6 m and a width of 0.6 m. The source has approximately the width and half the height of the ITER source; its modular driver concept will allow an extrapolation to the full size ITER source without any change of the source width.

An overview of the design of the ELISE facility is shown in Fig. 1.12. The main parameters of the new source are reported in Tab. 1.1.

1.9 The Neutral Beam Injector for ITER

The Neutral Beam Injection (NBI) is one of the Heating and Current Drive systems foreseen for ITER (Fig. 1.13). The NBI is based on the acceleration of Hydrogen or Deuterium negative ions up to 1 MV.

Isotope	H, D
Extraction Area	2000 cm ²
Ion current	40 A
Total Voltage	1 MV
Extraction Voltage	~10 kV
RF Power	720 kW (8 x 90 kW)
Pulse lenght	3600 s

Table 1.2: Main parameters of ITER NBI.

Each Injector of the ITER Neutral Beam system will deliver a power up to 16.5 MW and guarantee steady operation for one hour in H or in D. To fulfil these requirements, the Neutral Beam Injector (NBI) must provide a current up to 40 A of negative ions accelerated up to 1 MV.

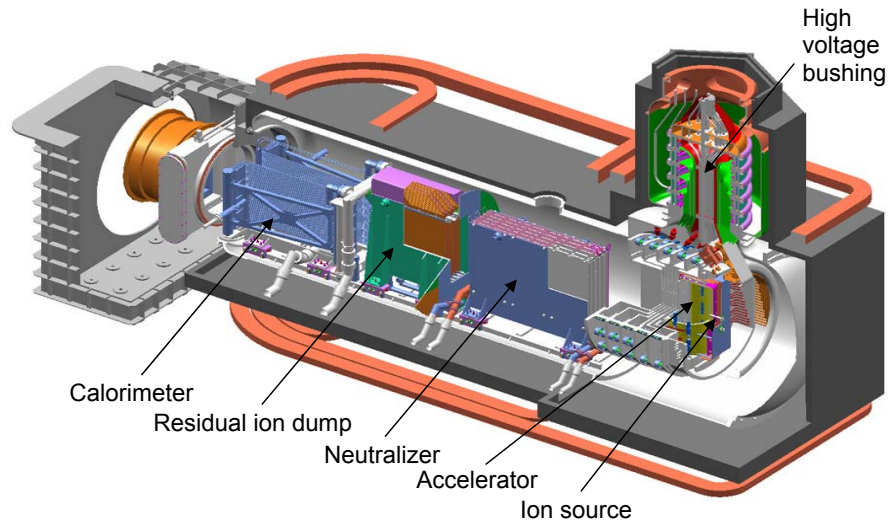


Figure 1.13: Overview of the ITER Neutral Beam Injector.

A test facility is foreseen to investigate the best design options for this device [25]. The most important components of the ITER NBI are:

- A high voltage bushing, that acts as a 1 MV feedthrough for the power supplies, cooling lines and diagnostic cables;
- An ion source, that provide the negative hydrogen or deuterium ions (H^- or D^-);
- An accelerator, that accelerates the ions up to the potential of 1 MV;
- A neutralizer, that transform the negative ions in neutrals;
- A residual ion dump, that dumps the remaining negative and positive ions still present in the beam.
- A calorimeter, that measures the power of the neutral beam. This component is always active in the Testbed Facility, while it can be open and closed in the ITER NBI, depending on the operation phase.

Bibliography

- [1] Various Authors. *The Physics and Technology of Ion Sources*. Edited by I.G. Brown; Wiley-VCH; 22 edition (November 10, 2004).
- [2] Various Authors. *Handbook of Ion Sources*. Edited by B. Wolf; CRC; 1 edition (August 31, 1995).

-
- [3] J. Pamela. *The physics of production, acceleration and neutralization of large negative ion beams*. Plasma Phys. Control. Fusion 37 (1995) 325-336.
 - [4] E. Speth. *Neutral Beam Heating of Fusion Plasmas*. Rep. Prog. Phys. 52 (1989) 57-121.
 - [5] M. Hanada, M. Kashiwagi, T. Inoue, K. Watanabe, and T. Imai. *Experimental comparison between plasma and gas neutralization of high-energy negative ion beams*. Rev. Sci. Instr. 75 (2004) 1813-1815.
 - [6] T. Inoue, M. Taniguchi, T. Morishita, M. Dairaku, M. Hanada, T. Imai, M. Kashiwagi, K. Sakamoto, T. Seki and K. Watanabe. *R&D on a high energy accelerator and a large negative ion source for ITER*. Nucl. Fusion 45 (2005) 790-795.
 - [7] R. S. Hemsworth, J.-H. Feist, M. Hanada, B. Heinemann, T. Inoue, E. Kussel, A. Krylov, P. Lotte, K. Miyamoto, N. Miyamoto, D. Murdoch, A. Nagase, Y. Ohara, Y. Okumura, J. Paméla, A. Panasenkov, K. Shibata, M. Tanii, and M. Watson. *Neutral beams for ITER*. Rev. Sci. Instrum. 67, Issue 3, 1120 (1996).
 - [8] P.L. Mondino, P. Bayetti, E. Di Pietro, R.S. Hemsworth, H. Iida, T. Inoue, K. Ioki, G. Johnson, A.I. Krylov, V.M. Kulygin, P. Massmann, K. Miyamoto, Y. Okumura, A.A. Panasenkov, R.T. Santoro, M. Sironi, Y. Utin, K. Watanabe and M. Yamada. *ITER neutral beam system*. Nuclear Fusion, Vol. 40, No. 3Y.
 - [9] T. Inoue, E. Di Pietro, M. Hanada, R.S. Hemsworth, A. Krylov, V. Kulygin, P. Massmann, P.L. Mondino, Y. Okumura, A. Panasenkov, E. Speth and K. Watanabe. *Design of neutral beam system for ITER-FEAT*. Fus. Eng. and Design 56-57 (2001) 517-521.
 - [10] K. N. Leung, K. W. Ehlers and M. Bacal. *Extraction of volume-produced H^- ions from a multicusp source*. Rev. Sci. Instrum. 54, 56 (1983).
 - [11] E. Speth, H.D. Falter, P. Franzen, U. Fantz, M. Bandyopadhyay, S. Christ, A. Encheva, M. Fröschle, D. Holtum, B. Heinemann, W. Kraus, A. Lorenz, Ch. Martens, P. McNeely, S. Obermayer, R. Riedl, R. Süss, A. Tanga, R. Wilhelm and D. Wunderlich. *Overview of the RF source development programme at IPP Garching*. Nucl. Fusion 46 (2006) 220-238.
 - [12] O. Vollmer, B. Heinemann, W. Kraus, P. McNeely, R. Riedl, E. Speth, R. Trainham and R. Wilhelm. *Progress in the development of a large RF negative ion source for fusion*. Fusion Eng. Des. 74 (2005) 351-357.

- [13] M. Hanada, M. Kashiwagi, T. Morishita, M. Taniguchi, Y. Okumura, T. Takayanagi and K. Watanabe. *Development of negative ion sources for the ITER neutral beam injector*. Fusion Eng. Des. 56 (2001) 505-509.
- [14] P. Franzen, H.D. Falter, U. Fantz, W. Kraus, M. Berger, S. Christ-Koch, M. Fröschle, R. Gutser, B. Heinemann, S. Hilbert, S. Leyer, C. Martens, P. McNeely, R. Riedl, E. Speth and D. Wunderlich. *Progress of the development of the IPP RF negative ion source for the ITER neutral beam system*. Nucl. Fusion 47 (2007) 264-270.
- [15] W. Kraus, P. McNeely, P. Franzen, A. Entscheva, M. Bandyopadhyay, B. Heinemann, R. Riedl, E. Speth and R. Wilhelm. *Development of large RF driven negative ion sources for neutral beam injection*. Fusion Eng. Des. 66-68 (2003) 491-495.
- [16] W. Kraus, B. Heinemann, H.D. Falter, P. Franzen, E. Speth, A. Entscheva, U. Fantz, T. Franke, D. Holtum, Ch. Martens, P. McNeely, R. Riedl and R. Wilhelm. *RF-source development for ITER: Large area H- beam extraction, modifications for long pulse operation and design of a half size ITER source*. Fusion Eng. Des. 74 (2005) 337-341.
- [17] N. Umeda, L.R. Grisham, T. Yamamoto, M. Kuriyama, M. Kawai, T. Ohga, K. Mogaki, N. Akino, H. Yamazaki, K. Usui, A. Honda, L. Guangjiu, K. Watanabe, T. Inoue, M. Hanada, M. Kashiwagi, T. Morishita, M. Dairaku and T. Takayanagi. *Improvement of beam performance in the negative-ion based NBI system for JT-60U*. Nucl. Fusion 43 (2003) 522-526.
- [18] M. Bandyopadhyay. *Studies of an inductively coupled negative ion radio frequency source through simulations and experiments*. PhD Thesis, Technische Universität München, 2004.
- [19] H. P. L. de Esch. *TW4-THHN-IITF2 Final Report- Deliverable 3.1.4.4 Electrons in SINGAP*. EFDA report 2006.
- [20] A. Simonin, J. Bucalossi, C. Desgranges, M. Fumelli, P. Massmann and J. Pamela. *Cadarache 1 MeV negative ion accelerator development for application in thermonuclear fusion research*. Proceedings of the Particle Accelerator Conference, 1997.
- [21] K. Watanabe, N. Ebisawa, Y. Fujiwara, A. Honda, T. Inoue, M. Kazawa, M. Kuriyama, K. Miyamoto, N. Miyamoto, K. Mogaki, A. Nagase, Y. Ohara, Y. Okumura, T. Takayasu, H. Usami and K. Usui. *Development of Negative Ion*

- Beam Accelerators for High Power Neutral Beam Systems*. Fusion Eng. Des. 26 (1995) 445-453.
- [22] H.P.L. de Esch, R.S. Hemsworth and P. Massmann. *SINGAP: the European concept for negative ion acceleration in the ITER neutral injectors*. Rev. Sci. Inst. 73 (2002) 1045.
- [23] H.P.L. de Esch, R.S. Hemsworth and P. Massmann. *Updated physics design ITER-SINGAP accelerator*. Fusion Eng. Des. 73 (2005) 329.
- [24] P. Zaccaria, P. Agostinetti, S. Dal Bello, M. Dalla Palma, D. Marcuzzi, A. Masiello, V. Antoni, J.J. Cordier, R. Hemsworth, B. Heinemann, C. Martens and R. Riedl. *Design integration of SINGAP accelerator and RF Source in the ITER NB injector*. 21st IEEE/NPSS Symposium on Fusion Engineering, Knoxville, Tennessee, 2005.
- [25] J.J. Cordier, R. Hemsworth, M. Chantant, B. Gravit, D. Henry, F. Sabathier, L. Doceul, E. Thomas, D. van Houtte, P. Zaccaria, V. Antoni, S. Dal Bello, D. Marcuzzi, A. Antipenkov, C. Day, M. Dremel and P.L. Mondino. *The ITER neutral beam test facility: Designs of the general infrastructure, cryosystem and cooling plant*. Fusion Eng. Des. 74 (2005) 397-402.

Chapter 2

Experimental measurements on ion sources and evaluations of data

SUMMARY - This chapter deals with the experimental measurement that has been done on some existing Radio Frequency ion sources. These measurements, after a proper post-processing, can give useful indications for the design of the new ion sources, in particular the ones for ITER. A new method for the post-processing of the experimental data is introduced, in order to evaluate more precisely the power loads on the various components of the ion source.

2.1 Introduction

The Radio Frequency ion sources at IPP Garching are at the moment the closest to the ITER requirements [1] . Hence, the operations on these sources can give many indications for the design of the next ion sources, in particular the ones foreseen for ITER.

Presently the test beds operating at IPP are :

- BATMAN. This ion source is mainly devoted to physical questions, like Cs consumption, plasma confinement, extraction optimization etc. and therefore well equipped with diagnostics, but operating with small extraction area ($< 100 \text{ cm}^2$) and short pulses ($< 5 \text{ s}$).
- MANITU. This ion source focuses on long pulse operation ($< 3600 \text{ s}$) and larger extraction areas ($< 300 \text{ cm}^2$).
- RADI. This ion source aims at demonstrating plasma uniformity by scaling the RF source to half the ITER source size.

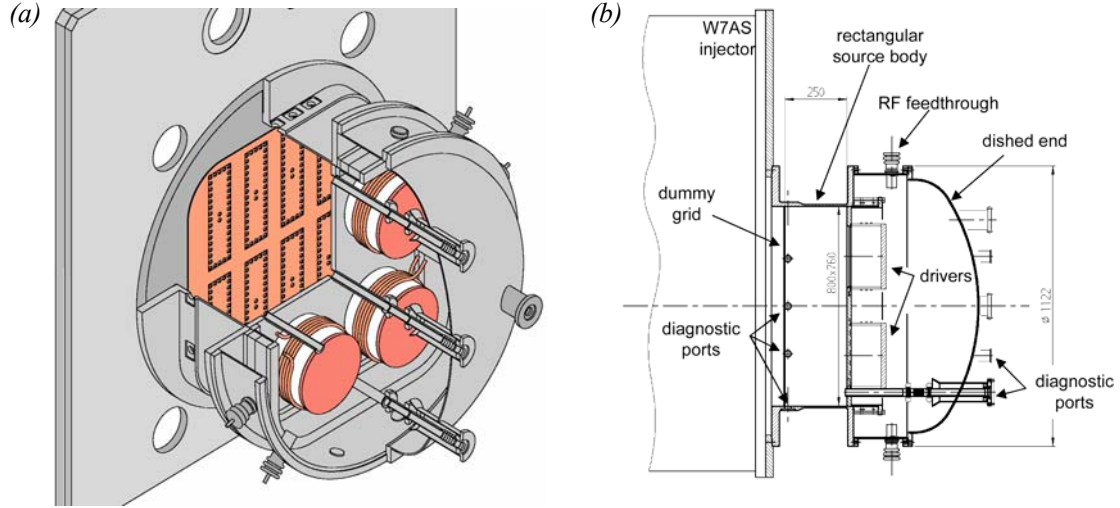


Figure 2.1: RADI ion source at IPP: (a) Isometric view; (b) Vertical cross section

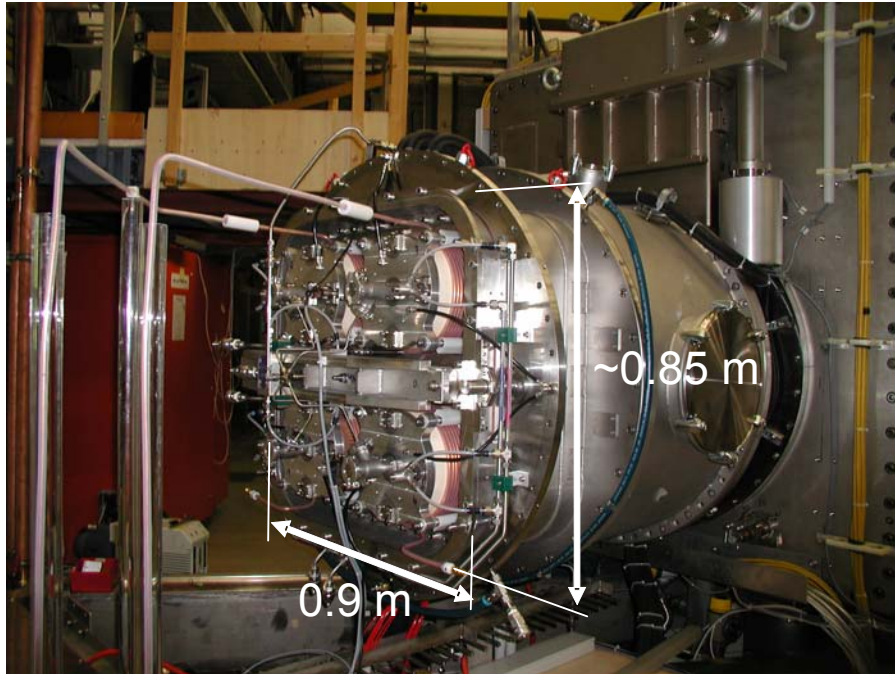


Figure 2.2: Overview of the RADI experiment.

2.2 The RADI ion source

The RADI ion source, shown in Figg. 2.1 and 2.2, is devoted to test the geometry and the optimum number of drivers, to test an ITER-like RF circuit, to investigate the Cs dynamics as well as to demonstrate the required homogeneity of large RF plasmas. The experience of the operation of this source will then be used in the final design of the ITER RF source [2, 3, 4].

This source is designed for pulsed operation (pulse length < 10 s) as the available power supplies can not work continuously. It consists of a rectangular source body and a drivers plate, equipped with 4 drivers. To simulate the ITER vacuum immersed source, the driver region of the source can be enclosed by a feedthrough ring and a dished end. It is equipped with a dummy plasma grid with 5 adjustable slits to simulate conductance of the ITER extraction system.

To get high D^- densities in the extraction region it is necessary to seed the source with cesium. From experiments on BATMAN, it is known that the cesium distribution inside the source strongly depends on the temperature of the source. Therefore its temperature will be controlled by regulating the temperature of the cooling water in the range of 20 to 60°C.

The expansion of the plasma from the drivers into the expansion vessel will largely determine the plasma uniformity. The source is therefore equipped with a movable backplate allowing to vary the distance between drivers and plasma grid. The sidewalls of the expansion volume are kept free from mechanical structures thus allowing freedom in optimizing the configuration of the confinement magnets which affect the plasma flow to the side wall.

The source body is actively water cooled for having a good temperature control of the source. The wall of the source body is made of 6mm thick stainless steel sheets with deep drilled cooling channels. The manifolds are integrated in the front and back flanges of the source. For a better distribution of the thermal load, a 1mm layer of copper is electro deposited on the inner surface. The source body has several diagnostic ports near the extraction region.

2.2.1 Former estimations on the power loads on RADI ion source

In the following, some estimates on the power balance are given, based on the experimental data. 100 kW/driver is the agreed assumption for the input power, that leads to 400 kW for the whole RADI ion source.

The main part of this power should be deposited on the Faraday shield (as sum

of direct heat load and joule effect of the eddy currents). An estimate of 30 to 60% of the total power was made, by measuring the temperature of the Faraday shields by means of thermocouples.

A preliminary evaluation of the Joule effect of the RF current flowing in the coils has given a value of 2.5 kW/coil, and 10 kW for the four drivers.

The power density on the source case has been measured at IPP on BATMAN as 30 kW/m²: the RADI source has a case surface 2 times bigger and an input power 4 times bigger. Then a power density of 60 kW/m² could be supposed, for an agreed total of around 120 kW on the whole case.

Summing up, then, a preliminary power balance shows 400 kW input power, that result in 200 kW on the FS, 120 kW on the source case, few tens on the HF circuit, the rest on plasma grid and other components (drivers back cover, for instance), apart from the power absorbed by the plasma itself. Dedicated measures and evaluations, reported in the next paragraph, aim at giving more precise information on the power balance of the source.

2.3 A new method for the post-processing of calorimetric measurements

Estimating precisely the thermal power load deposited on the components of the Radio Frequency Ion Source can be useful to dimension the cooling systems and also to calculate how much power is transferred to the plasma inside the source, giving important indications for the design of new ion sources. In order to increase the precision of the estimations, a new method based on the analytical integration of the calorimetric measures has been developed.

This method considers the flow and temperature data of the component inlet and outlet cooling water. The graph in Fig. 2.3 shows a simplified scheme of the general behaviour of the inlet and outlet cooling temperatures for a component. Three phases can be identified:

1. **Phase I:** before the pulse, when the outlet water temperature is the same as the inlet water temperature;
2. **Phase II:** the pulse, that lasts a time ($t_1 - t_0$), where the outlet water temperature grows quickly;
3. **Phase III:** after the pulse, when the outlet water temperature decreases slowly until it becomes same as the inlet one.

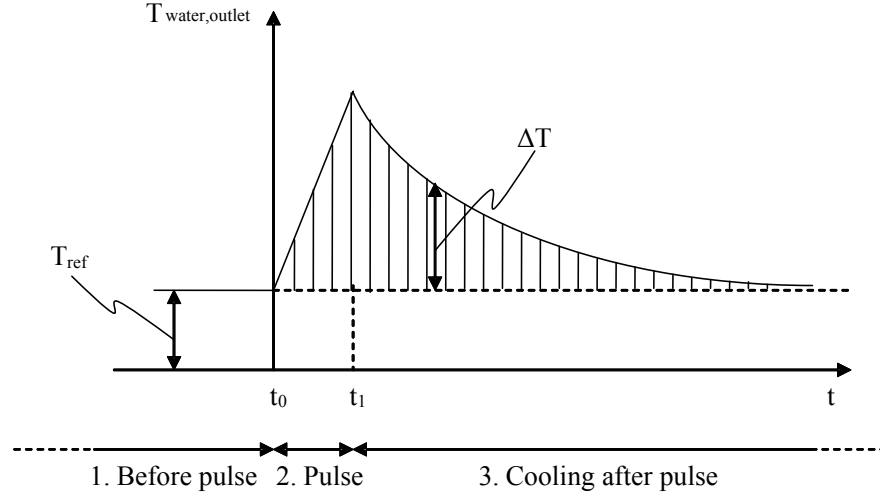


Figure 2.3: Simplified scheme to calculate the energy given by a pulse to a component.

With a simplified analytical model, it can be demonstrated that the outlet temperature tends asymptotically to the reference value T_{ref} , which is defined as the inlet water temperature at the beginning of the pulse.

The thermal energy coming from the pulse and stored in the component $e(t)$ can be written in function of time as:

$$e(t) = m \cdot C_{comp} \cdot \Delta T(t) \quad (2.1)$$

where m is the component mass, C_{comp} is the component specific heat (depending on the component material) and $\Delta T(t)$ is the difference between the component average temperature and the water average temperature.

The heat exchanged by convection between the component and the cooling water $q(t)$ can be written in function of time as:

$$q(t) = \alpha_{av} \cdot A_{exchange} \cdot \Delta T(t) \quad (2.2)$$

where α_{av} is the average Convection Heat Transfer (CHT) coefficient, $A_{exchange}$ is the total interface area between the component and the cooling water (given by the cooling circuit), and $\Delta T(t)$ is the thermal energy stored in the component.

In order to simulate the behaviour of this system in function of time, the equivalent electric model can be considered (see Fig. 2.4), where the component can be simulated with a capacitor working between the temperatures T_{ref} and $T_{ref} + \Delta T(t)$, and the cooling system with a resistor working between the same temperatures.

Inverting Eq. 2.1, the $\Delta T(t)$ can be written in function of the thermal energy

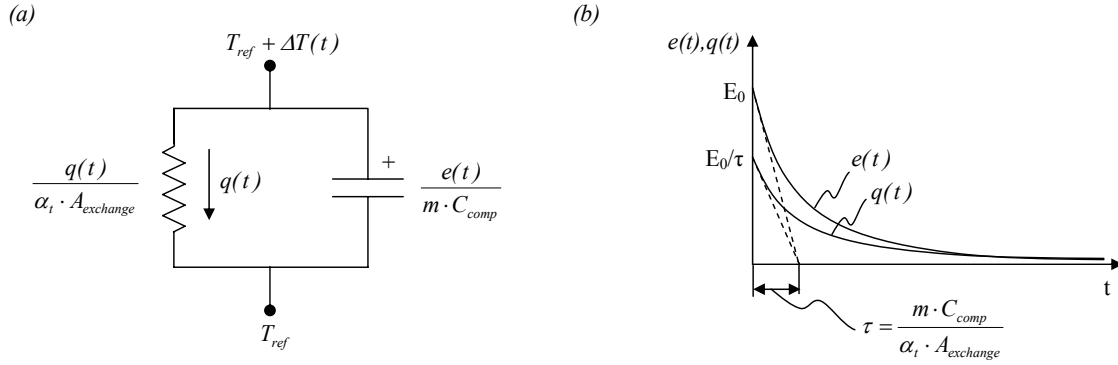


Figure 2.4: Equivalent electric model: (a) Equivalent circuit; (b) Exponential decay of the energy and heat curves.

stored in the component:

$$\Delta T(t) = \frac{e(t)}{m \cdot C_{comp}} \quad (2.3)$$

Inverting Eq. 2.2, the same quantity $\Delta T(t)$ can be written in function of the heat exchanged between water and copper:

$$\Delta T(t) = \frac{q(t)}{\alpha_{av} \cdot A_{exchange}} \quad (2.4)$$

As the heat exchanged is the first derivative of the stored energy against time,

$$q(t) = \frac{de(t)}{dt} \quad (2.5)$$

eq. 2.4 can be written as:

$$\Delta T(t) = \frac{1}{\alpha_{av} \cdot A_{exchange}} \cdot \frac{de(t)}{dt} \quad (2.6)$$

Applying the Kirchhoff's Voltage Law (KVL) to the circuit in Fig. 2.4a, we obtain:

$$\frac{1}{m \cdot C_{comp}} \cdot e(t) - \frac{1}{\alpha_{av} \cdot A_{exchange}} \cdot \frac{de(t)}{dt} = 0 \quad (2.7)$$

This differential equation has the solution:

$$e(t) = E_0 \cdot \exp\left(-\frac{t \cdot \alpha_{av} \cdot A_{exchange}}{m \cdot C_{comp}}\right) \quad (2.8)$$

where E_0 is the total energy due to the pulse and stored in the component (normally reached immediately after the pulse), t is the time, α_{av} is the average CHT coefficient,

$A_{exchange}$ is the heat exchange area, m is the component mass and C_{comp} is the average specific heat of the component. The equation can be written also as

$$e(t) = E_0 \cdot \exp\left(-\frac{t}{\tau}\right) \quad (2.9)$$

where

$$\tau = \frac{m \cdot C_{comp}}{\alpha_{av} \cdot A_{exchange}}$$

is the time constant of the system. The exponential decay of the energy $e(t)$ and a graphical representation of the time constant τ is graphically shown in Fig. 2.4b.

Deriving this expression, the heat exchanged between the component and the cooling water $q(t)$ can be calculated as

$$q(t) = E_0 \cdot \frac{\alpha_{av} \cdot A_{exchange}}{m \cdot C_{comp}} \cdot \exp\left(-\frac{t \cdot \alpha_{av} \cdot A_{exchange}}{m \cdot C_{comp}}\right) \quad (2.10)$$

which can be written as

$$q(t) = E_0 \cdot \frac{1}{\tau} \cdot \exp\left(-\frac{t}{\tau}\right) \quad (2.11)$$

Summarizing, it is demonstrated that $e(t)$ and $q(t)$ have an exponential decay, with a time constant proportional to the mass m and specific heat C_{comp} of the component, and inversely proportional to the average CHT coefficient α_{av} and heat exchange area $A_{exchange}$. The outlet water temperature T_{out} can be calculated as:

$$T_{out} = T_{in} + \frac{q(t)}{\dot{m} \cdot C_{water}} \quad (2.12)$$

where T_{in} is the inlet water temperature, $q(t)$ is the heat exchanged, $\dot{m}$ is the total cooling water flow and C_{water} is the water specific heat. Substituting Eq. 2.10, we find:

$$T_{out} = T_{in} + \frac{E_0 \cdot \alpha_{av} \cdot A_{exchange}}{m \cdot C_{comp} \cdot \dot{m} \cdot C_{water}} \cdot \exp\left(-\frac{t \cdot \alpha_{av} \cdot A_{exchange}}{m \cdot C_{comp}}\right) \quad (2.13)$$

or, using the time constant

$$T_{out} = T_{in} + \frac{E_0}{\tau \cdot \dot{m} \cdot C_{water}} \cdot \exp\left(-\frac{t}{\tau}\right) \quad (2.14)$$

Hence, it is demonstrated that also the outlet temperature T_{out} has an exponential decay. Moreover, it is explicated the link between the energy absorbed by the component during a pulse and T_{out} .

As a consequence, in a real cooling system, it is not easy to understand where to consider the end of the phase III.

Utilizing the theoretical model above, a very precise way to measure the total energy absorbed by the component can be given by the relation:

$$E_{comp} = \int_{t_0}^{+\infty} \dot{m} \cdot C_{water} \cdot (T_{out} - T_{in}) \cdot dt \quad (2.15)$$

that can usually be simplified to

$$E_{comp} = \dot{m} \cdot C_{water} \int_{t_0}^{+\infty} (T_{out} - T_{in}) \cdot dt \quad (2.16)$$

because the variation of the water flow and specific heat is usually negligible between water at inlet and outlet temperatures.

The main advantages of the new method are the following:

- The function $(T_{out} - T_{in})$ can be integrated also analytically, because it is known that it has an exponential form in the phase III.
- The integration can be made also recording only a little part of the phase III. This is a usual situation, because the cooling is normally very slow (compared to the pulse time scale) and in any case it is impossible to record the whole phase III, because in principle it is infinitely long.

2.3.1 Calorimetric measurements on ideal and non-ideal pulses

As a generic example, a simplified cooling system of a component is reported in Fig. 2.5a. This component is heated with a certain power load during the pulse, and cooled down by a certain water flow $\dot{m}$. The inlet and outlet temperatures T_{in} and T_{out} are recorded during and after the pulse, and their values during a typical experimental session is shown in Fig. 2.5b.

Considering an ideal pulse (pulse #1), the conventional approach can be used, evaluating the power load with a numerical integration of the power taken away from water:

$$E_{\#1} = \dot{m} \cdot C_{water} \int_{t_1}^{t_2} \Delta T \cdot dt \quad (2.17)$$

where $\Delta T = T_{out} - T_{in}$ and the integral $\int_{t_1}^{t_2} \Delta T \cdot dt$ is represented by the red area in Fig. 2.5b.

Let us consider now the pulse #3. This is not an ideal pulse for two reasons:

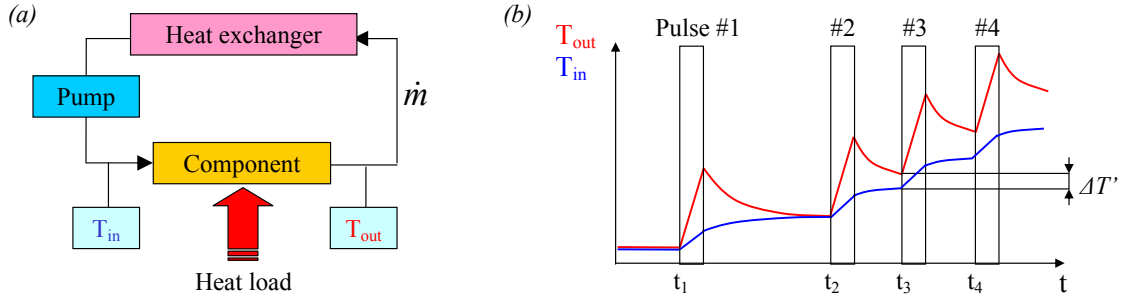


Figure 2.5: Generic application of the new analytical approach: (a) Cooling scheme of a generic component; (b) Scheme of a generic experimental session

- The inlet and outlet temperatures of the water are different at the beginning of the pulse ($\Delta T' \neq 0$). This is due to the fact that the heat loads deposited during the previous pulses (#1 and #2 in this example) are not swallowed yet.
- The cooling down phase (phase III) is not completely recorded, because of another pulse is made before the completion of this phase.

In this case, that represent the majority of the real pulses, the new approach can be used, with an analytical integration of the exponential function interpolating the available data of the function $(\Delta T - \Delta T')$.

$$E_{\#3} = \dot{m} \cdot C_{water} \int_{t_3}^{+\infty} (\Delta T - \Delta T') \cdot dt \quad (2.18)$$

This can be done also considering the very first seconds of the phase III, because it is demonstrated that the function $(\Delta T - \Delta T')$ has in principle an exponential shape.

2.4 Calorimetric measurements on the RADI source

The measurements are done in the RADI ion source at IPP (shown in Fig. 2.2), that for some aspects (concept, layout and dimensions) is at the moment the ion source test facility most similar to the one foreseen for the ITER NBI.

2.4.1 Description of the cooling system of the RADI ion source

A scheme of the cooling system of the Faraday shield and the Radio Frequency coils in the RADI source is shown in Fig. 2.6. The following signals are recorded during and after a pulse, in order to evaluate the power load on these two components:

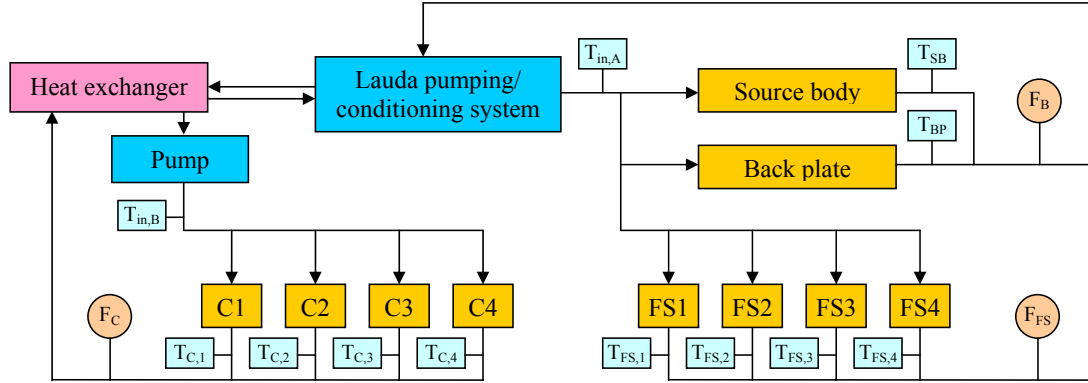


Figure 2.6: Scheme of the cooling system of the Faraday shield and the RF coils in the RADI source

- The measures given by the flow sensors F_{FS} (flow on the Faraday shield) and F_C (flow on the Radio Frequency coils);
- The measures given by the thermo-couples $T_{FS,1}$, $T_{FS,2}$, $T_{C,1}$, $T_{C,2}$ (outlet temperature from the Faraday shields and RF coils cooling circuits).

2.4.2 Description of the diagnostic system for calorimetric measures

The flow sensors are of the turbine type, shown in Fig. 2.7. These sensors can provide accurate measurement of instantaneous or total flow rates of low viscosity fluids such as water.

The main characteristics of these sensors are:

- fast response time and high resolution,
- wide temperature operating range,
- resistance to contamination with solids.

The temperature sensors are thermo-couples of the type NiCr(+)/Ni(-). They are completely surrounded by the water and insulated from the tube surface, in order to precisely measure the water temperature.

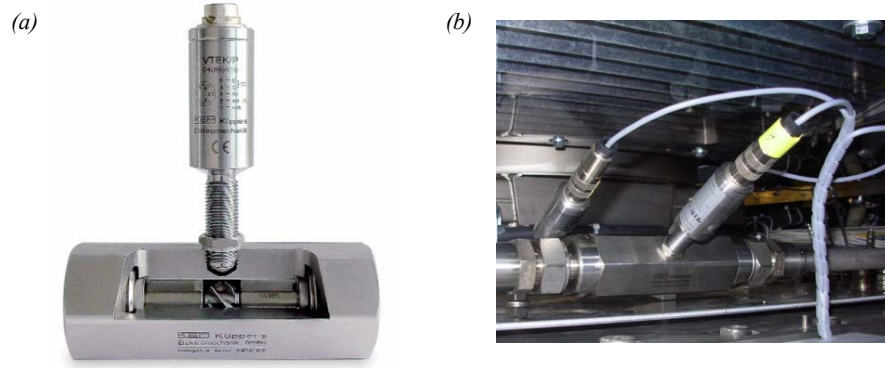


Figure 2.7: Turbine flow meter: (a) Section view; (b) Sensor mounted in RADI

2.4.3 Application of the new method for the post-processing of the calorimetric measurements on the Faraday shields

Fig. 2.8 shows a pulse in the RADI ion source. It lasts 5.6 s, with two active drivers and a power input (measured at the power supplies) that is $P_{input}=35$ kW for every driver. This pulse is considered as an example for the application of the new method described in paragraph 2.3. Making the hypothesis that the cooling of the Faraday Shields is given only by the water cooling system and not by thermal conductance on the solid parts, the amount of power going to the Faraday Shield of the driver can be estimated from the temperature of the cooling water.

For a Faraday shield, the power taken away from the water can be estimated as:

$$P_{water,FS} = \dot{m}_{FS} \cdot C_p \cdot \Delta T \quad (2.19)$$

where $\Delta T = \Delta_{water,outlet} - \Delta_{water,inlet}$. So, the total energy taken away from the FS after the pulse can be estimated as:

$$\begin{aligned} E_{water,FS} &= \int_{t_0}^{+\infty} \dot{m}_{FS} \cdot C_p \cdot \Delta T \cdot dt \\ &= \dot{m}_{FS} \cdot C_p \int_{t_0}^{+\infty} \Delta T \cdot dt \end{aligned} \quad (2.20)$$

The cooling phase is not completely recorded. Nevertheless, the outlet water temperature in function of time can be modelled with an exponential curve, and extrapolated till the complete cooling, like shown in Fig. 2.9. The phase 2 (during the pulse) is in this case negligible, by the energetic point of view.

So the integral of the temperature in the domain of time can be calculated as:

$$\begin{aligned} \int_{t_0}^{+\infty} \Delta T \cdot dt &= \left[-\frac{7.08}{0.0115} \cdot e^{-0.0115 \cdot t} \right] \\ &= \frac{7.08}{0.0115} \cdot e^{-0.0115 \cdot 6.3} \\ &= 572.6 \text{ } ^\circ\text{C} \cdot \text{s} \end{aligned} \quad (2.21)$$

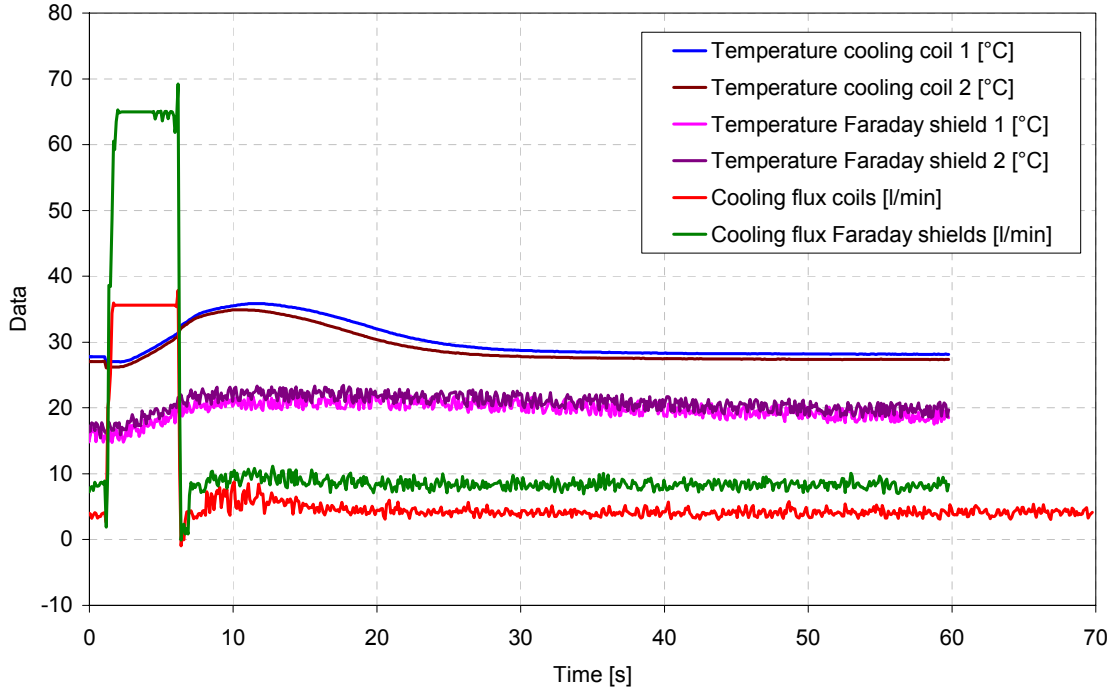


Figure 2.8: Record of a pulse in the RADI experiment.

And the energy taken away from water is estimated as:

$$\begin{aligned} E_{water,FS} &= 0.0315 \frac{kg}{s} \cdot 4188 \frac{J}{kg \cdot ^\circ C} \cdot 572.6 ^\circ C \cdot s \\ &= 75538 J \end{aligned} \quad (2.22)$$

where the flux is measured by a flow meter and the specific heat of water is calculated for the average temperature of 20 °C. If the following hypothesis, which considers negligible the cooling by thermal conduction and irradiation, is considered:

$$E_{water,FS} = E_{FS} \quad (2.23)$$

then the power to the FS during the pulse can be estimated as:

$$\begin{aligned} P_{FS} &= \frac{E_{FS}}{t_1 - t_0} \\ &= \frac{75538 J}{5.1 s} \\ &= 14.8 kW \end{aligned} \quad (2.24)$$

Hence in this case the power going to warm up the FS is approximately the 42% of the total input power.

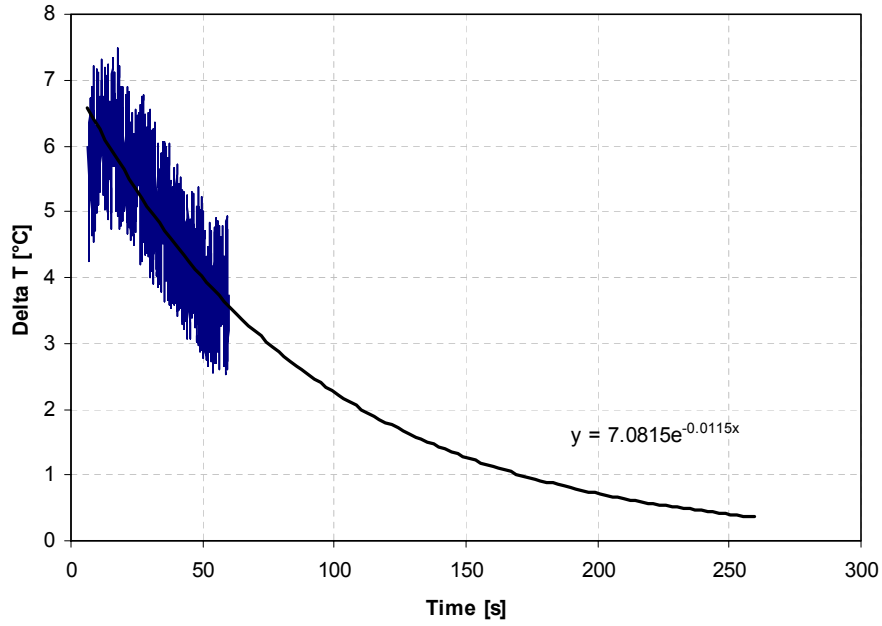


Figure 2.9: Water outlet temperature from the Faraday shield number 1. The signal is modeled with an exponential curve, and an extrapolation is made till the complete cooling.

2.4.4 Application of the new method for the post-processing of the calorimetric measurements on the Radio Frequency coils

The same procedure can be used to estimate the thermal power that goes to the coils. In this case, the highest value of the temperature is reached about 4 seconds after the pulse, and the cooling phase is much faster (see Fig. 2.10).

As the cooling phase is completely recorded, a numerical integration can be done in this case.

$$\int_{t_0}^{+\infty} \Delta T \cdot dt = 118 \text{ }^{\circ}\text{C} \cdot s \quad (2.25)$$

The energy taken away from water is estimated as:

$$\begin{aligned} E_{water,FS} &= 0.014 \frac{kg}{s} \cdot 4186 \frac{J}{kg \cdot ^{\circ}\text{C}} \cdot 118 \text{ }^{\circ}\text{C} \cdot s \\ &= 6915 \text{ J} \end{aligned} \quad (2.26)$$

where the flux is measured by a flow meter and the specific heat of water is calculated for the average temperature of 30 °C. Also in this case, the thermal conduction and irradiation is considered negligible:

$$E_{water,coil} = E_{coil} \quad (2.27)$$

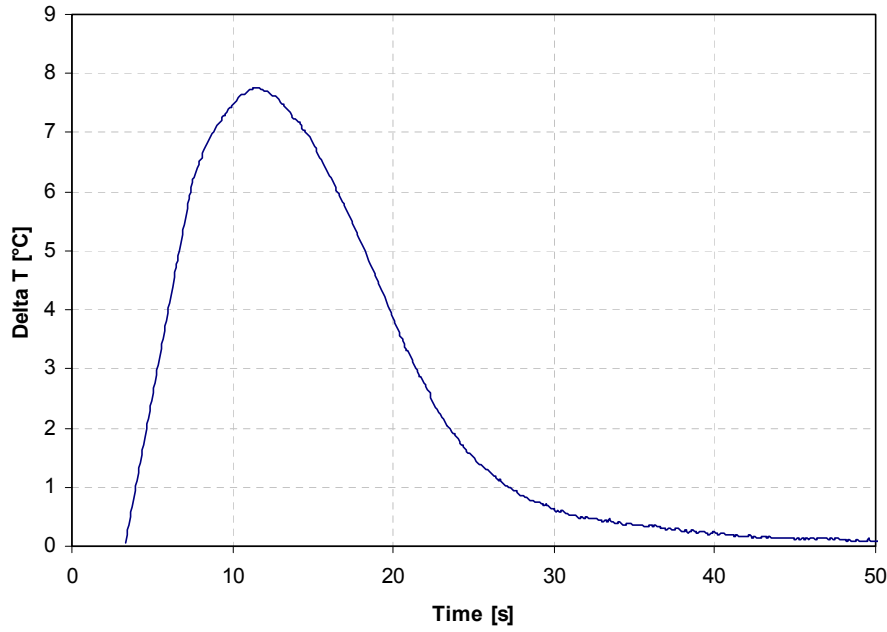


Figure 2.10: Water outlet temperature from coil 1, till the complete cooling. The phase 1 is depurated from the offset given by the RF magnetic field.

The power to the FS during the pulse can be estimated as:

$$\begin{aligned}
 P_{coil} &= \frac{E_{coil}}{t_1 - t_0} \\
 &= \frac{6013 \text{ J}}{5.6 \text{ s}} \\
 &= 1.23 \text{ kW}
 \end{aligned} \tag{2.28}$$

Hence in this case the power going to warm up the coils is approximately the 3.5% of the total input power.

2.5 Integration with numerical analyses

The calorimetric measurements made as the previous paragraph can be utilized to estimate the temperature, stresses and deformations of the Faraday shield in every point. In order to make this, it is necessary to build a FEM model of the component under consideration.

The model should be suitably simplified with respect to the real component design, in order to have a good compromise between these two targets:

- To obtain reliable results, by considering all the details that influence the thermo-mechanical behavior of the component.

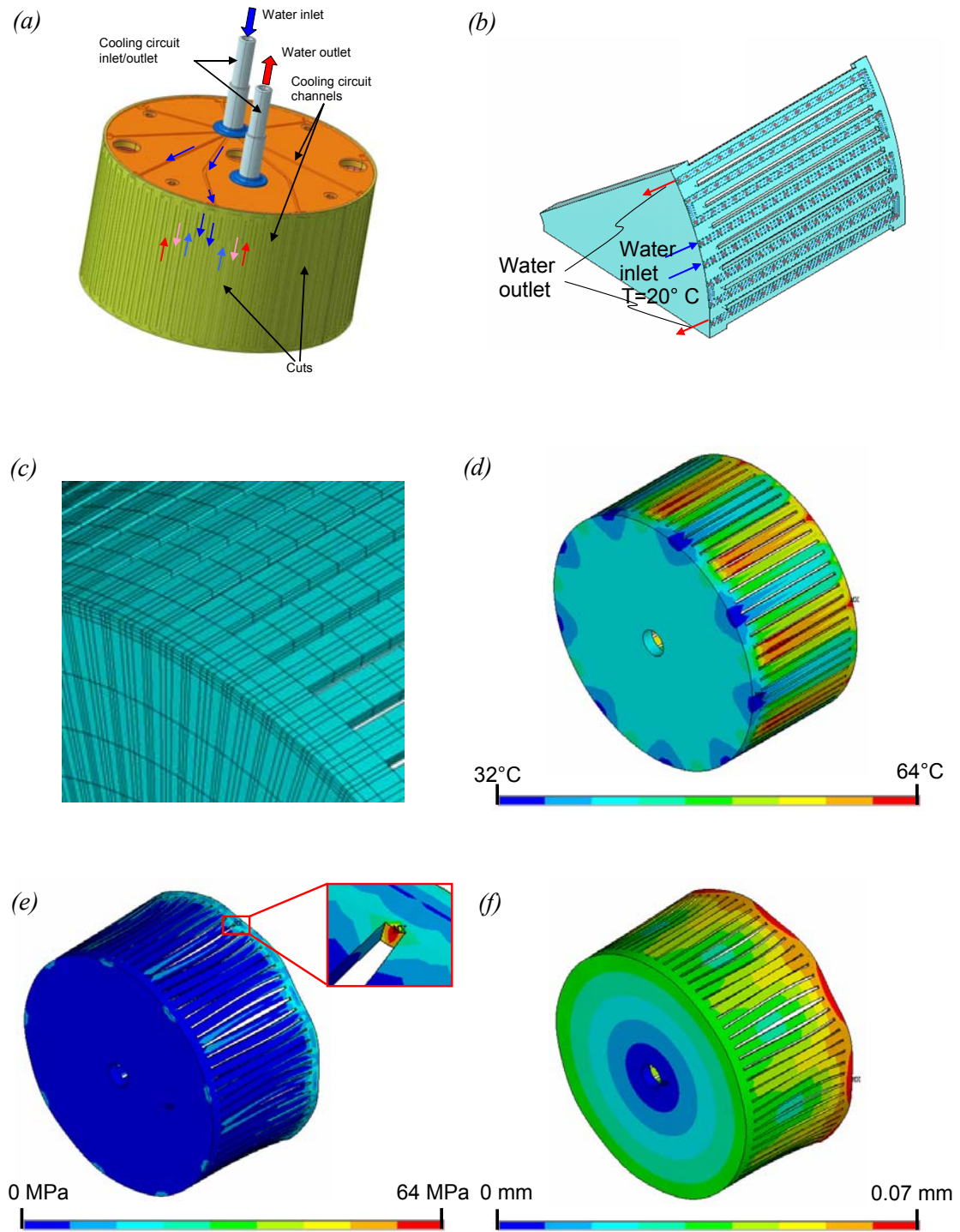
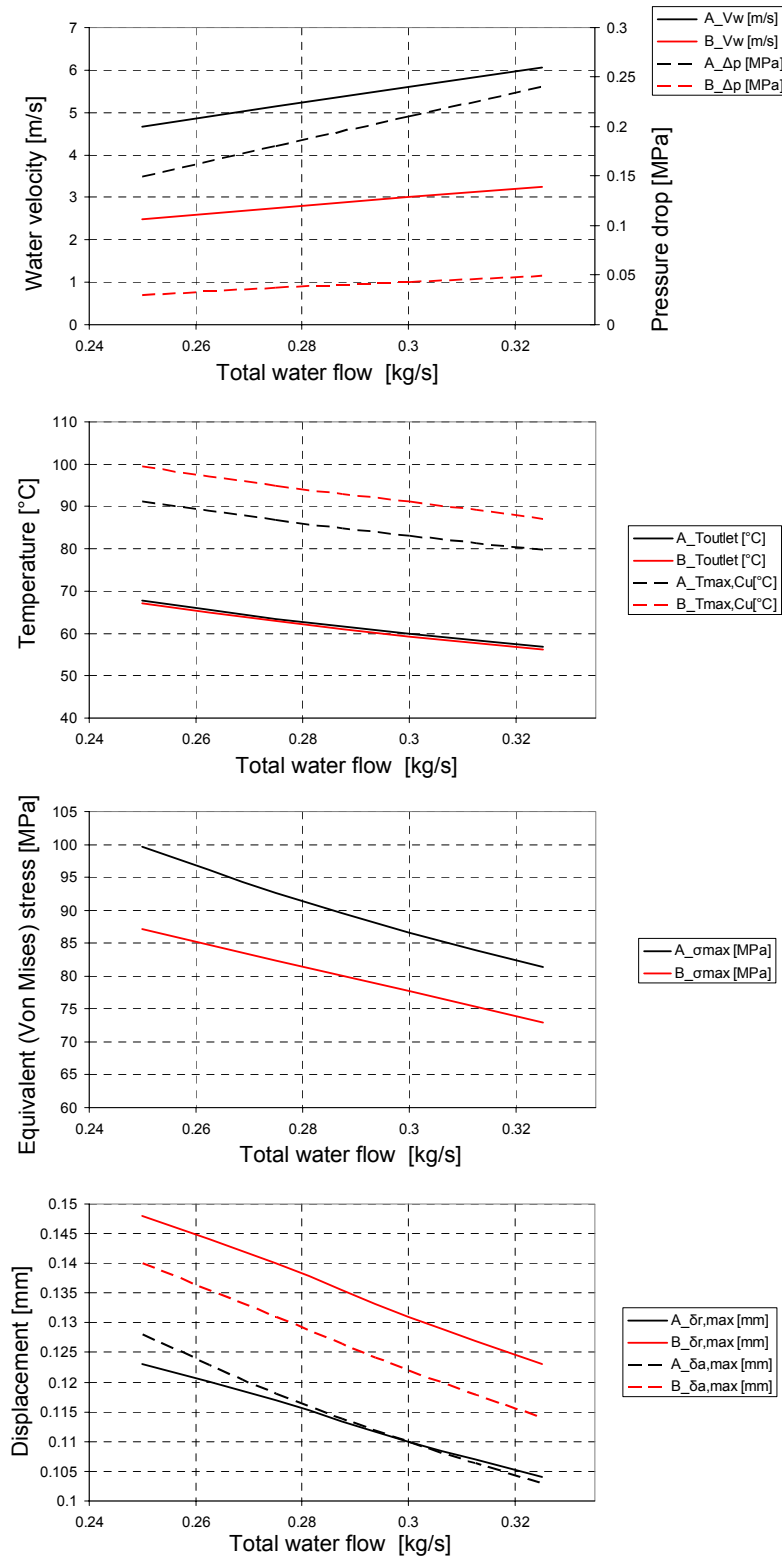


Figure 2.11: Thermo-mechanical model of the Faraday shield: (a) CAD model; (b) 40° sector showing two cooling channels; (c) Mesh detail; (d) Temperature contour plot; (e) Equivalent (Von Mises) stress contour plot; (f) Radial displacement contour plot.

**Considerations:**

Water velocity in geom. B is about 55% of the one in geom. A. Consequently, the pressure drop in geom. B is about 20% of the one in geom. A. The pressure drop is very high in geometry A (about 2 bar).

Considerations:

Outlet water temperature is the same with two geometries, because it depends on the heat load and water velocity. The maximum temperature on the shield is about 10% higher in geometry B. This is due to the fact that in this case the convection coefficient is lower, because of a lower Reynolds number.

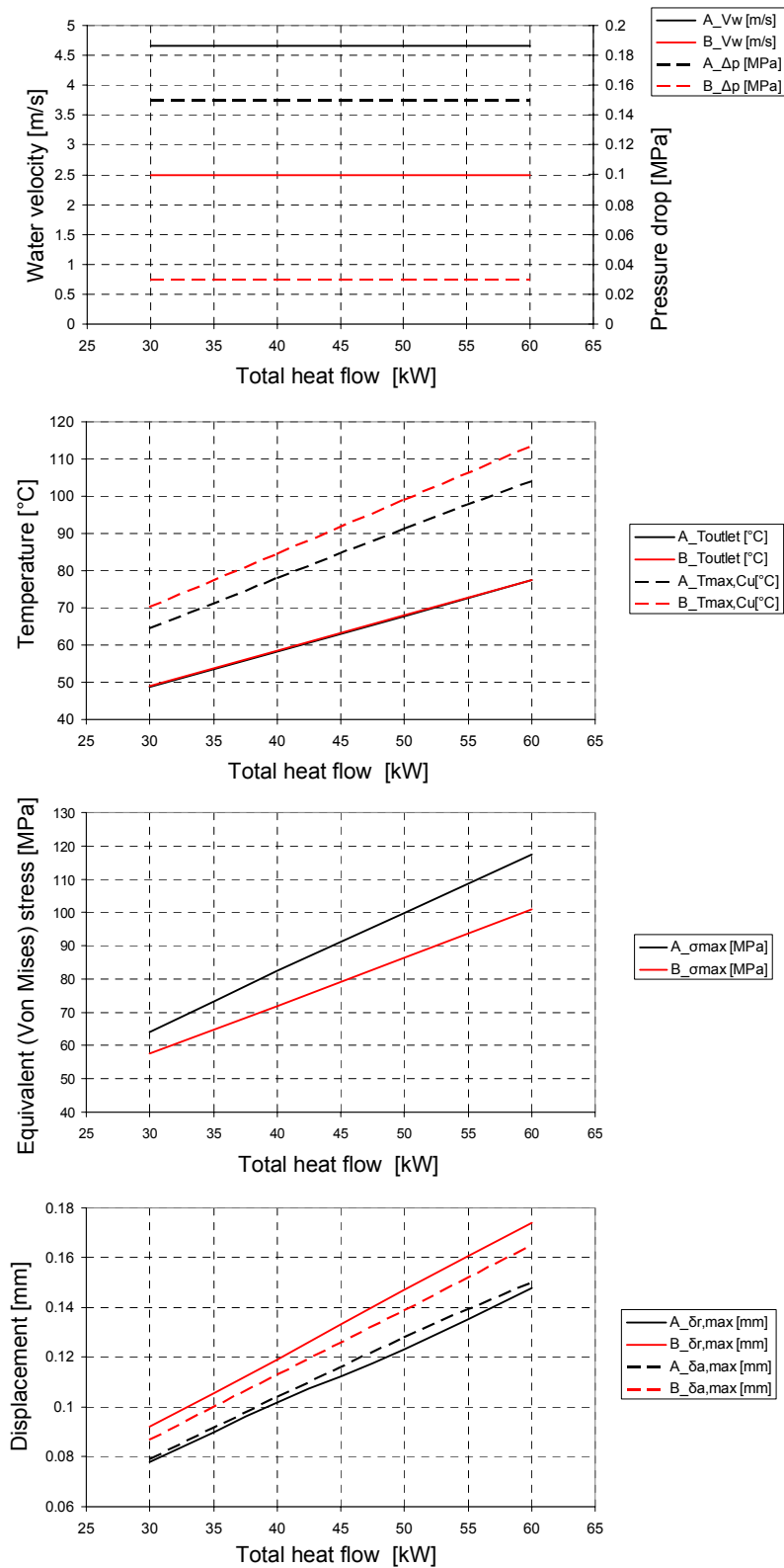
Considerations:

The maximum equivalent stress in the shield is 15% higher in geometry A, even if the peak temperature is 10% lower. This could be motivated by the fact that in geometry A there is a higher gradient of temperature, because the cooling effect is concentrated near the tubes. In geometry B, the cooling effect is better distributed.

Considerations:

The maximum radial and axial displacement are higher in geometry B (radial +20%, axial +10%) than in geometry A. This is because in geometry B the temperature is meanly higher.

Figure 2.12: Water flow sensitivity analysis, with a 50 kW total heat load.

**Considerations:**

Water velocity and pressure drop do not depend on heat load.
With regard to geometry sensitivity, the observations of Fig. 57 are still valid.

Considerations:

Outlet water temperature increases with heat load of about $0.9 \frac{^{\circ}\text{C}}{\text{kW}}$.
Maximum temperature on copper increases with heat load of about $1.4 \frac{^{\circ}\text{C}}{\text{kW}}$.

Considerations:

The maximum equivalent stress on the shield increases with heat load of about $1.4 \frac{\text{MPa}}{\text{kW}}$.

Considerations:

The maximum axial and radial displacements on the shield increase with heat load of about $0.0026 \frac{\text{mm}}{\text{kW}}$.

Figure 2.13: Heat load sensitivity analysis, with a 0.25 m/s water flow.

- To have a reasonable number of nodes and elements. Hence all the details that have a small (or null) influence on the thermo-mechanical behavior of the component can be neglected.

The heat loads applied to the FEM model can be extrapolated from the experimental measurement. In this case, a range of power ranging around the measured value on the RADI test facility is considered.

Fig. 2.11 show for example the Faraday shield FEM model and the contour plots of the significant outputs (temperature, stress and deformations), with a 30 kW total heat flux and a 0.25 kg/s total water flow.

2.5.1 Sensitivity analysis and robust design philosophy

Several static simulations must be performed to investigate the effect of various parameters (heat flux, water flow, model geometry) on the component behaviour, according to the robust design philosophy [5].

In these sensitivity analyses, an optimized geometry B (with a cooling channels cross section of $3 \times 1 \text{ mm}^2$) is compared with the reference one A B (with an enlarged cross section of $4 \times 1.4 \text{ mm}^2$). Figs. 2.12 and 2.13 report the sensitivity analyses focusing on the influence of a variation of heat flux or water flow on the main operating parameters (temperature, stress and radial deformation).

2.5.2 Indications for the design optimization

The numerical calculations confirm the experimental observation that the FS is a critic component from the thermo-hydraulic point of view. A suggestion for the actions to make in order to ameliorate the thermo-mechanical behavior for the component is given by the FEM analyses.

In the present case, the following considerations can be done:

- It is fundamental to verify the actual power balance on the source and in particular the heat load fraction on the FS; dedicated measurement should be implemented in IPP test beds to take care of this issue.
- Likely, it will be necessary to enlarge the cooling channels, with respect to current geometry, in order to optimize the functional parameters, i.e. allow a larger water flow (improvement of heat load evacuation) and/or reduce the pressure drop. The geometry B is here proposed as an improvement on the reference solution.

Bibliography

- [1] E. Speth, H.D. Falter, P. Franzen, U. Fantz, M. Bandyopadhyay, S. Christ, A. Encheva, M. Fröschle, D. Holtum, B. Heinemann, W. Kraus, A. Lorenz, Ch. Martens, P. McNeely, S. Obermayer, R. Riedl, R. Süss, A. Tanga, R. Wilhelm and D. Wunderlich. *Overview of the RF source development programme at IPP Garching*. Nucl. Fusion 46 (2006) 220-238.
- [2] U. Fantz, H.D. Falter, P. Franzen, M. Bandyopadhyay, B. Heinemann, W. Kraus, P. McNeely, R. Riedl, E. Speth, A. Tanga and R. Wilhelm. *Diagnostics of the cesium amount in an RF negative ion source and the correlation with the extracted current density*. Fusion Eng. Des. 74 (2005) 299-303.
- [3] P. Franzen, H.D. Falter, E. Speth, W. Kraus, M. Bandyopadhyay, A. Encheva, U. Fantz, Th. Franke, B. Heinemann, D. Holtum, C. Martens, P. McNeely, R. Riedl, A. Tanga and R. Wilhelm. *Status and plans for the development of a RF negative ion source for ITER NBI*. Fusion Eng. Des. 74 (2005) 351-357.
- [4] W. Kraus, P. Franzen, B. Heinemann, E. Speth and O. Vollmer. *Recent developments of long pulse RF ion sources for NBI systems*. Fusion Eng. Des. 56-57 (2001) 499-503.
- [5] M.S. Phadke. *Quality Engineering Using Robust Design*. Prentice Hall PTR, 1989.

Chapter 3

Methods for the design optimization of an ion source

SUMMARY - In this chapter, the operating conditions of the back plate and drivers of high power ion source (heat loads, sputtering, other considerations) are investigated, and different options for the design of the main components (mechanical design, cooling scheme, coating) are compared and discussed. In particular, the possibility of a tungsten or molybdenum coating for the back plate and Faraday shields (made by copper) is evaluated, in terms of advantages and disadvantages. In order to make this, a new procedure for the evaluation of the layer removed by sputtering during operation of the source is introduced.

3.1 Introduction

A Radio Frequency (RF) Ion Source, sketched in Fig. 3.1a, is made up of a main case that acts as an expansion region for the plasma generated in the drivers (Fig. 3.1b), which are feed by RF coils. The negative ions created next to the Plasma Grid (PG) are extracted and then accelerated by electrical fields applied by means of copper grids at different voltages [1, 2, 3].

The source features a certain number of drivers (eight for ITER NBI, four for ELISE) where the RF power is input; they are connected to the source case, an open box that faces the plasma grid in the beam source assembly. The source case cross-section dimensions exceed the plasma grid aperture array dimensions, in order to minimise beam border non-homogeneities. Ports are foreseen on the back cover of the drivers and on the source case surface, in order to allow the connection of the auxiliary systems and the diagnostic accesses.

The drivers are the components where the power is transferred into the RF ion

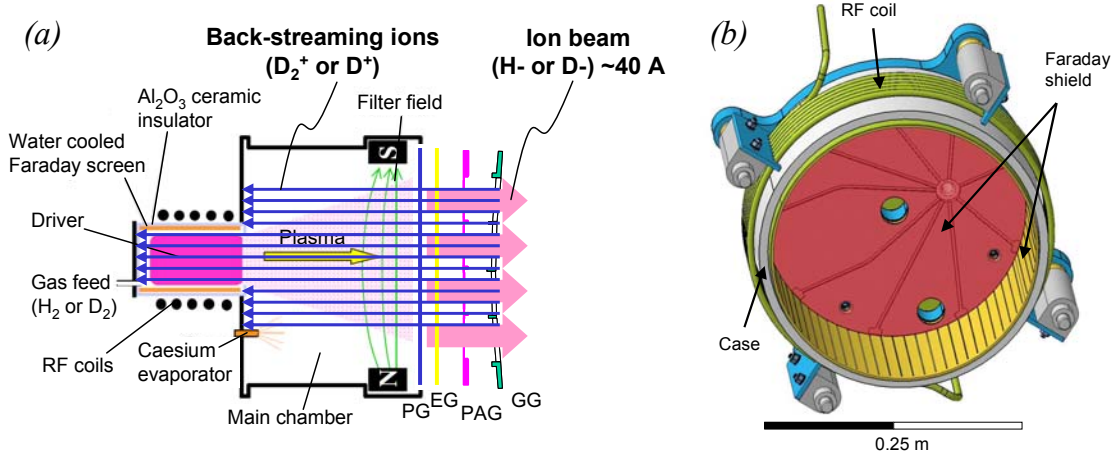


Figure 3.1: ITER NBI Beam source with RF source and SINGAP accelerator: (a) Sketch; (b) Driver detail

source and the plasma is generated. They are cylindrical chambers with about 250mm diameter, directly connected to the source case, and consisting of a lateral wall in alumina and a stainless steel back cover.

The back plate and Faraday screen of the Ion Source in the ITER Neutral Beam Injector (Fig. 3.2a) are subjected to back streaming ions coming from the accelerator and neutralizer [4]. These are high energetic hydrogen or deuterium positive ions (H^+/H^{2+} or D^+/D^{2+}), passing through the apertures of pre-accelerator grid, extraction grid and plasma grid in backward direction. As these three grids are designed to produce 1280 negative ion beamlets with a good optic, also the back-streaming beamlets have unfortunately a good optic [5]. In particular, the accelerator acts as a converging lens for the positive ion beams. The consequent heat loads on the back part of the ion source are quite high and localized in small areas, corresponding to the apertures positions on the grids. As it is visible on Fig. 3.2b, some of the 1280 hot spots are located on the back plate, some on the Faraday screen, and some at the edges between the two.

Evaluations on the consequent heat loads and on the material layers removed by sputtering will be made in this chapter considering both the design of ITER NBI and ELISE devices.

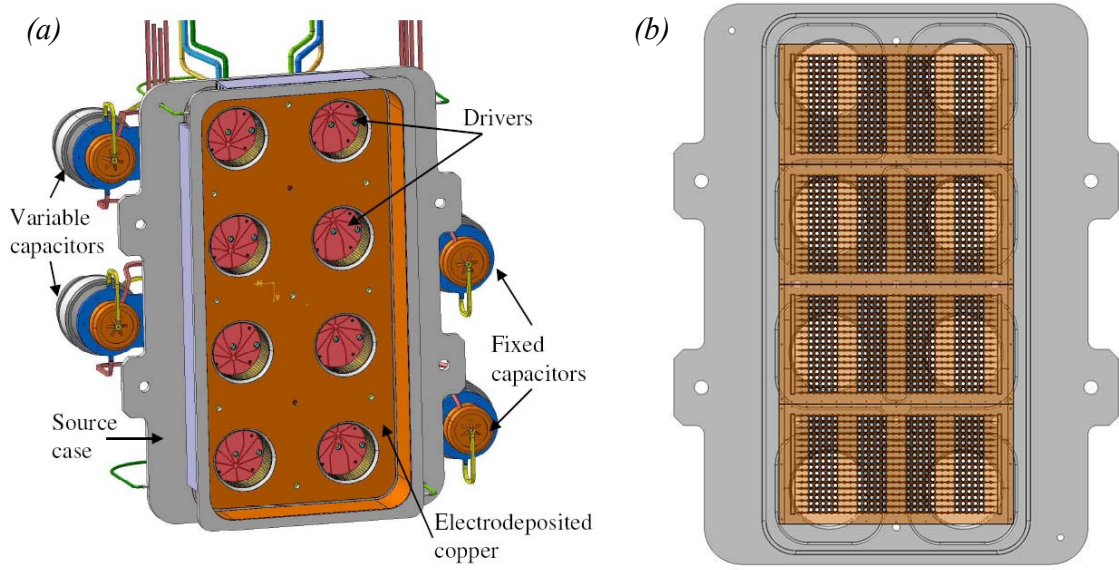


Figure 3.2: RF Ion Source back plate for ITER NBI: (a) Isometric view; (b) Plasma Grid “footprint” on back plate and Faraday screens in ITER NBI. The 1280 hot spots are heating the back plate as well as the Faraday shields

3.1.1 Back-streaming ions at the Large Helical Device

As an example of a real case, Fig. 3.3 reports two pictures of the ion source back plate of the experimental device Large Helical Device (LHD) in Japan. The neutral beam injector of LHD is 60 A/180 keV negative ion based one. The ion source is an Arc Chamber with tungsten electrodes, able to extract an ion current density up to 35 mA/cm^2 with an operating pressure of 0.3 Pa. From these pictures, it is clear where the back-streaming ion beamlets are impinging, still it is not clear what is their effect on the copper surface.

Dr. Kenichi Nagaoka, who is working on LHD NBI, reported that the spots were produced by the exposure to back-streaming ions and the different colours surrounding them show the different Cs coverage condition on the copper surface [6]. In LHD NBI, the back-streaming ions are not so powerful to melt the back-plate of the ion source (at the present status of operations). A side effect is the increase of arc current, because the back-streaming ions are heating the filaments during the beam extraction.

In the National Institute for Fusion Science (NIFS), an experiment on the beam acceleration using a SINGAP type grounded grid was also carried out by Dr. Katsuyosi Tsumori, but this scheme was not applied to the injectors in LHD. One of the main reasons was the concentration of back-streaming ions: a large positive ion

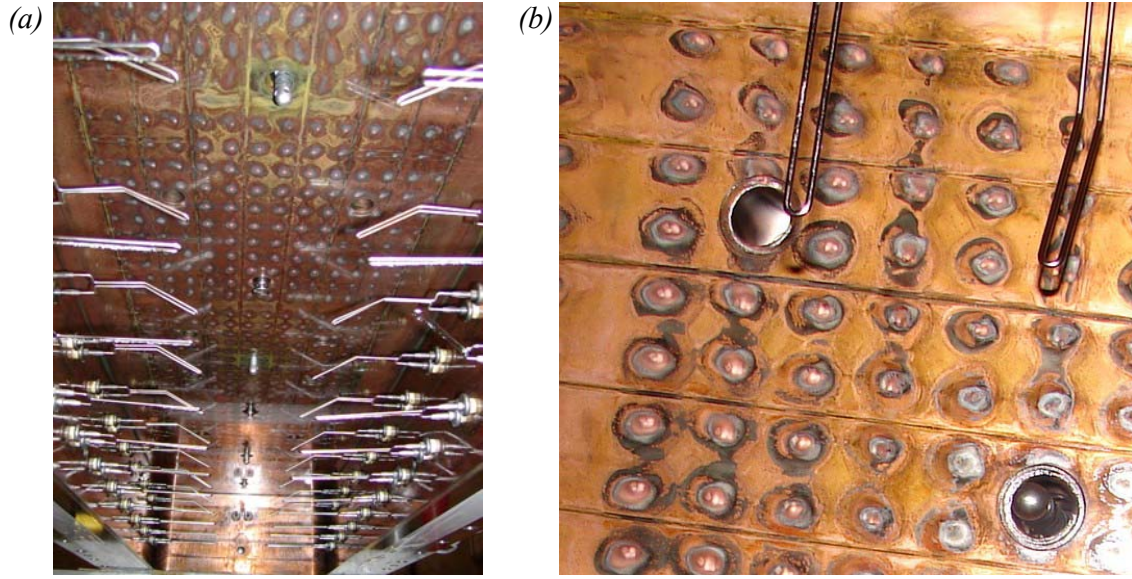


Figure 3.3: Back-streaming ions marks in the LHD ion source back plate: (a) Overview of the ion source; (b) Detail of the marks

current came back through grounded grid from the wide area down stream of the accelerator, and focused on the backside of steering grid (SG) due to a fringe effect of the acceleration field. The SG is a thin grid fixed to the extraction grid at the back side and used for optics regulation/optimization. The central part of the SG was melted due to the strongly focused back-streaming beam in this experiment, and no solutions for this problem have been found so far.

3.2 Power load estimations

The average energy of the positive ions is calculated by R. Hemsworth to be 540 keV in the SINGAP accelerator [4]. A similar estimate, referred to the MAMuG concept, is not currently available.

The composition of the back-streaming ions, calculated by G. Fubiani [5] referring to MAMuG concept, is foreseen to be:

- Deuterium bi-atomic ions, D^{2+} : 84% (~ 620 kW)
- Deuterium mono-atomic ions, D^{+} : 16% (~ 120 kW). A similar estimate, referred to the SINGAP concept, is not currently available.

The total power load due the back-streaming ions estimated for the back-plate and drivers is calculated to be in the range 880 kW for MAMuG [5]. For the SINGAP

	MAMuG	SINGAP
Average energy of back-streaming ions	/	540 keV
Composition of back-streaming ions	84% of D_2^+ 16% of D^+	/
Total power load on back plate and Faraday shields	880 kW	700 kW
Maximum power density	60 MW/m ² (in a circular area with $\varnothing \sim 2$ mm)	/

Table 3.1: Estimations regarding the back-streaming ions.

accelerator, two estimates for the total power are available: 450 kW and 700 kW [5].

The maximum power density foreseen in the hot spots (corresponding to the apertures) is calculated to be approximately 60 MW/m² in the MAMuG concept. This maximum is reached on a circular area with the diameter of approximately 2 mm, as it can be seen from Fig. 3.4. No similar estimates are currently available for SINGAP.

Tab. 3.1 summarizes the data currently available for the back-streaming positive ions in the ITER NBI back plate and Faraday shields. The bold numbers are used for the evaluations in this paper.

In the figures below, the power densities (calculated for MAMuG) corresponding to a single hot spot are shown, for three positions along the beam axis direction:

- At the Plasma Grid (Fig. 3.4a).
- Inside the Ion Source, at a 200 mm distance from the Plasma Grid in the upstream direction (3.4b). This location approximately corresponds to the Back Plate.
- Inside the Ion Source, at a 400 mm distance from the Plasma Grid in the upstream direction (3.4c). This location approximately corresponds to the Faraday screen back part.

As the back-streaming positive ion beam is converging, the maximum power density is higher in the location *b* and *c* (~ 60 MW/m²), in comparison to location *a* (~ 25 MW/m²). As a consequence of this high power density localized in very small areas, the material, if it is not properly cooled, can be subjected to localized melting.

Hence, in order to have a suitable design, the copper temperature and stresses must be kept under acceptable values. In particular, a cooling system has to be designed

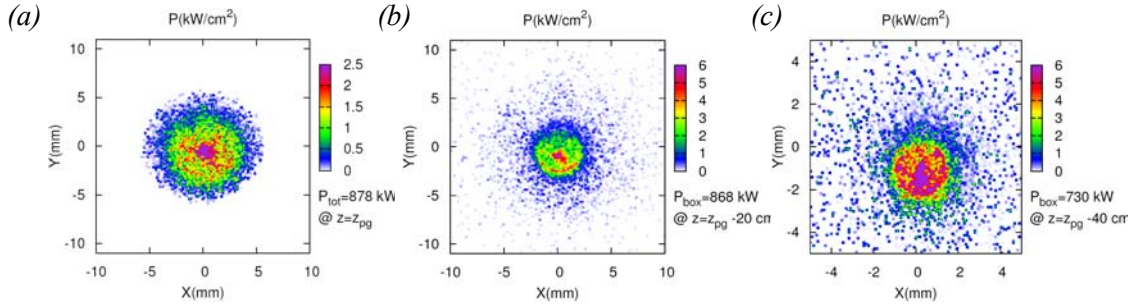


Figure 3.4: Power density given by back streaming positive ions in MAMuG: (a) at the Plasma Grid; (b) 200 mm upstream of the Plasma Grid; ; (c) 400 mm upstream of the Plasma Grid

for the ion source back plate, able to withstand such heat loads. This topic will be treated in paragraph 3.3.

Another problem that must be addressed regards the plasma composition inside the ion source: it is important to keep the impurities at the lowest possible level. The presence of these impurities - mainly metallic ions - is caused by the wall-plasma interaction (i.e. sputtering) and could degrade the performance of the ion source, for example by interacting with the cesium. Moreover, the sputtering process could remove from the surface a consistent layer of material, with the consequent risk of having water leaks. For these reasons the possibility of coating the back plate and Faraday shields, which are subjected to the back streaming ions, is investigated. This topic will be treated in paragraph 3.4.

Also in the ELISE source some back-streaming positive ions are impinging on the back plate and Faraday shields. The positive ions in this case should be mainly formed in a zone where the potential is 30÷60 kV higher than the source potential. Hence, the back-streaming ions should not be critical for the power load on the back plate and Faraday shield (they don't have a high energy). Nevertheless, their contribution on sputtering must be considered, as it could be not negligible.

3.3 Back plate cooling system

Various solutions are currently taken into consideration for the cooling system of the ITER NBI back plate, in order to withstand the loads caused by the back-streaming ions, foreseen by the calculations. The most significant four are here below summarized, and the advantages and disadvantages of every solution are pointed out.

3.3.1 Solution 1

The first considered solution (3.5a) foresees a number of little cooling channels bended around the drivers holes. The channels are embedded inside the back plate and made by milling/electro-deposition technique. A number of inlet/outlet manifolds feed these cooling channels.

- Advantages: Cooling efficiency
- Disadvantages: Complicated cooling scheme, difficulty to reach all the heated zones.

3.3.2 Solution 2

The second considered solution 3.5b foresees vertical cooling channels running along the hot spots columns, feed by “half moon” manifolds that run around the drivers holes. The channels are embedded inside the back plate and made by milling followed by electrodeposition.

- Advantages: Simpler cooling scheme
- Disadvantages: The pressure drop inside the “half moon” manifolds could cause un-uniformity of the water flow (hence of cooling efficiency) in the vertical channels

3.3.3 Solution 3

The third considered solution (Fig. 3.5c) features separated vertical swirl tube elements like in the NB injector calorimeter, next to the drivers plate: in a configuration (of the cooling channels) similar to solutions 2, vertical tubes are placed in correspondence of the hot spots and connected by circumferential manifolds running around drivers seats. This system, separate from the plate, should be fixed mechanically to the plate itself and tubes could be adjacent to proper recesses machined on the plate. Electrodeposition of the plate should be foreseen similar to current configuration, RADI-like: power deposition on the plate should be limited (no back streaming positive ions).

- Advantages: the back plate is covered by a sort of calorimeter that protects it from the back streaming ions. This calorimeter could be easily substituted, for example in case of damage or design change.

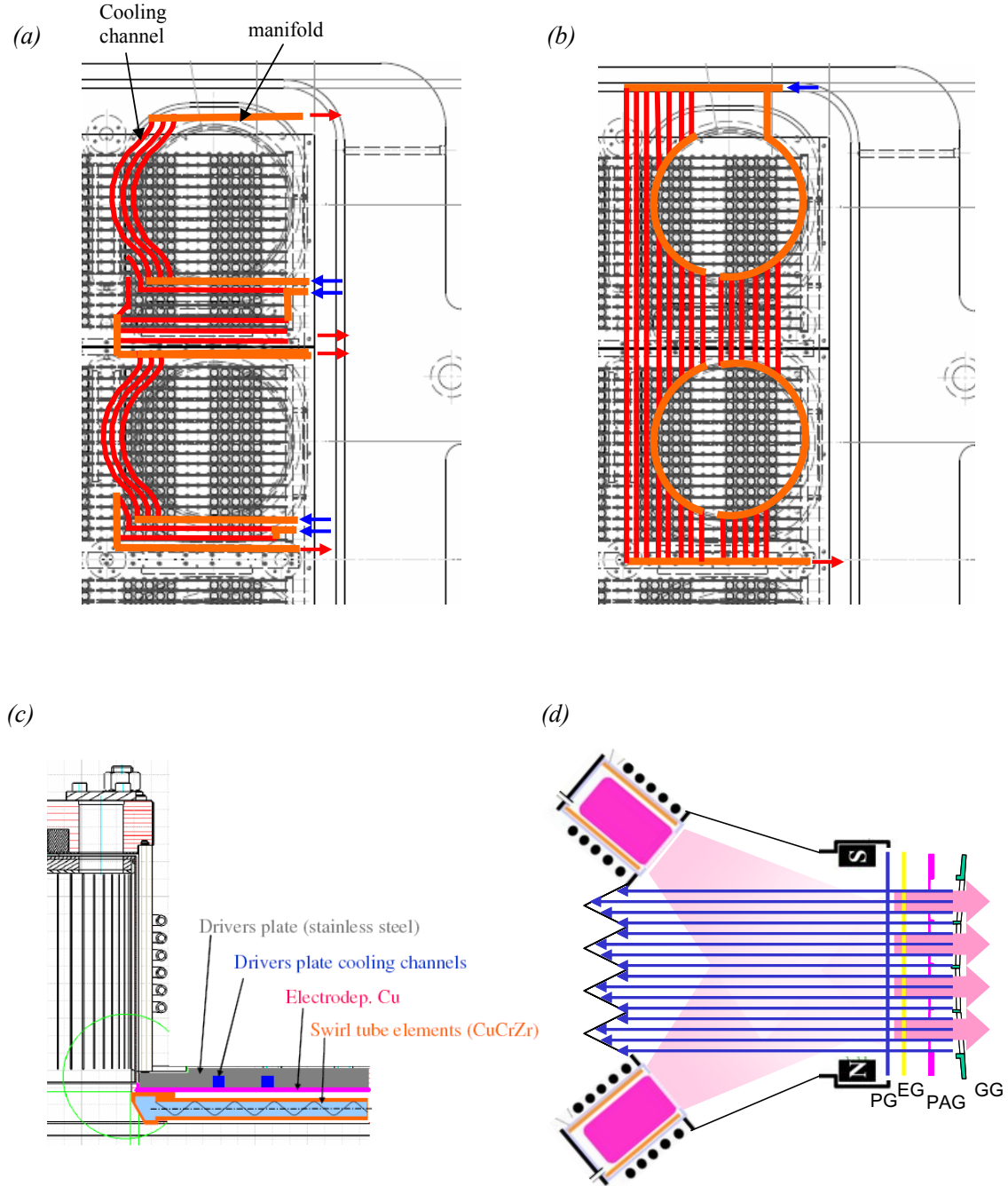


Figure 3.5: Four schemes to upgrade the cooling system of the back plate: (a) Solution 1; (b) Solution 2; (c) Solution 3; (d) Solution 4

- Disadvantages: Can this set of swirl tube elements lay on the drivers plate? Is it somehow interfering with the plasma? How much confidence we have about the actual positions of the hot spots?

3.3.4 Solution 4

The fourth solution (Fig. 3.5d) features a sort of “calorimeter” in the central part of the back plate and drivers shifted at the sides.

- Advantages: The possibility to put oblique panels could permit a large decrease of maximum power density. The Faraday shields are not hit by back-streaming ions. The cooling of the back plate is uniform (no border effects like in the other 3 solutions)
- Disadvantages: The source has a completely different design, so many physical aspects (effect on plasma and beam, uniformity, magnetic fields, cesium distribution etc.) and engineering aspects (layout inside the Beam Source Vessel, compatibility with other components) must be checked.

3.4 Sputtering analysis

The sputtering yield is defined as the average number of atoms (neutral or ionized) ejected from a surface by the impact of an ion. It depends on the energy of primary ions, on their mass, on their incidence angle and on the nature of the sputtered material [7, 8, 9].

The sputtering properties of copper, tungsten and molybdenum surfaces are considered and compared, in order to evaluate the possibility of coating the back plate and Faraday shields (made by copper).

In the following graphs, taken from [10], the sputtering yields at normal ions

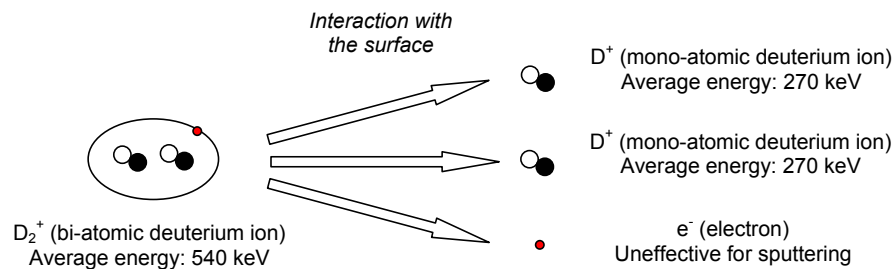


Figure 3.6: Scheme of ions-wall interaction

incidence versus ions energy are presented (Fig. 3.7). They reports experimental and calculated data for the sputtering yields of the three considered materials. Both experimental and calculated data are fitted (separately) with the revised Bohdansky formula:

$$Y(E_0, \alpha = 0^\circ) = Q S_n^{K_r C}(\varepsilon) \left(1 - \left(\frac{E_{th}}{E_0}\right)^{2/3}\right) \left(1 - \frac{E_{th}}{E_0}\right)^2 \quad (3.1)$$

where E_0 is the projectile energy, $S_n^{K_r C}(\varepsilon)$ the nuclear cross section, and ε the reduced energy

$$\begin{aligned} \varepsilon &= \frac{E_0}{E_{TF}} \frac{M_2}{M_1 + M_2} \frac{a_L}{Z_1 Z_2 e^2} \\ &= \frac{E_0}{E_{TF}} \end{aligned} \quad (3.2)$$

where E_{TF} is the Thomas-Fermi energy

$$E_{TF} = \frac{Z_1 Z_2 e^2}{a_L} \frac{M_1 + M_2}{M_2} \quad (3.3)$$

Z_1 and Z_2 are the nuclear charges, and M_1 and M_2 the masses of the projectile and the target atom, respectively. e is the electron charge. The Lindhart screening length a_L is given by

$$\begin{aligned} a_L &= \left(\frac{9\pi^2}{128} a_B (Z_1^{\frac{2}{3}} + Z_2^{\frac{2}{3}}) \right)^{-\frac{1}{2}} \\ &= 0.4685 \left(Z_1^{\frac{2}{3}} + Z_2^{\frac{2}{3}} \right)^{-\frac{1}{2}} \end{aligned} \quad (3.4)$$

where a_B is the Bohr radius. The values of Q and E_{th} are used as parameters to fit the data.

Open data points correspond to experimental values, solid points to calculated values. The dashed curves indicate the fit of experimental data and the solid curves the fit of calculated data. The ion energy of 540 keV, calculated as an average value of the back streaming deuterium ions in ITER NBI, is considered as a reference value for the comparison.

According to the calculations by G. Fubiani, most (86%) of the back-streaming ions are bi-atomic (D^{2+}). In order to estimate the sputtering effect, the conservative hypothesis will be made, to consider all the back-streaming ions as bi-atomic deuterium ions (D^{2+}).

Following a simple scheme of ion-wall interaction (see Fig. 3.6), every bi-atomic D^{2+} ion splits into two deuterium ions D^+ and an electron when it touches the wall surface [11]. Hence, by the sputtering point of view, we can consider the equivalent effect given by two mono-atomic deuterium ions, each with half of the initial energy. So the energy of 270 keV and a double ion flux of D^+ will be considered for the sputtering estimations.

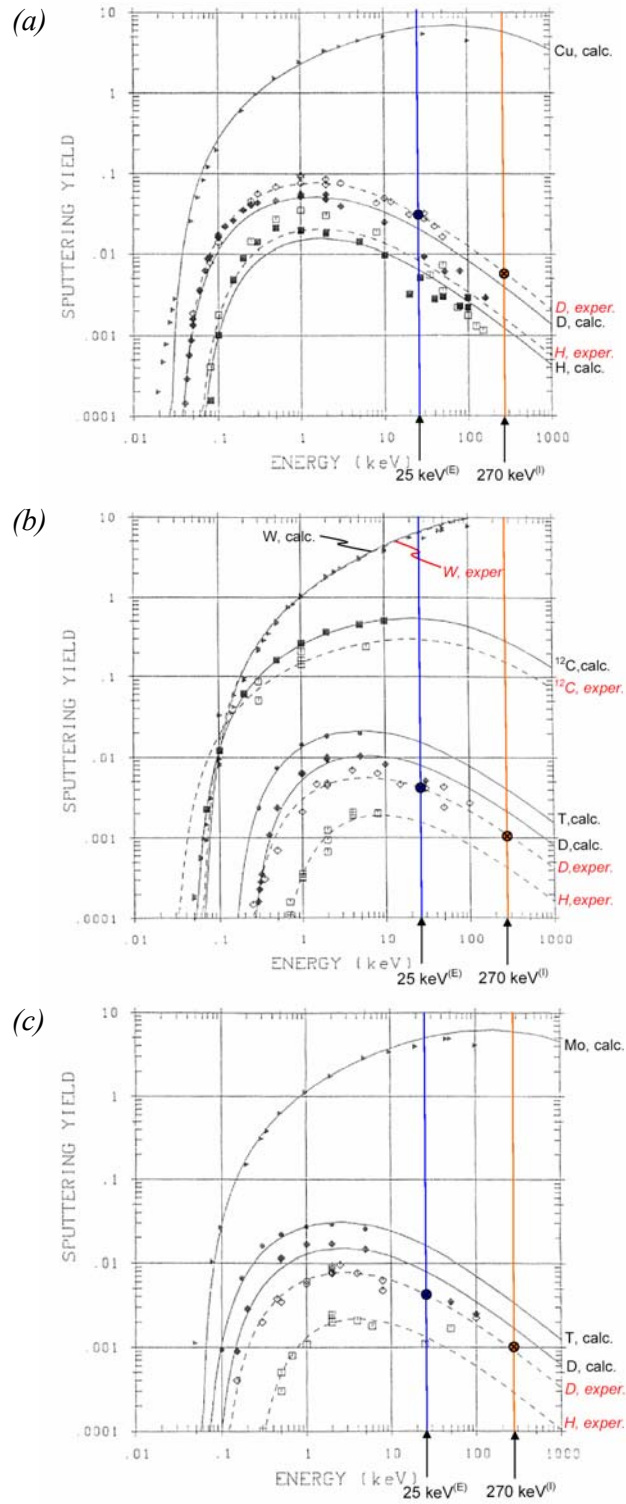


Figure 3.7: Energy dependence of the sputtering yield ($^{(E)}$ Back-streaming ions energy foreseen for ELISE, $^{(I)}$ foreseen for ITER): (a) Copper; (b) Tungsten; (c) Molybdenum.

The ion energy of 50 keV can be estimated as a reasonable average value of the back streaming deuterium ions in ELISE. As a first rough estimation, also for this ion source we can make the hypothesis that the back-streaming ions are exclusively made of D^{2+} . Analogously to what done for ITER NBI, we can estimate an equivalent sputtering effect, by considering an ion energy of 25 keV (half the initial energy) and a double ion flux of D^+ . In both cases (ITER NBI and ELISE), the experimental data fit curve for deuterium ions is considered as a reference for comparing the different surfaces materials.

3.4.1 Copper surface

The maximum sputtering yield for copper corresponds to an ion energy of about 2 keV (see Fig. 3.7a). The foreseen sputtering yield for deuterium ions impinging on a copper surface is 0.006 for 270 keV (ITER NBI reference), 0.03 for 25 keV (ELISE reference).

3.4.2 Tungsten surface

The maximum sputtering yield for tungsten corresponds to an ion energy of about 4 keV (see Fig. 3.7b). The foreseen sputtering yield for deuterium ions impinging on a tungsten surface is 0.001 for 270 keV (ITER NBI reference), 0.0045 for 25 keV (ELISE reference).

3.4.3 Molybdenum surface

The maximum sputtering yield for molybdenum corresponds to an ion energy of about 3 keV (see Fig. 3.7c). The foreseen sputtering yield for deuterium ions impinging on a molybdenum surface is 0.001 for 270 keV (ITER NBI reference), 0.004 for 25 keV (ELISE reference).

3.4.4 Comparison on sputtering yields

If the back plate and drivers surface are coated with tungsten or molybdenum, the sputtering yields given by deuterium back-streaming ions decrease sensibly, as it is visible from the Tab. 3.2. In particular considering 270 keV as D^+ ions energy (value foreseen for ITER) they are both decreasing of 83%, compared to the values obtained with copper.

If lower ions energies are considered the decrease of the sputtering yield becomes even larger. In particular, considering value of 25 keV for the D^+ ions energy (foreseen for ELISE) the estimated decrease relatively to pure copper is 85% with

Ion Energy [keV]	Sputtering Yield Cu	Sputtering Yield W	Variation Cu $\rightarrow$ W	Sputtering Yield Mo	Variation Cu $\rightarrow$ Mo
1	0.07	0.003	-96%	0.006	-91%
10	0.05	0.005	-90%	0.006	-88%
25^(E)	0.03	0.045	-85%	0.004	-87%
100	0.013	0.002	-85%	0.002	-85%
270^(I)	0.006	0.001	-83%	0.001	-83%

Table 3.2: Sputtering yield comparison with deuterium ions and different surface materials: (E) Back-streaming ions energy foreseen for ELISE, (I) foreseen for ITER.

Ion Energy [keV]	Sputtering Yield Cu	Sputtering Yield W	Variation Cu $\rightarrow$ W	Sputtering Yield Mo	Variation Cu $\rightarrow$ Mo
1	0.07	0.003	-96%	0.006	-91%
10	0.05	0.005	-90%	0.006	-88%
25^(E)	0.03	0.045	-85%	0.004	-87%
100	0.013	0.002	-85%	0.002	-85%
270^(I)	0.006	0.001	-83%	0.001	-83%

Table 3.3: Self-sputtering yield comparison with different surface materials.

tungsten coating and 87% with molybdenum coating.

A comparison for the self-sputtering (sputtering given by the ions of surface material) is reported on Tab. 3.3.

If the copper surface is coated with W or Mo, the self-sputtering yield (sputtering given by the ions of surface material) is lower than with Cu at low energies (1÷50 keV), higher or equal to Cu at high energies (>50 keV). Nevertheless, as suggested by the IPP physicists, the energy of the self-sputtered ions inside the chamber is estimated to be below the sputtering threshold (in the range 1÷100 eV), hence this ions should not give any self-sputtering.

3.4.5 Estimation of the sputtered layers thickness

The maximum power density at the hot spot in ITER NBI (with a MAMuG accelerator) is estimated as 6 kW/cm², as it is shown in Fig. 3.4. We will conservatively assume that all the back-streaming ions are bi-atomic deuterium ions (D²⁺). This is not very far from the estimations, that foresee an 86% percentage of bi-atomic ions. The current density, given by the ions at the hot spot, can be estimated as:

$$\begin{aligned}
 j_{D_2^+} &= \frac{6 \cdot 10^3 \frac{W}{cm^2}}{540 \cdot 10^3 \frac{eV}{cm^2}} \\
 &= 0.011 \frac{A}{cm^2}
 \end{aligned} \tag{3.5}$$

The flux of impinging D^2+ ions (in a square centimeter area and in a second) can be estimated as:

$$\begin{aligned}\Phi_{D_2^+} &= \frac{0.011 \frac{A}{cm^2}}{e} \\ &= 6.9 \cdot 10^{16} \frac{ions}{cm^2 \cdot s}\end{aligned}\quad (3.6)$$

where e is the electron charge. The sputtering effect given by this bi-atomic ions can be considered equivalent to the one given by a double flux of the corresponding mono-atomic ions with half the energy. This equivalent flux is equal to:

$$\begin{aligned}\Phi_{D^+} &= 2 \cdot \Phi_{D_2^+} \\ &= 13.8 \cdot 10^{16} \frac{ions}{cm^2 \cdot s}\end{aligned}\quad (3.7)$$

Multiplying the number of the impinging deuterium ions times the sputtering yield, it is possible to estimate how many atoms of the surface material are ejected in a second. This calculation depends on the surface material. In the case of copper, the number of ejected atoms can be estimated as:

$$\begin{aligned}\Phi_{sp,Cu} &= \Phi_{D^+} \cdot SY_{Cu} \\ &= 8.3 \cdot 10^{14} \frac{atoms}{cm^2 \cdot s}\end{aligned}\quad (3.8)$$

where SY_{Cu} is the sputtering yield for copper caused by 270 keV deuterium ions (Fig. 3.7). The volumetric atomic density of copper can be calculated as:

$$\begin{aligned}\rho_{AT,Cu} &= \frac{\rho_{Cu}}{M_{Cu} \cdot N_A} \\ &= 8.4 \cdot 10^{22} \frac{atoms}{cm^3}\end{aligned}\quad (3.9)$$

where ρ_{Cu} is the copper density, M_{Cu} the copper atomic mass and N_A the Avogadro number. Hence, the linear atomic density is:

$$\begin{aligned}\lambda_{AT,Cu} &= \sqrt[3]{\rho_{AT,Cu}} \\ &= 4.4 \cdot 10^7 \frac{atoms}{cm}\end{aligned}\quad (3.10)$$

and a Cu monolayer has the thickness of:

$$\begin{aligned}ML_{Cu} &= \frac{1}{\lambda_{AT,Cu}} \\ &= 2.3 \cdot 10^{-8} \text{ cm} \\ &= 0.23 \text{ nm}\end{aligned}\quad (3.11)$$

The surface atomic density is:

$$\begin{aligned}\sigma_{AT,Cu} &= \rho_{AT,Cu} \cdot ML_{Cu} \\ &= 1.91 \cdot 10^{15} \frac{atoms}{cm^2}\end{aligned}\quad (3.12)$$

The beam-on time to remove a monolayer by sputtering can be estimated as:

$$\begin{aligned}t_{ML,Cu} &= \frac{\sigma_{AT,Cu}}{\Phi_{sp,Cu}} \\ &= 2.3 \text{ s}\end{aligned}\quad (3.13)$$

	ρ Density [g/cm ³]	M Atomic mass [g/mol]	ρ_{AT} Atomic volum. density [atoms/cm ³]	σ_{AT} Atomic surface density [atoms/cm ²]	ML Monolayer thickness [nm]	SY Sputtering yield for D ⁺ ions /	Φ_{sp} Sputtered atoms flux [atoms/(cm ² ·s)]	$t_{\mu m}$ Beam-on time to remove 1 μm [hours]
Cu	8.9	63.54	$8.4 \cdot 10^{22}$	$1.9 \cdot 10^{15}$	0.23	0.006	$8.3 \cdot 10^{14}$	2.8
W	19.3	183.85	$6.2 \cdot 10^{22}$	$1.58 \cdot 10^{15}$	0.25	0.001	$1.4 \cdot 10^{13}$	12.6
Mo	10.28	95.94	$6.4 \cdot 10^{22}$	$1.60 \cdot 10^{15}$	0.25	0.001	$1.4 \cdot 10^{13}$	12.9

Table 3.4: Estimation of the sputtered layers thickness for ITER NBI.

and the beam-on time to remove a μm of surface material on the hot spot by sputtering is approximately:

$$\begin{aligned}
 t_{\mu m, Cu} &= t_{ML, Cu} \cdot \frac{1000}{ML_{Cu}} \\
 &= 10084 \text{ s} \\
 &= 2.8 \text{ h}
 \end{aligned} \tag{3.14}$$

With the same method, we can calculate the time required to remove a μm of tungsten or molybdenum. The data are summarized in the table 3.4.

For ELISE, an rough estimation can be done assuming the same back-streaming ion current at the hot spots, as calculated for ITER NBI.

$$j_{D^+} = 0.0011 \frac{A}{cm^2} \tag{3.15}$$

and correspondingly flux of impinging ions (in a square centimeter area and in a second):

$$\begin{aligned}
 \Phi_{D_2^+} &= \frac{0.0011 \frac{A}{cm^2}}{e} \\
 &= 6.9 \cdot 10^{16} \frac{ions}{cm^2 \cdot s}
 \end{aligned} \tag{3.16}$$

$$\begin{aligned}
 \Phi_{D^+} &= 2 \cdot \Phi_{D_2^+} \\
 &= 13.8 \cdot 10^{16} \frac{ions}{cm^2 \cdot s}
 \end{aligned} \tag{3.17}$$

The sputtering yields, depending on the ions energy, are in this case higher, and consequently the estimated times to remove a 1 μm layer of surface material from the hot spot are shorter, as it is shown in the Tab. 3.5.

3.4.6 Comments

A molybdenum coating seems to give the best sputtering behavior, because it has the minimum sputtering yield from the back streaming deuterium ions (-83% in ITER

	ρ Density [g/cm ³]	M Atomic mass [g/mol]	ρ_{AT} Atomic volum. density [atoms/cm ³]	σ_{AT} Atomic surface density [atoms/cm ²]	ML Monolayer thickness [nm]	SY Sputtering yield for D ⁺ ions /	Φ_{sp} Sputtered atoms flux [atoms/(cm ² ·s)]	$t_{\mu m}$ Beam-on time to remove 1 μm [hours]
Cu	8.9	63.54	$8.4 \cdot 10^{22}$	$1.9 \cdot 10^{15}$	0.23	0.02	$4.2 \cdot 10^{15}$	0.56
W	19.3	183.85	$6.2 \cdot 10^{22}$	$1.58 \cdot 10^{15}$	0.25	0.003	$5.5 \cdot 10^{14}$	3.1
Mo	10.28	95.94	$6.4 \cdot 10^{22}$	$1.60 \cdot 10^{15}$	0.25	0.003	$5.5 \cdot 10^{14}$	3.2

Table 3.5: Estimation of the sputtered layers thickness for ELISE.

NBI, -87% in ELISE compared to copper) and at the same time a self-sputtering yield lower than copper. Moreover, it is estimated to have the lowest beam-on time to remove 1 μm of surface material on the hot spot by sputtering (both in ITER NBI and ELISE cases).

3.4.6.1 Estimation of layer removed by sputtering during the whole ITER life

Considering that the ITER NBI is foreseen to have a total beam-on time of about 5500 hours, not negligible layers of material are foreseen to be removed by sputtering from the back plate and Faraday shield surfaces. With the input data here considered of a maximum power density of 60 MW/m², the thicknesses of material removed from the hot spots can be estimated as:

$$\begin{aligned}
 \delta_{Cu} &= \frac{5500 \frac{h}{\mu m}}{2.8 \frac{h}{\mu m}} \\
 &= 1964 \mu m \\
 &= 1.964 mm
 \end{aligned} \tag{3.18}$$

with a copper surface,

$$\begin{aligned}
 \delta_W &= \frac{5500 \frac{h}{\mu m}}{12.6 \frac{h}{\mu m}} \\
 &= 436 \mu m \\
 &= 0.436 mm
 \end{aligned} \tag{3.19}$$

with a tungsten surface, and

$$\begin{aligned}
 \delta_{Mo} &= \frac{5500 \frac{h}{\mu m}}{12.9 \frac{h}{\mu m}} \\
 &= 426 \mu m \\
 &= 0.426 mm
 \end{aligned} \tag{3.20}$$

with a molybdenum surface.

These values for the sputtered layer are extremely high and suggest a careful verification of the assumptions considered for the physics calculations, that are estimating the enormous power density of 60 MW/m^2 impinging on the hot spot. By the experimental point of view, there is no evidence of such a large removal of material by sputtering at IPP, where the negative ion current is similar to one foreseen for ITER. In these sources, only a slight increase of cesium concentration inside the source can be observed. This increase could be caused by the fact that the back plate and Faraday shields, hit by the back-streaming ions during the beam-on time, are becoming hotter, and the cesium flux from these surface is consequently higher.

A survey is currently on-going to understand with more detail how is the situation with the back plate in LHD device (Fig. 3.3). In particular, if there is sputtering, how much are the heat loads, if there is cesium compound formation on and around the hot spots etc.

It would be important to do a similar survey also for the Kamaboko source at Naka. In fact, this is the source with operating parameters (voltage, working pressure, current density) similar to the one required for the ITER NBI, and it is known that the back plate (that is made of stainless steel in this case) has encountered some problems of surface integrity (it is not clear if they were due to melting or sputtering events), with consequent water leaks.

3.4.7 Discussion

In the following, the main topics on the design of a negative ion source are discussed with the physicists and engineers of the *Technologie* group at IPP Garching [11]. The answers are based on many years of operations on the three ion sources BATMAN, RADI and MANITU.

1. By the plasma physics and operating point of views, are all the four solutions for cooling acceptable?

The first two solutions do not present particular issues by the physics and operating point of views. Some problems might arise from the fact that the back plate is working at a high temperature (higher than 50° C , which is the suggested temperature in the IPP sources) and with less uniformity. The third solution, foreseeing the swirl tubes over the back plate surface, present some aspects that could cause problems:

- The space between the swirl tubes and the back plate could be a trap for cesium and for impurities. The physicists of IPP [11] underlines that we do not know

exactly what is happening to the surfaces in that zone. The conditioning of the ion source, which is a long-lasting operation also with flat surfaces (about one week in BATMAN), could be further lengthened in the case of a more complex surface of the back plate. In particular, the cleaning of the surfaces covered by the swirl tube would be probably more difficult.

- During the different operating phases (conditioning, half power, full power etc.), the back-streaming ions could impinge not exactly on the foreseen location (which is covered by the swirl tube) but at a certain distance from it. If the shifted position is not covered by the swirl tube, the back-streaming ions would hit the back plate, which is less cooled. This shifting of the hot spot could be caused for example by a misalignment of the grids, for example during operating conditions very different from the nominal one (for example, during conditioning).
- In the sources at IPP, the back plate temperature is controlled to have a good interaction with cesium. A value of 50° , with good uniformity, seems to give the best results. Hence, the back plate must be warmed and kept at the right temperature. The temperature control of the back plate and swirl tubes surfaces seems more complicated than controlling the only the back plate, like it is in the solutions 1 and 2.

The fourth solution would need some years of operations to understand if it is an applicable scheme [11]. In principle, it could work, but it is not possible to estimate the working parameters (current density, uniformity etc.), as the physics inside the main chamber could be completely different.

2. How important is it for the ion source walls (in particular for the back plate) to have a uniform and controlled temperature distribution?

The ideal case would be to have the ion source walls temperature as uniform as possible, and with a value around 50° , which is the suggested value for the sources at IPP. This is not possible due to the highly concentrated hot spot given by the back streaming ions. Nevertheless, the IPP experience suggests that the influence of the back plate temperature, if maintained inside a reasonable range (for example, average temperature lower than 100° C), should not have a big influence on the ion source performances.

3. Do you suggest a coating of the back plate and Faraday shields?

The experience with the molybdenum coated Faraday shields in Batman was positive, as the sputtering of copper was reduced. Having less heavy elements inside

the chamber means having a better effect of the cesium in creating negative ions on the PG surface. In particular, the cesium deposited on the PG is not “buried” by other elements (copper, tungsten etc.) coming from the chamber walls. Another good reason to limit the sputtering inside the ion source is represented by the high values for the thickness of removed material, foreseen during the life of ITER NBI. For this reasons, a coating of W or Mo, materials that present a sputtering yield 5-6 times lower than copper, is suggested for both the Faraday shields and the back plate of the ion source.

4. The PVD (Physical Vapour Deposition technique) is a possible option for an eventual coating of the back plate and Faraday shield. With this technique, only a thin layer (some μm) of coating can be applied in a reasonable time. Is this layer sufficient if the ion energy is as high as 540 keV (average energy foreseen for the deuterium ions)?

From the experience, it seems that with the PVD technique is not possible to deposit more than some μm (maximum 10) onto a surface [11].

5. Is it technologically feasible to coat with some 500 μm W or Mo layer, in order to withstand the sputtering for the whole life of ITER NBI? Discussion on the possibility to start from a 1÷2 mm thick Mo plate and then fix it - by Hot Isostatic Pressing or brazing - to a copper plate.

With plasma spray coating technique, a thicker layer of coating can be deposited, but the porosity of the layer seems to be an issue of these treatments. A survey on this topic is on-going at IPP. About starting with a molybdenum plate and then fix it to a copper one by means of HIP or brazing, this is a complicated solution, to be considered only if it becomes clear that a really large coating thickness is needed and it is not possible to make it by a coating technique.

6. Could the cesium layer in a certain measure “protect ” from sputtering the surface of the back plate and Faraday shields?

No, the cesium layer is not present on the hot spots, because it is completely evaporated due to the high temperatures [11].

Bibliography

- [1] D. Marcuzzi, P. Agostinetti, M. Dalla Palma, H.D. Falter, B. Heinemann and R. Riedl. *Design of the RF source for the Iter NBI*. Fusion Eng. Des. 82 (2007) 798-805.

- [2] E. Speth. *Neutral Beam Heating of Fusion Plasmas*. Rep. Prog. Phys. 52 (1989) 57-121.
- [3] P. Agostinetti, S. Dal Bello, M. Dalla Palma, D. Marcuzzi, P. Zaccaria, R. Nocentini, M. Fröschle, B. Heinemann and R. Riedl. *Thermo-mechanical design of the ITER Neutral Beam Injector grids for Radio Frequency Ion Source and SINGAP Accelerator*. 22nd IEEE/NPSS Symposium on Fusion Engineering, Albuquerque, New Mexico, 2007.
- [4] R.S. Hemsworth. *Positive ions in the SINGAP ITER Accelerator*. CEA internal report, 2006.
- [5] G. Fubiani. *Secondary emission processes in the ITER negative ion based electrostatic accelerator*. CCNB 2006, Naka.
- [6] *Private communication*.
- [7] W. Eckstein and R. Preuss. *New fit formulae for the sputtering yield*. Journal of Nuclear Materials 320 (2003), 209-213.
- [8] J. Bohdanský. *A universal relation for the sputtering yield of monoatomic solids at normal ion incidence*. Nuclear Instruments and Methods in Physics Research B2 (1984), 587-591.
- [9] H.L. Bay and J. Bohdanský. *Sputtering yields for light ions as a function of angle incidence*. Applied Physics 19 (1979), 421-426.
- [10] W. Eckstein, C. Garcia-Rosales, J. Roth and W. Ottenberg. *Sputtering Data*. IPP Report 9/82, February 1993.
- [11] *Discussion at the Max-Planck-Institut für Plasmaphysik, Garching, Germany, 2006*.

Chapter 4

Decision making methods for thermo-mechanical design

SUMMARY - The decision making techniques help to make the best decisions possible with the information available. These tools aim at mapping out the likely consequences of decisions and help choosing the best course of action to take. They can be used to choose a suitable solution or to optimize an existing one in many applications. This chapter deals with these techniques, with a particular focus on the most suitable methods for thermo-mechanical design choice processes.

4.1 Introduction

Decision making is the cognitive process leading to the selection of a course of action among possible options. Every decision making process produces a final choice. It can be an action or an opinion. It begins when we need to do something but don't know what, or when we have to select a solution among different choices and typically this selection process involves several people. Therefore, decision making is a reasoning process which can be rational or irrational, can be based on explicit assumptions or tacit assumptions.

Decision making techniques aim at supporting decision makers who are faced with making numerous and conflicting evaluations, by highlighting these conflicts and deriving a way to come to a compromise, or better to a win-win decision in a transparent process.

For example, the European Parliament may apply these techniques to help assess whether the introduction of software patents in Europe would help or destroy the European software industry. Since a decision process always involves a certain

element of subjectiveness, the morals and ethics play a significant part in the accuracy and fairness of conclusions. The most used and best known decision making methods are:

- the decision-matrix method, or Pugh method;
- the Analytic Hierarchy Process (AHP);
- the Multi-Attribute Global Inference of Quality (MAGIQ);
- the Goal Programming;
- the Data Envelopment Analysis;
- the Evidential Reasoning Approach;
- the Dominance-based Rough Set Approach (DRSA);

These are widely used in fields like: operational research, economics, politics, production theory and finance. Some of them are quite complicated and can be used only having a big amount of historical data on the subject and high computational power. The choice of which method is most appropriate depends on the problem at hand and may be to some extent dependent on which model the decision maker is most comfortable with.

In the present case, the aim is to apply these methods to thermo-mechanical design choice processes, and there is not much statistical data to handle. Hence, the simpler approaches given by the Pugh and AHP method were found to be the most suitable. These two methods are described in detail in the following.

4.2 Description of the Pugh Matrix method

4.2.1 Generalities

The decision-matrix method, also known as Pugh method, is a quantitative technique used to rank the multi-dimensional options of an option set. It is frequently used in engineering for making design decisions but can also be used to rank investments options, vendor options, product options or any other set of multidimensional entities.[1, 2]

A basic decision matrix consists of establishing a set of weighted criteria upon which the potential options can be decomposed, scored, and summed to gain a total score which can then be ranked.

The advantage of this approach to decision making is that subjective opinions

about one alternative versus another can be made more objective. Another advantage of this method is that sensitivity studies can be performed. An example of this might be to see how much your opinion would have to change in order for a lower ranked alternative to out-rank a competing alternative.

A Pugh matrix evaluates and prioritizes a list of options. The team first establishes a list of weighted criteria and then evaluates each option against those criteria. It is a chart that allows a team or individual to systematically identify, analyze, and rate the strength of relationships between sets of information. The matrix is especially useful for looking at large numbers of decision factors and assessing each factor's relative importance.

It is frequently used during quality planning activities to select product/service features and goals and to develop process steps and weigh alternatives. For quality improvement activities, a decision matrix can be useful in selecting a project, in evaluating alternative solutions to problems, and in designing remedies.

4.2.2 How to build up and use a Pugh Matrix

The following main steps should be done in order to build up a decision matrix.

1. **Identify alternatives.** Depending upon the team's needs, these can be product/service features, process steps, projects, or potential solutions. List these across the top of the matrix.
2. **Identify decision/selection criteria.** These key criteria may come from a previously prepared affinity diagram or from a brainstorming activity. Make sure that everyone has a clear and common understanding of what the criteria mean. Also ensure that the criteria are written so that a high score for each criterion represents a favorable result and a low score represents an unfavorable result. List the criteria down the left side of the matrix.
3. **Assign weights.** If some decision criteria are more important than others, review and agree on appropriate weights to assign (e.g., 1, 2, 3).
4. **Design scoring system.** Before rating the alternatives, the team must agree on a scoring system. Determine the scoring range (e.g., 1 to 5 or 1, 3, 5) and ensure that all team members have a common understanding of what high, medium, and low scores represent.
5. **Rate the alternatives.** For each alternative, assign a consensus rating for each decision criterion. The team may average the scores from individual team members or may develop scores through a consensus-building activity.

6. **Total the scores.** Multiply the score for each decision criterion by its weighting factor. Then total the scores for each alternative being considered and analyze the results.

The success of the method depends in particular on the first two steps, for which it is useful to maintain an environment open to the ideas and the constructive discussion, like in a brainstorming session.

Assigning different weights to the criteria is not a mandatory step. The weights can be assumed equal for all the criteria, and only a +/- (better/worse) evaluation can be done for every option. This gives a much faster procedure, but somehow less precise. Nevertheless, some author think that if the Pugh method is applied without weighting the criteria (like it is in the example of paragraph 4.2.5) there is somehow more room for brainstorming and the birth of new ideas, that can take the better aspects of the many competing solutions.

At the end of whole procedure, a concept should be identified, which behaves better than the others according to the given selection criteria. Nevertheless, the best selected concept could not be the best possible one, and a further optimization is advisable, in order to maximize the positive aspects of all the concepts and minimize the negative ones.

Hence, the Pugh matrix process can be iterated in repeated round to help develop “super concepts”. To make this, after the first round of ranking, new concepts can be created by:

- Synthesizing the best features of the different alternatives into new concepts.
- Enhancing the strongest concepts by adding features from the unselected concepts to address weak areas.

4.2.3 When to use a Decision Matrix

The Decision Matrix is particularly suitable when a list of options must be narrowed to one choice, when the decision must be made on the basis of several criteria. The typical situations to apply this method are:

- When one improvement opportunity or problem must be selected to work on.
- When only one solution or problem-solving approach can be implemented.
- When only one new product can be developed.

4.2.4 Review the concepts

After every application of the Pugh matrix, a design review should be done in order to choose the best solution. This process objectively evaluates the quality of a design at various stages of the design process. It provides an opportunity for voices external to the design team to provide feedback on the design as the design team develops the process, product, or service.

A well-conducted design review ensures that the design will function effectively to produce a high-quality process, product, or service. Design reviews are quality control tools applied to the design process. They ensure the effectiveness of the design and the efficiency of the design process, i.e. the teams responsible for the various elements of the design are working in a coordinated fashion that minimizes rework and duplicate efforts.

When conducting a design review, it is important to:

- Ensure that the design review gets both internal and external input
- Focus on identifying and resolving problems during the reviews, and use the feedback you receive to make changes immediately to the design

A design review should be organized whenever external feedback appears appropriate or when there are coordination issues. In particular, it is important to conduct:

- A concept review after identifying one or two key concepts and determining their feasibility.
- A high-level design review after designing and testing a selected concept to some level of detail but before beginning the detailed design
- A pre-pilot design review after completing the detailed design, but before the process, product or service is ready to be piloted.

A concept review should obtain feedback from all the interested parties (customer, suppliers, other organizational entities etc.) about the concepts that were selected.

The feedback obtained during the concept review is a critical check on the design team's thinking. It can provide insights that lead to modifications of the selected concept(s) early in the design process when changes are easier to make and risk and costs can be minimized.

4.2.5 Example

In the following example, the Pugh method is applied for the optimization of the design of a car hornet [3]. Four design concepts are selected during a first brainstorming session, shown in Fig. 4.1. Design A is the datum (the best existing

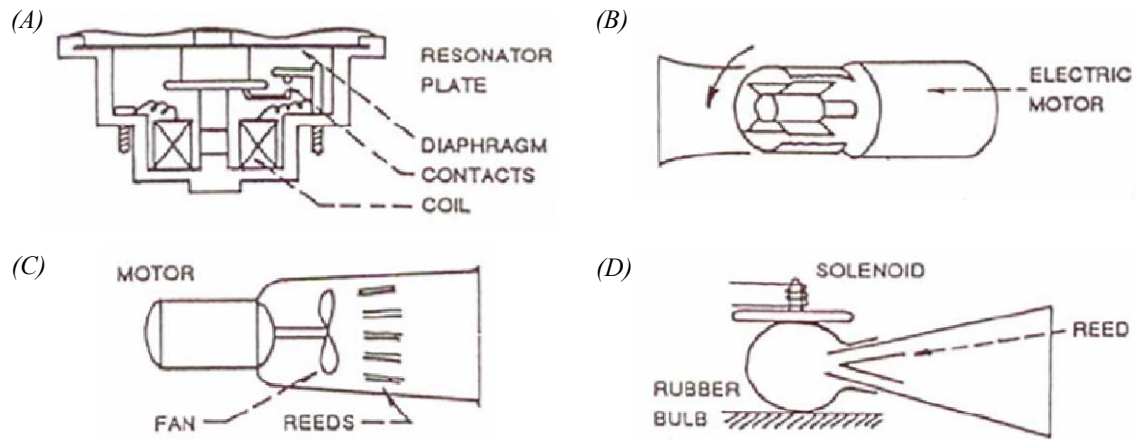


Figure 4.1: Four concepts for the design of a car horn.

product); it is entered in the first column of the Pugh matrix (see Fig. 4.1) next to the list of criteria. Each new design concept was then evaluated against the datum for each criterion.

The total scores for each design are obtained. The positives and negatives are added separately since positives cannot cancel out negatives. The results from show that some concepts were able to improve on the existing design; however, all concepts accumulated a large number of negative marks. Therefore, the next activity concentrates on making the concepts better by trying to eliminate as many of the negatives as possible.

If low manufacturing cost is very important and cannot be reduced for this design, then other concepts that do not have this barrier need to be optimized further. Although a team may decide to quickly throw out a few of the low-scoring concepts, this should be done with caution. Some of the better features or improved components of these concepts may be merged with other concepts for a better design. They should be examined for stepping stone ideas; thus they provide a valuable service. During this review and discussion of each design in an effort to make improvements, amended or new concepts are added to the evaluation matrix as new designs.

4.3 Description of the Analytic Hierarchy Process

4.3.1 Generalities

The Analytic Hierarchy Process (AHP) is a technique for decision making where there are a limited number of choices, but where each has a number of different attributes, some or all of which may be difficult to formalize [4, 5]. It is especially

		Concepts			
		Scheme A (datum)	Scheme B	Scheme C	Scheme D
Criteria	Ease of achieving 100 – 125 decibel (sound level)		–	–	+
	Ease of achieving 2000 – 5000 Hertz (sound frequency)		S	S	S
	Resistance to corrosion (water, pollutants)		–	–	–
	Resistance to vibration, shock, acceleration/deceleration, wear-and-tear		–	–	S
	Resistance to temperature cycling and extremes		–	–	–
	Low power consumption		–	–	–
	Ease of maintenance		+	+	+
	Small size		–	–	–
	Long service life		–	–	S
	Low manufacturing cost		S	+	+
	Ease of installation		S	S	+
	<i>Total positives</i>		<i>1+</i>	<i>2+</i>	<i>4+</i>
	<i>Total negatives</i>		<i>7–</i>	<i>7–</i>	<i>4–</i>

Table 4.1: Pugh matrix for the design of a car horn.

applicable when decisions are being made by a team.

AHP can assist with identifying and weighting selection criteria, analyzing the data collected for the criteria, and expediting the decision-making process. It helps capture both subjective and objective evaluation measures, providing a useful mechanism for checking the consistency of the evaluation measures and alternatives suggested by the team [6, 7].

The process is based on a series pairwise comparisons which are checked for internal consistency and then combined. The procedure can be summarized as:

1. The alternatives and the significant attributes are identified.
2. For each attribute, and each pair of alternatives, the decision makers specify their preference (for example, whether the location of alternative “A” is preferred to that of “B”) in the form of a fraction between 1/9 and 9.
3. Decision makers similarly indicate the relative significance of the attributes. For example, if the alternatives are comparing potential real-estate purchases, the investors might say they prefer location over price and price over timing.
4. Each matrix of preferences is evaluated by using eigenvalues to check the consistency of the responses. This produces a “consistency coefficient” where a value of “1” means all preferences are internally consistent. This value would

be lower, however, if a decision maker said X is preferred to Y, Y to Z but Z is preferred to X (such a position is internally inconsistent). It is this step that that causes many users to believe that AHP is theoretically well founded.

5. A score is calculated for each alternative.

4.3.2 Criticisms

Despite its widespread use as a decision method, the AHP has been the subject of criticisms, notably about the possibility of different hierarchies being applied to identical problems; about possible major changes in results if the hierarchy is changed in minor ways; and about the absence of statistical theory to underlie the process. While the concerns are mostly theoretical or speculative, the process works well in practice and is extremely popular among decision-makers in the private and public sectors.

AHP, like many systems based on pairwise comparisons, can produce “rank reversal” outcomes. That is a situation where the order of preference is, for example, A, B, C then D. But if C is eliminated for other reasons, the order of A and B could be reversed so that the resulting priority is then B, A, then D. It has been proven that any pairwise comparison system will still have rank-reversal solutions even when the pair preferences are consistent.

Many alternatives to AHP are economically viable, especially for larger, riskier decision. Methods from decision theory and various economic modeling methods can be applied [8, 9].

4.3.3 Example

In the following, the AHP method is applied to make a decision between four job offers: Acme Manufacturing, Bankers Bank, Creative Consulting, and Dynamic Decision Making. Factors such as location, salary, amount of management science, and long term prospects are important for this choice, but the relative importance is not easy to evaluate.

The first step in AHP is to ignore the jobs and just decide the relative importance of the objectives. This can be done by comparing each pair of objectives and ranking them on the following scale: Comparing objective i and objective j (where i is assumed to be at least as important as j), give a value a_{ij} as follows:

1	Objectives i and j are of equal importance
3	Objective i is weakly more important than j
5	Objective i is strongly more important than j
7	Objective i is very strongly more important than j
9	Objective i is absolutely more important than j
2,4,6,8	Intermediate values

Table 4.2: Pairwise comparison values.

Of course, we set $a_{ii}=1$. Furthermore, if we set $a_{ij}=k$, then we set $a_{ji}=1/k$. The following table resumes the comparison results:

	Location	Salary	MS	Long
Location	1	1/5	1/3	1/2
Salary	5	1	2	4
MS	3	1/2	1	3
Long	2	1/4	1/3	1

Table 4.3: Preferences on Objectives.

Now, the AHP is going to make some simple calculations to determine the overall weight that we are assigning to each objective: this weight will be between 0 and 1, and the total weights will add up to 1. We do that by taking each entry and dividing by the sum of the column it appears in. For instance the (Location,Location) entry would end up as $\frac{1}{1+5+3+2} = 0.091$, (Salary,Location) as $\frac{5}{1+5+3+2} = 0.445$ and (Location, Salary) as $\frac{\frac{1}{5}}{\frac{1}{5}+1+\frac{1}{2}+\frac{1}{4}} = 0.102$ The whole matrix become:

	Location	Salary	MS	Long	Average
Location	0.091	0.102	0.091	0.059	0.086
Salary	0.455	0.513	0.545	0.471	0.496
MS	0.273	0.256	0.273	0.353	0.289
Long	0.182	0.128	0.091	0.118	0.130

Table 4.4: Weights on Objectives.

This suggests that about half of my objective weight is on salary, 30% on amount of management science, 13% on long term prospects, and 9% on location.

If we read down the first column in the original matrix, we have the values of each of the objectives, normalized by setting the value of location to be 1. Similarly, the second column are the values, normalizing with salary equals 1. For a perfectly consistent decision maker, each column should be identical, except for the normalization. By dividing by the total in each column, therefore, we would expect

identical columns, with each entry giving the relative weight of the row's objective. By averaging across each row, we correct for any small inconsistencies in the decision making process.

Our next step is to evaluate all the jobs on each objective. For instance, if we take Location, if we prefer to be in the northeast (and preferably Boston), and the jobs are located in Pittsburgh, New York, Boston, and San Francisco respectively, then we might get the following matrix:

	A	B	C	D
A	1	1/2	1/3	5
B	2	1	1/2	7
C	3	2	1	9
D	1/5	1/7	1/9	1

Table 4.5: Location Scores.

Again we can normalize (divide by the sums of the columns, and average across rows to get the relative weights of each job with regards to location. In this case, we get the following:

	A	B	C	D	Average
A	0.161	0.137	0.171	0.227	0.174
B	0.322	0.275	0.257	0.312	0.293
C	0.483	0.549	0.514	0.409	0.489
D	0.032	0.040	0.057	0.045	0.044

Table 4.6: Relative Location Scores.

In words, of the the total "Location Value" available, Job C has about 50%, B has about 30%, A has about 17% and D has about 4%. We can go through a similar process with Salary, amount of MS, and long term prospects. Suppose the relative values for the objectives can be given as follows:

	A	B	C	D
Location	0.174	0.293	0.489	0.044
Salary	0.050	0.444	0.312	0.194
MS	0.210	0.038	0.354	0.398
Long	0.510	0.012	0.290	0.188

Table 4.7: Relative scores for each objective.

Recalling our overall weights, we can now get a value for each job. The value for Acme Manufacturing is

$$(0.174)(0.086) + (0.050)(0.496) + (0.210)(0.289) + (0.510)(0.130) = 0.164$$

Similarly, the value Bankers Bank is

$$(0.293)(0.086) + (0.444)(0.496) + (0.038)(0.289) + (0.012)(0.130) = 0.256$$

The value for Creative Consultants is 0.335, and that for Dynamic Decision is 0.238. Hence, Creative Consultants is the best option.

The Analytic Hierarchy Process is a method for formalizing decision making where there are a limited number of choices but each has a number of attributes and it is difficult to formalize some of those attributes. Note in this example, we did not collect any data (like miles from a preferred point or salary numbers). Instead, we use phrases like “much more important than” to extract the decision makers preferences.

The AHP has been used in a large number of applications to provide some structure on a decision making process. Note that the system is somewhat ad-hoc (why 1-9 range?) and there are a number of “hidden assumptions” (if i is weakly preferred to j and j weakly preferred to k, then a consistent decision maker must have i absolutely preferred to k, which is not what my idea of the words means). Furthermore, an unscrupulous can easily manipulate the rankings to get a preferred outcome (by using a non-management science technique called “lying”) Despite the rather arbitrary aspects of the procedure, however, it can provide useful insight into the tradeoffs embedded in a decision making problem.

Bibliography

- [1] D. Ginn and E. Varner. *The Design for Six Sigma Memory Jogger: Tools and Methods for Robust Processes and Products*. GOAL/QPC 2004.
- [2] R. Ramaswamy. *Design and Management of Service Processes*. Addison-Wesley Pub., 1996.
- [3] Various Authors. *Creative Problem Solving: Thinking Skills for a Changing World, College Custom Series*. McGraw-Hill, 1993.
- [4] J. McCaffrey. *Test Run: The Analytic Hierarchy Process*. MSDN Magazine, June 2005.
- [5] J. R. Grandzol. *Improving the Faculty Selection Process in Higher Education: A Case for the Analytic Hierarchy Process*. IR Applications 6, 2005.
- [6] N. Bushan and K. Rai. *Strategic Decision Making: Applying the Analytic Hierarchy Process (Decision Engineering)*. Springer-Verlag, 2004.

- [7] M. Berrittella. *An Analytic Hierarchy Process for the Evaluation of Transport Policies to Reduce Climate Change Impacts*. Fondazione Eni Enrico Mattei (Milano), 2007.
- [8] W. Atthirawong. *An Application of the Analytical Hierarchy Process to International Location Decision-Making*. Gregory, Mike, Proceedings of The 7th Annual Cambridge International Manufacturing Symposium: Restructuring Global Manufacturing, Cambridge, 2002.
- [9] P. K. Dey. *Analytic Hierarchy Process Analyzes Risk of Operating Cross-Country Petroleum Pipelines in India*. Natural Hazards Review, 2003.

Chapter 5

Decision making methods applied to the accelerator design

SUMMARY - The accelerator of a negative based neutral beam injector has to fulfil at the same time physical and mechanical requirements. The first group comprises requirements of good optics, high ion current density and low electron current density. The second group comprises requirements of acceptable working temperatures, stresses and deformation. In this chapter, after a brief description of the design of the accelerator for the ITER NBI in SINGAP version, a design optimization process following the most important decision making techniques will be described.

5.1 Overview of the accelerator design

The grids of RF source and SINGAP accelerator are: the Plasma Grid (PG), the Extraction Grid (EG), the Pre-Acceleration Grid (PAG) and the Grounded Grid (GG) [1, 2, 3].

Several analyses have been carried out to assess the reliability of the design of these grids, and to optimize the design parameters in order to fulfil the ITER requirements [4, 5].

The overall dimensions of the grids are about 800x1600 mm². The first three grids (PG, EG and PAG) feature 1280 apertures, where the ion beamlets are extracted from the ion source and pre-accelerated up to 40 kV (Fig. 5.1). The last grid (GG) features 16 large apertures, where the beamlets are merged and accelerated up to potential of 1 MV. The total ion current is 40 A, the total power of the ion beam is 40 MW.

All the grids are made by electrodeposition of pure copper onto a copper base

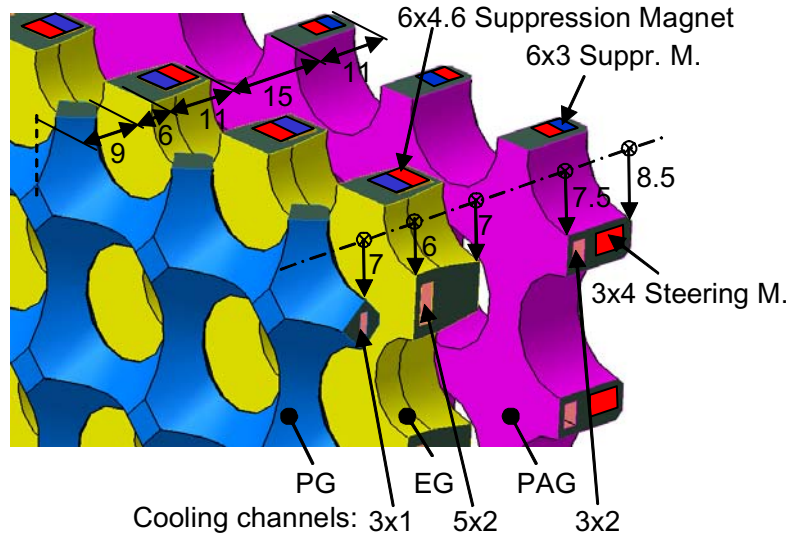


Figure 5.1: PG, EG and PAG detail (dimension in mm).

plate. This technique permits to obtain a very complex geometric shape (with very small cooling channels that run inside the grid and embedded magnets) and to have good mechanical properties, due to the high purity and to the very small grain size. Four independent water cooling systems, shown in Fig. 5.2 are foreseen in order to control the temperature (and consequently stress and deformations) of the four grids.

The plasma grid is heated by the plasma inside the RF ion source, with a surface power density that is estimated to range between 3 and 20 kW/m² (IPP experimental results). This grid is required to operate at a temperature of about 150° C in order to enhance the caesium effect for negative ions surface generation. Hence its cooling system is designed to obtain a precise control of the temperature, with a good uniformity on the whole surface. For the same reason, this grid is Molybdenum coated on the plasma side. The apertures are designed with conical chamfers on the upstream and downstream sides of the grid. A larger surface for ion production is obtained with this solution, and its efficacy has been demonstrated by experimental results on the Batman ion source at IPP Garching, visible in Fig. 5.3a [6]. A 4 kA current flows in the vertical direction, to provide a horizontal magnetic field that reduces electron temperature and the number of co-extracted electrons. Flexible electric connections are foreseen between the four horizontal segments of the PG. They feature a number of flexible lamellae in order to allow vertical and horizontal relative displacements between the segments (Fig. 5.4).

The number of co-extracted electrons could be very different during the various phases of the operations (conditioning, partial power, full power etc.), mainly be-

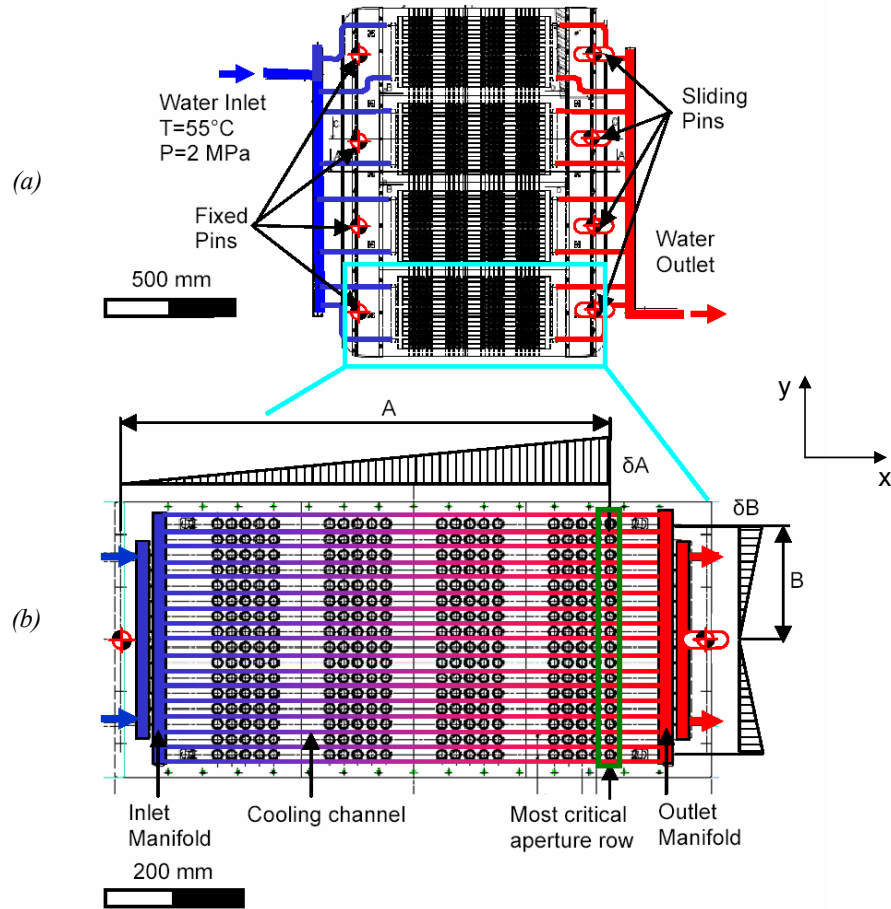


Figure 5.2: PG, EG and PAG cooling and positioning scheme: (a) Whole grid;(b) Segment.

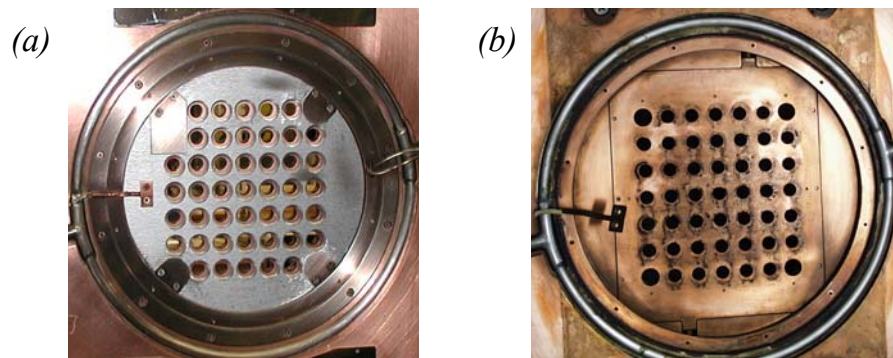


Figure 5.3: BATMAN accelerator grids: (a) View of the Plasma Grid, made of molybdenum; (b) View of the Extraction Grid, made of copper

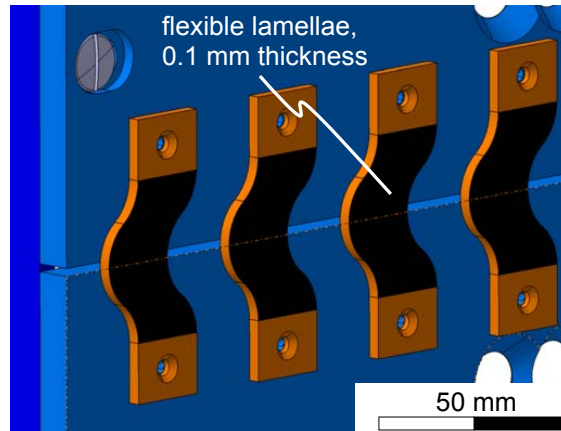


Figure 5.4: Typical flexible connection between PG segments.

cause the electron-to-ion ratio is expected to be a function of caesium deposition on the PG, which depends on PG temperature and operating conditions [6]. Hence, the EG and PAG are designed in order to satisfy the thermal, structural and alignment requirements in different scenarios with the total power ranging from zero to the double of the nominal power (which is calculated with electron-to-ion ratio equal to the unit).

The extraction grid has an electric potential that is 9.6 kV higher than the PG, so that the negative charged ions (H^- or D^-) can be properly extracted from the RF expansion chamber. Suppression magnets, embedded in the grid, have the function to deviate the trajectories of the co-extracted electrons, making them collide with the grid surface. The consequent power loads are quite high (464 kW over the whole grid in the nominal operating conditions), hence this grid is the most critical by the structural point of view, and is designed with a high performance cooling system. In case of insufficient cooling, serious damages can happen on the surface of this grid, as shown in Fig. 5.3b.

The pre-acceleration grid has the function to accelerate the ion beamlets up to a potential of about 40 kV. Also on this grid there are embedded magnets (suppression and steering). The total power deposition foreseen on this grid in the nominal operating condition is 89 kW.

The GG assembly (Fig. 5.5) is positioned about 350 mm downstream the pre-acceleration grid. This grid is relatively thick (70 mm) with sixteen (4x4) large apertures, and “V-shaped” along the vertical direction to provide vertical beam groups steering [4].

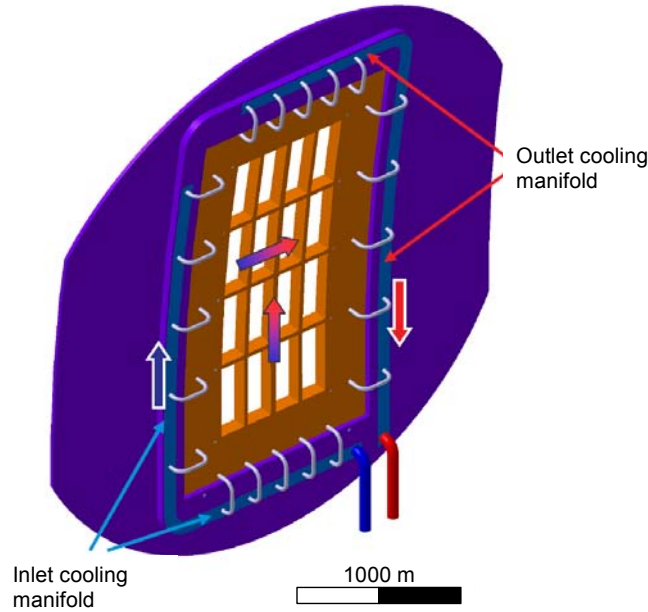


Figure 5.5: GG assembly view with cooling scheme.

5.2 FEM models description

Two different kinds of models of the accelerator grids have been developed for the first three grids, in order to optimize different aspects of the design:

- **Segment models:** These FEM models, which represent a whole segment of the grids, were used to evaluate different cooling and fixing schemes. Several different design solutions were evaluated in terms of segment deformation (on plane and out of plane), pressure drop, water velocity and other hydraulic parameters.
- **Aperture models:** These FEM models, that represent only a little part of the grid, were used to precisely estimate the stresses and strain of the material near the hot spot. In particular, the analyses were made for the extraction grid, which is the most critical, are reported in the next chapter. As the material is subjected also to plastic deformation in these hot spot, elasto-plastic elements were chosen, to better simulate the material behaviour.

5.2.1 Material properties

The thermo-mechanical properties of electrodeposited copper are not presently available. The properties of fully annealed oxygen free high conductivity copper (OFHC-Cu), taken from the ITER Material Properties Handbook (MPH) [7] are used for

Temperature	Density	Young's Modulus	Poisson's Ratio	Mean Thermal Expansion	Thermal Conductivity	Specific Heat
[°C]	[Kg/m ³]	[GPa]	adimensional	[10 ⁻⁶ , 1/K]	[W/(m·K)]	[J/(kg·K)]
20	8940	117	0.33	16.9	391	383
50		116		17.0	389	386
100		114		17.2	385	392
150		112		17.4	382	397
200		110		17.7	378	403
250		107		18.1	375	408
300		104		18.4	372	413
350		101		18.8	368	419
400		98		19.2	365	424
450		94		19.7	361	430
500		90		20.1	358	435

Table 5.1: Physical properties of pure copper.

all the analyses (see Tab. 5.1). Discussions with experts of European Associations pointed out that the actual properties of electrodeposited copper should be close to the OFHC-Cu ones, except the tensile strength that should be sensibly higher in case of electrodeposited copper. The IPP experience confirmed this information.

Anyway a full test campaign to measure the actual properties of electro-deposited Cu is foreseen, especially regarding the fatigue properties at different temperatures.

5.2.2 Convection heat transfer calculation

Heat transfer in the cooling channel is given by convection process, and is proportional to film coefficient at and difference between tube inner wall surface temperature T_i and bulk fluid temperature T_f :

$$q(T_i) = \alpha(T_i - T_f) \quad (5.1)$$

The Sieder-Tate correlation (valid for forced convection in the single phase flow) [8] for the heat transfer coefficient is utilised :

$$\alpha = Nu_t \cdot \frac{\lambda}{D_h} Nu_t = 0.027 \cdot Re^{0.8} \cdot Pr^{\frac{1}{3}} \quad (5.2)$$

where:

- λ is the thermal conductivity of water
- $Re = \frac{v \cdot D_h}{\nu}$ is the Reynolds number
- $Pr = \frac{C_{water} \cdot \mu}{\lambda}$ is the Prandtl number

- v is water velocity inside the channel
- D_h is the hydraulic diameter of the channel
- ν is the cinematic water viscosity
- C_{water} is the water specific heat
- μ is the water dynamic viscosity

This formula is implemented in the FEM model to calculate heat transfer.

5.2.3 Pressure drop calculation

Pressure drop along the cooling channels is calculated with Darcy-Weisbach formula [9] :

$$dp = \xi \cdot \frac{\rho v^2}{2} \cdot \frac{dL}{D_h} \quad (5.3)$$

where:

- dp is the pressure drop in a dL segment of channel
- ξ is the friction factor in a cooling channel
- ρ is water density
- v is water velocity inside the channel
- D_h is the hydraulic diameter of the channel

The Colebrook-White formula is utilized, that gives the friction factor ξ in function of Re and e/D_h (relative roughness)[9] :

$$\frac{1}{\sqrt{\xi}} = -2 \cdot \left[\frac{2.51}{Re \cdot \sqrt{\xi}} + \frac{\varepsilon}{3.71 \cdot D_h} \right] \quad (5.4)$$

with $\varepsilon = 0.01$ mm (maximum absolute roughness in copper tubes, conservative value) [10]

Type	Fluid-thermal analysis	Structural analysis
1. Solid elements	SOLID70	SOLID45
2. Fluid elements	FLUID116	Not used
3. Tube surface elements	SURF152	Not used

Table 5.2: Segment model elements summary.

5.2.4 Segment models

These FEM models, which represent a whole segment of the grids, were used to evaluate different cooling and fixing schemes. Several different design solutions were evaluated in terms of segment deformation (on plane and out of plane), temperature and stress distribution, pressure drop and water velocity. The software utilized for this analysis is ANSYS 11.0 Classic. The analyses are non-linear thermo-mechanical, being the material properties not constant with temperature (see paragraph 5.2.2) and also the equations of heat exchange not linear functions of the water parameters (see paragraph 5.2.3).

To obtain better performance from the software, we had to perform first the fluid-thermal analysis, which calculates the temperature on the grid and the hydraulic quantities along the fluid line, and then the structural analysis, which calculates stresses and displacements (due both to the temperature distribution and to the fluid pressure inside the tubes).

As regards to fluid-thermal analysis, the used elements are:

- SOLID70 (1 degree of freedom per node: Temperature) for grid (copper thermal properties)
- FLUID116 (2 degree of freedom per node: Temperature and pressure) for fluid line (water thermal and hydraulic properties)
- SURF152 (1 degree of freedom per node: Temperature) to calculate the convection heat transfer between solid and fluid elements (using the formulas of the previous paragraph). These elements cover channels' surfaces.

As regards to structural analysis, the used elements is:

- SOLID45 (3 degree of freedom per node: x,y,z translations) for grid (copper mechanical properties).

In order to link together the solid, surface and fluid elements, the mesh of these models had to be made completely "by hand". For this reason the shape and position of the elements are very regular.

5.3 Plasma Grid analyses on segment models

The plasma grid is heated by the plasma inside the RF ion source, with a surface power density that is estimated to range between 3 and 20 kW/m² (IPP experimental results). This grid is required to operate at a temperature of about 150° C in order to enhance the caesium effect for negative ions surface generation. Hence its cooling system is designed to obtain a precise control of the temperature, with a good uniformity on the whole surface.

The model considers one segment without lateral edges (main dimensions 760 mm x 395 mm) of the Plasma Grid. The 320 apertures have a minimum diameter of 14 mm. The aperture shape is conical, with a converging part at the upstream side and a diverging part at the downstream side. The main requirements that drive the PG thermo-mechanical design are:

- To have an optimized PG surface shape to increase the ion current density. This means to found a compromise between two opposite requirements:
 1. A PG surface as larger as possible permits to create as many as possible H⁻/D⁻ negative ions. This is due to the fact that the ions are mainly created on the PG surface on the RF chamber side.
 2. The negative ions created on the surface must be able to enter on the apertures to be extracted and then accelerated.
- The active surface can be increase designing a conical chamfer around the aperture: the lower is the cone angle, the larger is the surface on the RF chamber side. But if the angle is too small, the cesium cannot reach efficiently the surface, and also the negative ions have a high probability to collide with the PG surface, without being extracted. A good compromise was experimentally found using a 90° angle for the conical chamfer at the upstream side. The chamfer at downstream side, instead, has an conical aperture angle of 80° and is used to ameliorate the optics, in particular the meniscus shape.
- To have a PG surface temperature in the range 150°C±10°. The value of 150°C comes from the experiments with the sources BATMAN and MANITU, where it was investigated the relation between the extracted ion density and the PG temperature. It was found that the temperature of 150° C is the optimum for the surface generation of negative ions, enhanced by cesium. For this reason, the water is inlet at 150°C temperature. The range ±10° comes from the observation in the same experiments that the ion current density is highly dependent on the PG temperature around the aperture. Hence, to

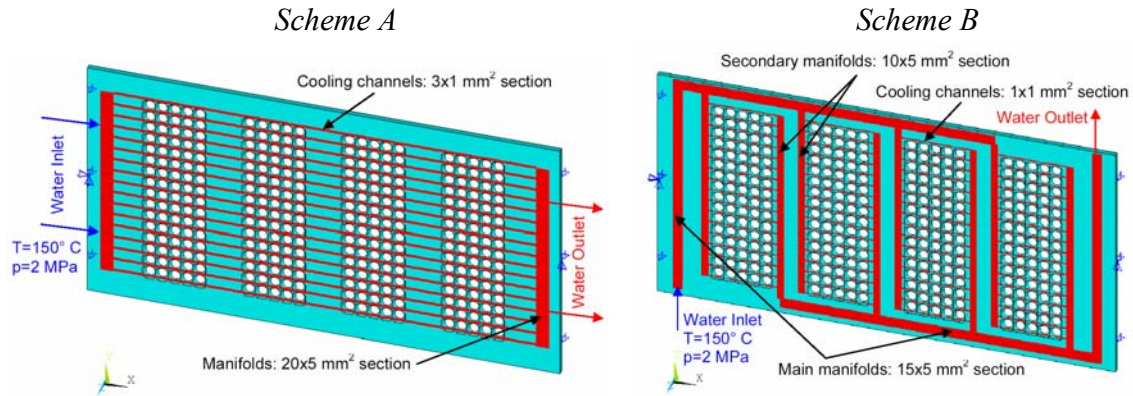


Figure 5.6: Two concepts for the cooling system of the Plasma Grid, with structural boundary conditions.

obtain a uniform ion current density in all the apertures, it is important to have the nearly the same temperature around all the apertures.

Two different cooling schemes have been considered (see Fig. 5.6):

- Scheme A: Similar to the reference scheme of DDD5.3, it features 17 horizontal channels in parallel that pass between the aperture rows. Two big manifolds at the edges have the function to feed the cooling channels and to balance the pressure, so that the water flow is the similar in all the channels.
- Scheme B: Proposed by IPP engineers, it features cooling channels that are approximately one fourth of the one in scheme A in length. These are feed by vertical manifolds that run vertically between the apertures 16x5 matrixes. Two main manifold (inlet and outlet) feed the secondary manifolds.

Both the models are here presented with a total water flow of 0.2 kg/s for the whole segment (0.8 kg/s for the whole grid).

5.3.1 Heat loads and boundary conditions

5.3.1.1 Heat loads

The heat power density on the plasma grid, coming from the plasma inside the main chamber (mainly by irradiation) is foreseen to be in the range between 3 and 20 kW/m², depending on the operating phase. The upper value is considered in the analysis, as it is more conservative. As the conical chamfers at the upstream side are inclined with an angle of 40° with respect to the aperture axis (see Fig. 5.7),

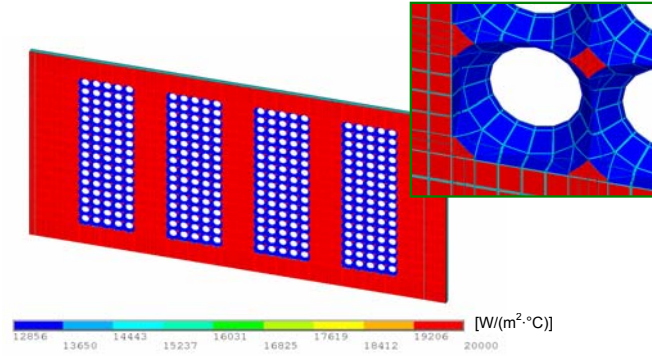


Figure 5.7: Heat load foreseen on the Plasma Grid.

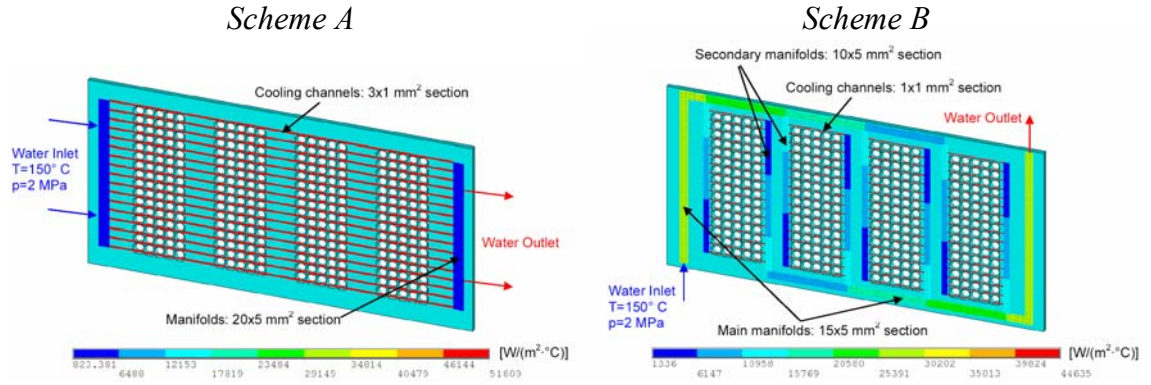


Figure 5.8: Convection Heat Transfer Factor on Plasma Grid with scheme A and B.

the power density impinging on them is estimated as:

$$\phi_{PG, chamfers} = \phi \cdot \sin(40^\circ) \quad (5.5)$$

$$= 12.8 \frac{kW}{m^2} \quad (5.6)$$

5.3.1.2 Structural boundary conditions

The structural boundary conditions (shown in Fig. 5.6) are applied in such a way, that they can simulate the fixing system shown in Fig. 5.2. An internal pressure of 2 MPa is applied inside the tubes and the manifolds.

5.3.2 Hydraulic results

The CHT coefficients in the cooling channels (see Fig. 5.8) are similar for the two cooling schemes, as they have a strong dependence from the water velocity (same in the two simulations considered) but only a weak one from the channels'

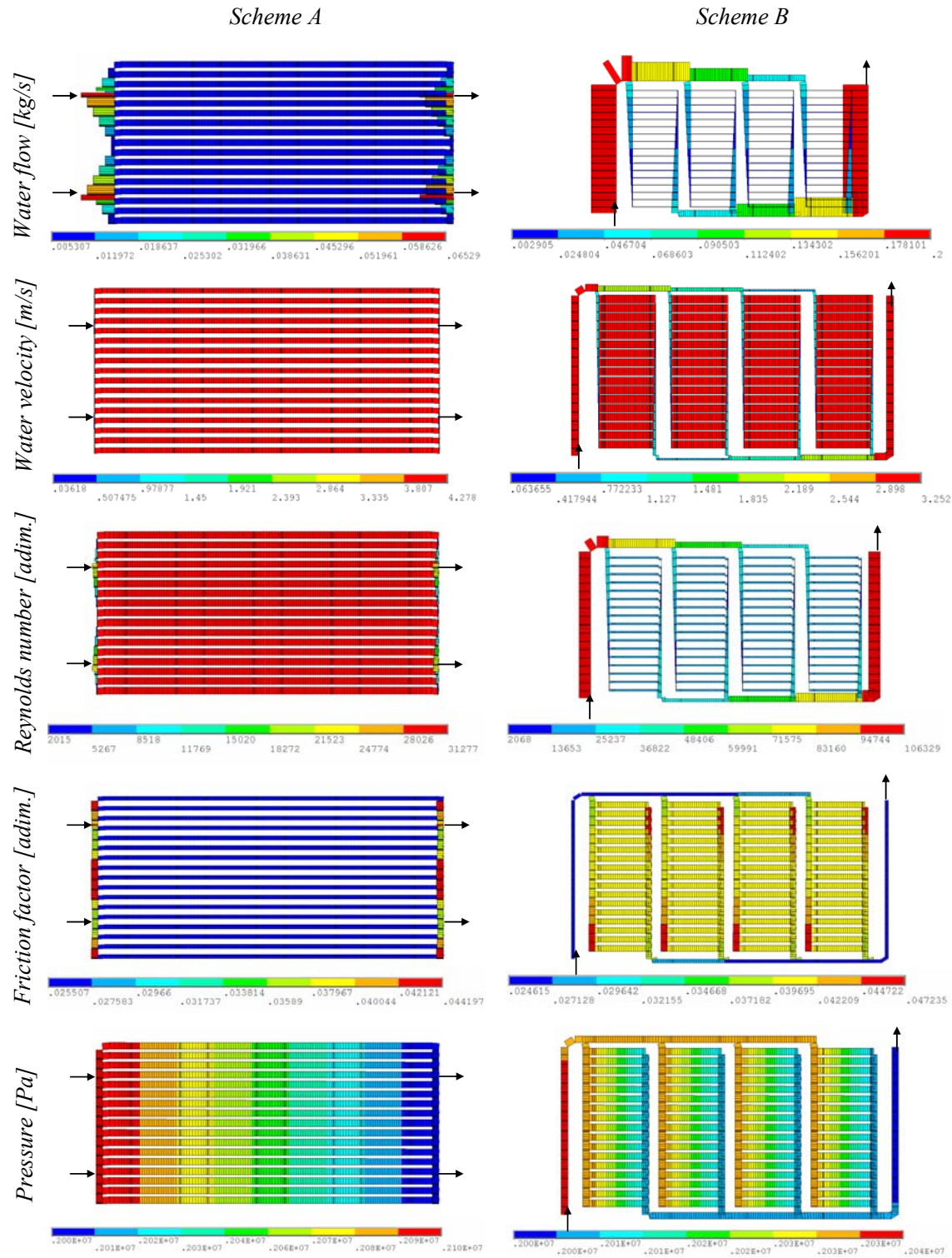


Figure 5.9: Comparison between hydraulic parameters on Plasma Grid with scheme A and B.

section. The upper and lower frames are better cooled with the scheme B, which foresees horizontal manifolds; the water velocity inside them is in the same order of magnitude of the one inside the cooling channels.

In Fig. 5.9 the most important outputs coming from the hydraulic analyses are compared. The values of the Reynolds number are averagely higher with the scheme B, while the friction factors (to apply in the Darcy-Weisbach formula) are similar with the two schemes. The total pressure drop is acceptable in both cases, but there is an important difference: while for scheme A there is almost no pressure drop inside the manifolds, for scheme B almost 40% of the pressure drop is inside the manifolds. This fact is due to the fact that the velocity inside the manifolds, that is higher with scheme B, that has smaller manifolds because of room limitations.

This can be a problem because the uniformity of the water flux is inversely proportional to the percentage of pressure drop inside the manifolds. In the case of the PG, this problem can be solved by using a smaller section of the cooling channels (in this way the considered percentage decreases), but for the EG, as the requested cooling requirements are more critical, this solution cannot be considered, as will be seen in the next chapter.

5.3.3 Thermal results

The main thermal results are shown in Fig. 5.10. With both cooling schemes, the water temperature increase is always acceptable, as it is lower than 10°C. As well, the grid temperature is inside the required range $150 \pm 10^\circ$ on the surface of the aperture area. Nevertheless, some hotter areas are present with both the schemes, where there are no cooling channels. However, the temperature of these zones are only slightly higher than the rest of the grid segment.

5.3.4 Structural results

5.3.4.1 Equivalent (Von Mises) stress

With both the cooling schemes, the maximum stress is caused by the water pressure inside the channels, as it is visible from Fig. 5.11. In both cases, these peaks are to be considered acceptable, as they are lower than the yield stress of the material. With scheme A, the highest peak (of 32.6 MPa) is located over the cooling channels, while with scheme B, the peak reaches 48.7 MPa and is located at the edge of a manifold.

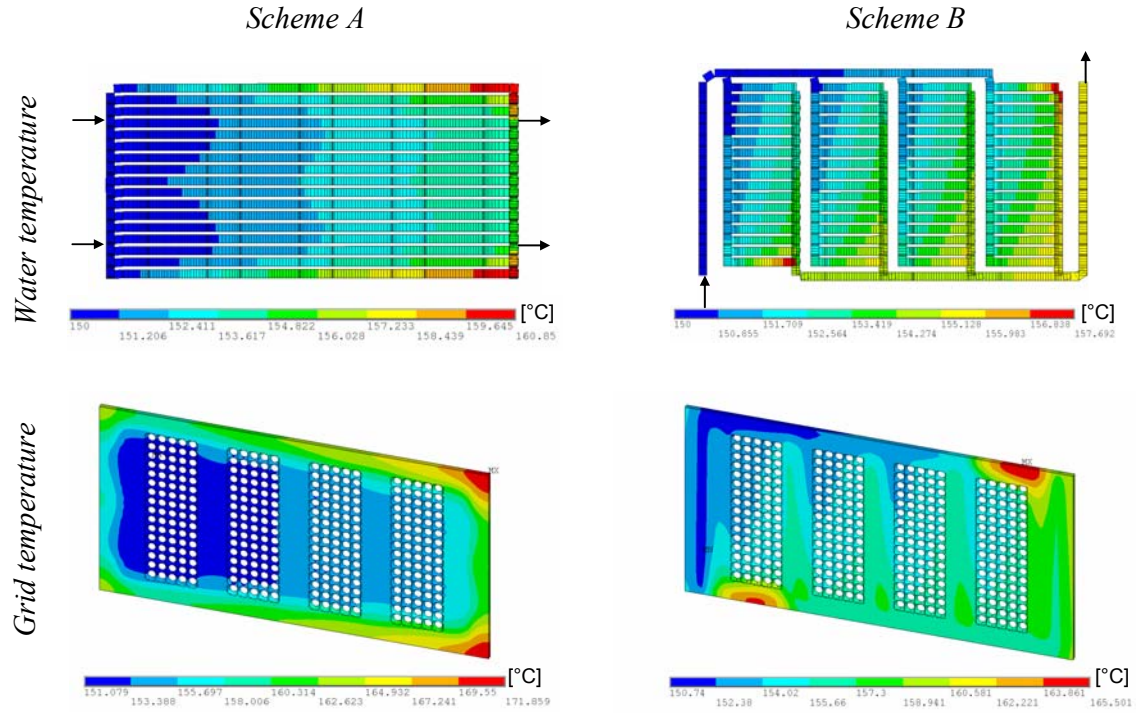


Figure 5.10: Temperature plots on Plasma Grid with scheme A and B.

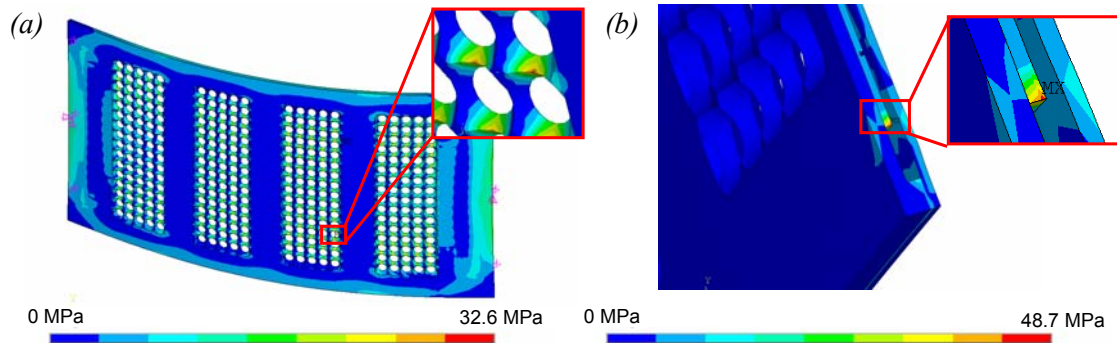


Figure 5.11: Equivalent (Von Mises) stress plots on Plasma Grid: (a) Maximum with cooling scheme A; (b) Maximum with cooling scheme B.

5.3.4.2 Grid deformation

The deformation along x and y axes (in-plane deformation) is almost identical with the two schemes. The deformation along z axis (out-of-plane deformation or bowing) is more than four times larger with scheme A (0.6 mm) compared to scheme B (0.14 mm), as shown in Fig. 5.12.

5.3.5 Alignment analysis

5.3.5.1 Analytical calculations to estimate in-plane deformations

The alignment between the corresponding apertures of first three grids (PG, EG and PAG) is an important parameter for the optic quality of the ion beam exiting from the accelerator. For this reason, the in-plane deformation (along x and y axis of Fig. 5.2) of these grid must be evaluated with good precision and in different operating scenarios.

A method is here illustrated to calculate analytically the horizontal displacement of the apertures (it can be used also to evaluate the vertical displacement). Firstly, the heating power deposited on a grid segment can be written as a function of the water flow parameters:

$$P = \dot{m} \cdot C_{water} \cdot (T_{water,out} - T_{water,in}) \quad (5.7)$$

where P is the power deposited on a segment, $\dot{m}$ the cooling water flow on a segment, C_{water} the specific heat of water, $T_{water,in}$ and $T_{water,out}$ the inlet and outlet temperatures of the cooling water. For every of the first three grids, the power P can be estimated with physics code [1], $T_{water,in}$ and $\dot{m}$ are imposed parameters, and C_{water} can be in first instance calculated as function of $T_{water,in}$. Hence we can estimate the outlet water temperature as:

$$T_{water,out} = T_{water,in} + \frac{P}{\dot{m} \cdot C_{water}} \quad (5.8)$$

The average temperature of the cooling water is:

$$\begin{aligned} T_{water,av} &= \frac{T_{water,in} + T_{water,out}}{2} \\ &= T_{water,in} + \frac{P}{2 \cdot \dot{m} \cdot C_{water}} \end{aligned} \quad (5.9)$$

Secondly, the heating power deposited on a grid segment can be written also as a function of the heat exchange parameters:

$$P = q_{av} \cdot A_{exchange} \quad (5.10)$$

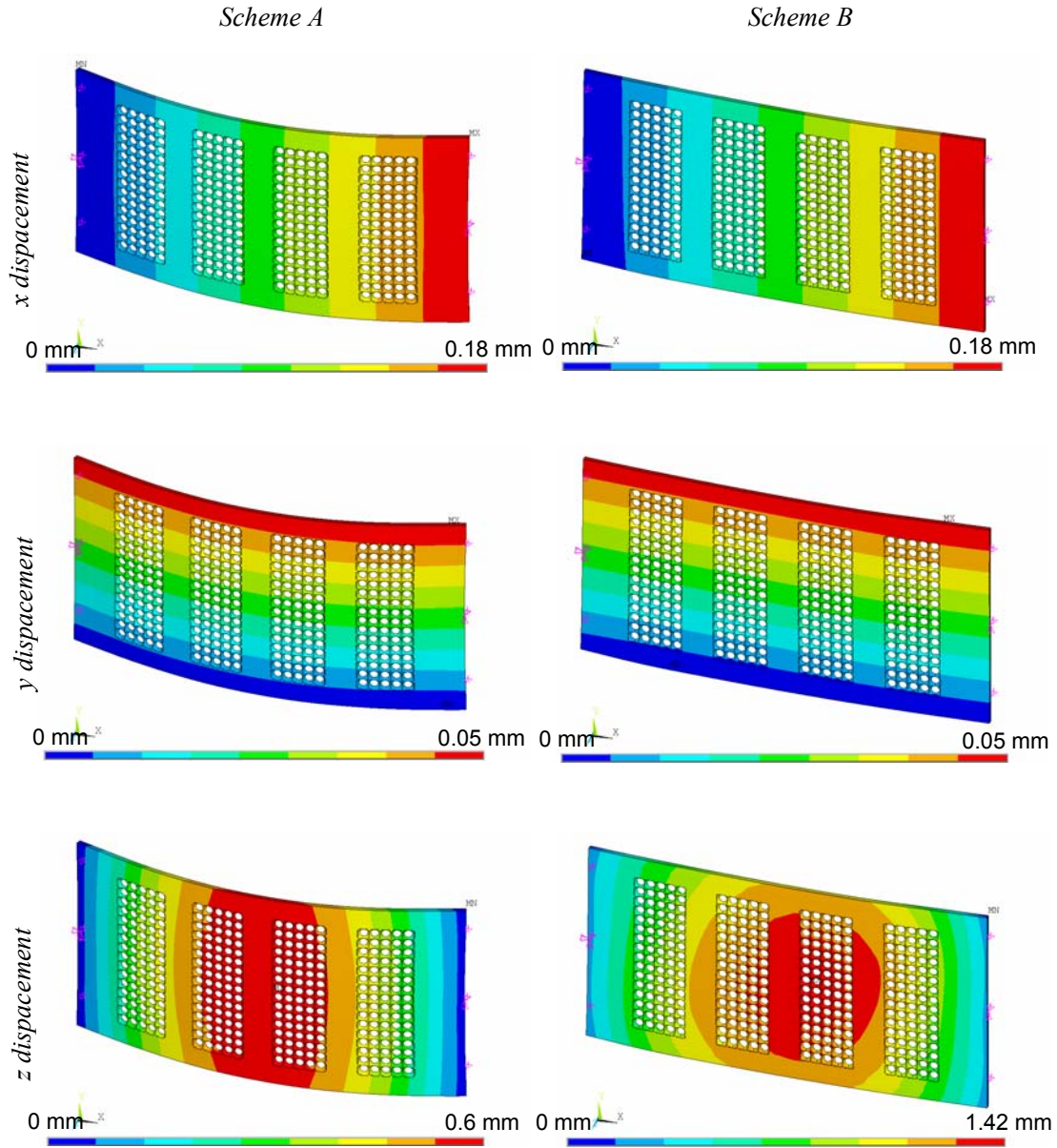


Figure 5.12: Comparison between displacements on Plasma Grid with scheme A and B (deformations amplified x200).

where q_{av} is the average power density exchanged between water and copper, while $A_{exchange}$ is the total interface area between water and copper, where there can be heat exchange. This can be calculated as the sum of the lateral walls of the cooling channels and the manifolds (for the PG, all the red areas in Fig. 5.8).

q is a function of the CHT coefficient α and the difference between walls and water temperature (equation 5.1). Considering average values for all the parameters, we obtain:

$$q_{av} = \alpha_{av}(T_{wall,av} - T_{water,av}) \quad (5.11)$$

The average CHT coefficient α_{av} can be estimated with the Sieder-Tate correlation :

$$\begin{aligned} \alpha_{av} &= Nu_{t,av} \cdot \frac{\lambda}{D_h} \\ &= 0.027 \cdot Re_{av}^{0.8} \cdot Pr_{av}^{\frac{1}{3}} \cdot \frac{\lambda}{D_h} \end{aligned} \quad (5.12)$$

where the average Re_{av} and Pr_{av} can be estimated as functions of the water and channels parameters (velocity, average temperature, hydraulic diameter).

Hence, inverting equation 5.11 we can calculate $T_{wall,av}$ as:

$$T_{wall,av} = T_{water,av} + \frac{q_{av}}{\alpha_{av}} \quad (5.13)$$

where $q_{av} = \frac{P}{A_{exchange}}$ (inverse of equation 5.10) and α_{av} is estimated with equation 5.12.

The thermal conductance of copper is very high, compared with the water/copper interface conductance:

$$\frac{\frac{\lambda_{Cu} \cdot A_{grid}}{s}}{\alpha_{av} \cdot A_{exchange}} > 20 \quad (5.14)$$

where λ_{Cu} is the thermal conductance of copper (taken from Tab. 5.1), A_{grid} is the area of a grid side (taken equal to 0.3 m²), s is a typical copper layer thickness (taken equal to 1 mm), α_{av} is an average CHT coefficient (taken equal to 50000 W/(°C · m²)) and $A_{exchange}$ is the water/copper heat exchange area (taken equal to 0.1 m²).

So, the hypothesis that the average grid temperature is equal to the average temperature of the wall channels can be made:

$$T_{grid,av} = T_{wall,av} \quad (5.15)$$

Summarizing in a single formula all the evaluations made, we can write $T_{grid,av}$ in function of known parameters:

$$T_{grid,av} = T_{water,in} + \frac{P}{2 \cdot \dot{m} \cdot C_{water}} + \frac{P}{A_{exchange} \cdot 0.027 \cdot Re_{av}^{0.8} \cdot Pr_{av}^{\frac{1}{3}} \cdot \frac{\lambda}{D_h}} \quad (5.16)$$

For a better approximation, this formula can be applied iteratively, in order to set the water properties in function of $T_{water,av}$.

For the three grids, the maximum absolute horizontal shift δA of the apertures (referred to the most critical aperture row highlighted in Fig. 5.2) can be so estimated:

$$\delta A_{PG} = \gamma_{Cu} \cdot A_{PG} \cdot (T_{average,PG} - T_{ref}) \quad (5.17)$$

$$\delta A_{EG} = \gamma_{Cu} \cdot A_{EG} \cdot (T_{average,EG} - T_{ref}) \quad (5.18)$$

$$\delta A_{PAG} = \gamma_{Cu} \cdot A_{PAG} \cdot (T_{average,PAG} - T_{ref}) \quad (5.19)$$

where the three $T_{average}$ are the average temperatures of the first three grids calculated as in paragraph 5.3.5.1, γ_{Cu} is the thermal expansion coefficient (from Tab. 5.1), the three A are the maximum horizontal distance of apertures from the fixed pin (see Fig. 5.2) and T_{ref} is the reference temperature (20° C).

The values calculated with this analytic model are almost identical to the ones calculated with numeric models (for example Fig. 5.12 shows the deformations for the PG), confirming that the hypothesis made are consistent.

5.3.5.2 Alignment optimization

The first three grids must be designed in such a way that the corresponding apertures are well aligned, in order to obtain good beam optics [11]. For this reason and for manufacturing requirements they are vertically split in four segments, independently supported with a fixed pin at the left side and with a sliding pin at the right side, as shown in Fig. 5.2.

A maximum allowable misalignment of ± 0.4 mm between corresponding apertures of PG, EG and PAG was identified as a compromise between the beam optic requirements and reasonable engineering values:

- ± 0.2 mm misalignment was estimated for manufacturing and positioning and the remaining
- ± 0.2 mm was then fixed as the maximum apertures misalignment due to thermal expansions of the grids.

In order to limit the apertures vertical displacements δ_B (see Fig. 5.2), the four horizontal segments are independently fixed and flexible electrical connections are foreseen between PG segments. Each segment is fixed at the left side of the grids and horizontally sliding at the right side. Hence the apertures located at the right-end part of the grids (highlighted with the green rectangle in Fig. 5.2) are the most critical with regard to horizontal misalignment.

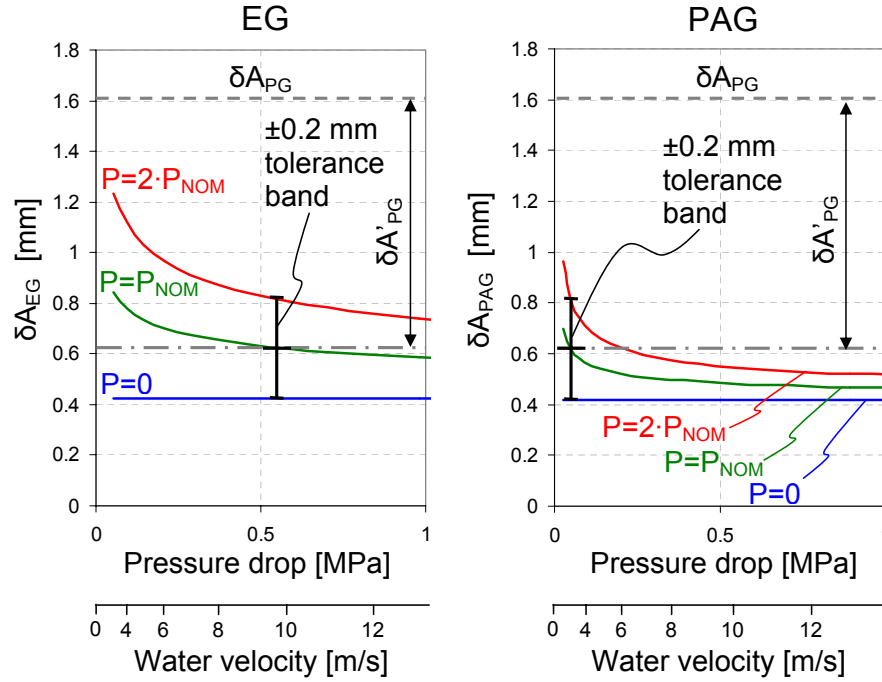


Figure 5.13: Identification of PG apertures pre-offset and of the minimum EG and PAG cooling parameters for alignment optimization).

The maximum horizontal misalignments M of the EG and PAG apertures with respect to the PG apertures are:

$$M_{EG} = |\delta A_{PG} - \delta A_{EG}| \quad (5.20)$$

$$M_{PAG} = |\delta A_{PG} - \delta A_{PAG}| \quad (5.21)$$

As the working temperature of the three grids are different (around 150°C for the PG, in the range $50 \div 70^\circ\text{C}$ for EG and PAG, depending on the power scenario), a possible solution to maintain the alignment is to introduce a mechanical offset $\delta A'_{PG}$ of the PG apertures. Introducing this mechanical offset, the misalignments M'_{EG} and M'_{PAG} between corresponding apertures become:

$$M'_{EG} = |\delta A_{PG} - \delta A'_{PG} - \delta A_{EG}| \quad (5.22)$$

$$M'_{PAG} = |\delta A_{PG} - \delta A'_{PG} - \delta A_{PAG}| \quad (5.23)$$

Imposing the alignment requirements $M'_{EG} < 0.2 \text{ mm}$ and $M'_{PAG} < 0.2 \text{ mm}$ under power loads ranging from zero to the double of nominal power, the horizontal pre-offset of the PG apertures can be identified ($\delta A'_{PG} = 1 \text{ mm}$), as well as the minimum water velocities along the channels (10 m/s for EG and 4 m/s for PAG) and the

corresponding pressure drops (see Fig. 5.13).

As it is visible from Fig. 5.12, the behavior with schemes A and B is almost identical as regards to the in-plane deformations (x and y directions); hence, the aspect of the alignment optimization is not important for the choice between the two cooling schemes.

5.3.6 Overall comparison between the two schemes

A comparison between the two cooling schemes is made, considering the Pugh Method and the Analytical Hierarchy Process.

5.3.6.1 Pugh Method

The Pugh Method is applied considering the scheme A as datum. Ten Critical to Quality (CTQ) criteria are introduced to compare the two schemes (see Tab. 5.3). For every criterion, a plus sign is given to the scheme that behaves better, a minus to the other; if the behavior is similar regarding that criterion, an equal sign is given to both the schemes.

At the end of the comparison, the plus and minus signs are counted for scheme B, and a discussion is made to check if it is worth to change the cooling scheme from A to B, and especially to consider other solutions that may arise from the existing ones.

In the present case, the negative aspects of the scheme B make it less suitable than scheme A for the ITER NBI Plasma Grid. Nevertheless, the Pugh method process considered has helped the design team to fix some aspects that could be improved in the reference scheme A, like the section and position of the cooling channels. At the end, it can be said that the final solution took some parts also from the discarded concepts.

5.3.6.2 Analytic Hierarchy Process

The Analytic Hierarchy Process is applied considering the following criteria, considered the most important for the operation of the component:

- **Low pressure drop.** This gives less energy consumption and less risk of boiling inside the channels.
- **Temperature uniformity.** Together with a uniform cesium distribution on the PG, a uniform temperature gives a uniform production of negative ions near the PG.

CTQ	Scheme A (datum)	Scheme B
Pressure drop (Lower is better)	Higher pressure drop (but still acceptable) (-)	Lower pressure drop, due to the fact that the cooling channels are not "in array" but "in parallel". (+)
Cooling channels section (Larger is better)	Larger cross section ($3 \times 1 \text{ mm}^2$) (+)	Smaller cross section ($1 \times 1 \text{ mm}^2$). The cross section must be smaller than in scheme A to obtain a good uniformity of the water flow on the grid. This can be done only if most of the pressure drop (at less 70%) is concentrated in the cooling channels. If a $3 \times 1 \text{ mm}^2$ cross section is utilised with scheme B, the pressure drop is mostly concentrated in the manifolds, which gives bad flow uniformity. In fact, in this case the water flow higher at the borders and lower in the middle of the grid. (-)
Water velocity along manifolds (Lower is better)	The water velocity along the manifolds is very low. (+)	The water velocity along the manifolds is on the same order of magnitude of the one inside the cooling channels. Hence, there could be some problems of corrosion and excessive pressure drop. (-)
Pressure drop along manifolds (Lower is better)	At the sides, there is enough space to make manifolds with a big cross section (similar to tanks). Hence, the pressure drop in the manifolds is negligible. (+)	The manifolds are quite long and with ramification, and there is not much space to build manifolds in the upper/lower edge and between the aperture groups. The pressure drop on the manifolds is not negligible. (-)
Temperature uniformity (High uniformity is better)	The temperature is higher at the borders (maximum 10°C higher), but this disuniformity can be avoid by using two more cooling channels in the upper and lower border. (+)	There are two hotter regions, where the temperature near the apertures is about 10°C higher than the rest of the grids. This can be avoided by adding two more tubes in that region. The upper region, where the cold waters is inlet, is slightly colder (5°C in average) than the lower region. This effect is spread on a large area, and could cause a different expansion of the higher and lower part of the segment. (-)
Stress (Lower is better)	The maximum VM stress, due to the water pressure (primary stress), is reached along the cooling channels. It is to be considered acceptable. (=)	The maximum VM stress, due to the water pressure (primary stress), is reached at the ramification of the manifolds. It is to be considered acceptable. (=)
Manufacturing (Easier is better)	Easier manufacturing (+)	More difficult manufacturing. The cooling channels have a smaller width (only 1 mm) and they are not continuous from side to side. There are more manifolds. (-)
Cost (Cheaper is better)	Cheaper manufacturing (+)	More expensive manufacturing. In particular, the milling cost of the cooling channels is about 6 times higher. (-)
In plane deformation (Lower is better)	The average values for in plane deformation are not depending on the cooling scheme but only on the heat load and water temperature flow and inlet temperature. (=)	
Out of plane deformation (Lower is better)	Higher out of plane deformation (maximum 0.6 mm). According to the optics calculations, this deformation is to be considered acceptable. (-)	Lower out of plane deformation (maximum 0.14 mm) (+)
Total positives		2+
Total negatives		6-

Table 5.3: PG: Overall comparison between scheme A and B, following the Pugh Method.

	Low pressure drop	Temp. uniformity	Low stress	Low deformations	Low cost
Low pressure drop	1	1/3	1/2	1/5	2
Temp. uniformity	3	1	1/2	2	7
Low stress	2	2	1	1/3	6
Low deformations	5	1/2	3	1	9
Low cost	1/2	1/7	1/6	1/9	1

Table 5.4: Preferences on Objectives.

- **Low stress.** This gives a high margin in static verifications and a high fatigue life for the component
- **Low deformations.** The out of plane deformations of the grid can change the properties of the ion beam (optics, power etc.), so they could be as low as possible
- **Low cost.** The cost is always an important parameter for the choice of a design concept or solution.

A comparison between couples of criteria is made, based on the experimental data and on the opinion of experienced people.

For example, low deformation is considered 5 times more important than low pressure drop, and temperature uniformity 7 times more important than cost. In this way, a matrix with the preference on objectives is created (Tab. 5.4). The matrix above can be normalized, in order to obtain the “weight on objectives” matrix (Tab. 5.5) that reports on the last column the normalized relative importance of every criteria (with sum equal to the unity).

The next step is to evaluate the suitability of the two cooling schemes A and B, with respect to the five considered criteria. This evaluation is made considering the FEM analysis reported in the previous paragraph, and manufacturing considerations. For example, as the peak stress is 50% higher with scheme B (and the corresponding fatigue life about three times less), this scheme is considered three times less suitable to satisfy the “low stress” criterion.

Tab. 5.7 summarizes the normalized values taken from the “scores” matrixes in Tab. 5.6. These values are multiplied for the weights of Tab. 5.5, in order to obtain a total score for the two solutions, taking into account all the evaluations done.

	Low pressure drop	Temp. uniformity	Low stress	Low deformations	Low cost	Average
Low pressure drop	0.087	0.084	0.097	0.055	0.080	0.080
Temp. uniformity	0.261	0.251	0.097	0.549	0.280	0.288
Low stress	0.174	0.503	0.194	0.091	0.240	0.240
Low deformations	0.435	0.126	0.581	0.274	0.360	0.355
Low cost	0.087	0.084	0.097	0.055	0.080	0.080

Table 5.5: Weight on Objectives.

Low pressure drop			Temp. uniformity			Low stress			Low deformations			Low cost		
	A	B		A	B		A	B		A	B		A	B
A	1	1/2	A	1	4	A	1	3	A	1	1/2	A	1	7
B	2	1	B	1/4	1	B	1/3	1	B	2	1	B	1/7	1

Table 5.6: Scores.

	A	B
Low pressure drop	0.333	0.667
Temp. uniformity	0.8	0.2
Low stress	0.75	0.25
Low deformations	0.333	0.667
Low cost	0.875	0.125

Table 5.7: Relative scores for each objective.

The total score for the scheme A is:

$$(0.08)(0.333) + (0.288)(0.8) + (0.240)(0.75) + (0.355)(0.333) + (0.03)(0.875) = 0.587$$

while the total score for the scheme B is:

$$(0.08)(0.667) + (0.288)(0.2) + (0.240)(0.25) + (0.355)(0.667) + (0.03)(0.125) = 0.413$$

As the total score is higher for the cooling scheme A, this solution can be considered better suitable and is chosen as reference for the ITER NBI. Nevertheless, if the boundary conditions are different, like for the ELISE facility at IPP, also the design choice to be considered optimal can be different.

Bibliography

- [1] H. P. L. de Esch. *TW4-THHN-IITF2 Final Report- Deliverable 3.1.4.4 Electrons in SINGAP*. EFDA report 2006.
- [2] H.P.L. de Esch, R.S. Hemsworth and P. Massmann. *SINGAP: the European concept for negative ion acceleration in the ITER neutral injectors*. Rev. Sci. Inst. 73 (2002) 1045.
- [3] H.P.L. de Esch, R.S. Hemsworth and P. Massmann. *Updated physics design ITER-SINGAP accelerator*. Fusion Eng. Des. 73 (2005) 329.
- [4] P. Agostinetti, S. Dal Bello, M. Dalla Palma, D. Marcuzzi, P. Zaccaria, R. Nocentini, M. Fröschle, B. Heinemann and R. Riedl. *Thermo-mechanical design of the ITER Neutral Beam Injector grids for Radio Frequency Ion Source and SINGAP Accelerator*. 22nd IEEE/NPSS Symposium on Fusion Engineering, Albuquerque, New Mexico, 2007.
- [5] P. Agostinetti, S. Dal Bello, M. Dalla Palma, P. Zaccaria. *Thermo-mechanical design of the SINGAP accelerator grids for ITER NB Injectors*. Fusion Eng. Des. 82 (2007) 860-866.
- [6] E. Speth, H.D. Falter, P. Franzen, U. Fantz, M. Bandyopadhyay, S. Christ, A. Encheva, M. Fröschle, D. Holtum, B. Heinemann, W. Kraus, A. Lorenz, Ch. Martens, P. McNeely, S. Obermayer, R. Riedl, R. Süss, A. Tanga, R. Wilhelm and D. Wunderlich. *Overview of the RF source development programme at IPP Garching*. Nucl. Fusion 46 (2006) 220-238.
- [7] Various Authors. *ITER MPH. Material Properties Handbook*. ITER Document No. G 74 MA 16.

-
- [8] J.H. Lienhard IV and J.H. Lienhard V. *A Heat Transfer Handbook*. Third ed., Phlogiston Press, 2003.
 - [9] M. E. Meadows and T. M. Walski. *Computer Applications In Hydraulic Engineering*. Haestad Press, 2002.
 - [10] Various Authors. *Manuale dell'Ingegnere Meccanico*. Hoepli, second ed., January 2005.
 - [11] H.P.L. de Esch. *ITER-SINGAP sensitivity study*. Final report EFDA contract TW4-THHN-IITF2, Deliverable 3.1.4.3.

Chapter 6

Numerical methods for the design of high performance cooling systems

SUMMARY - In this chapter the most important models for Computational Fluid Dynamics (CFD) are described. In order to select the best CFD model to simulate the water flowing along a cooling channel, the three approaches are compared to the analytical formulas and experimental data. A new FEM method, that integrates non-linear CFD, thermal and structural analysis is then presented.

6.1 Introduction

Computational fluid dynamics (CFD) is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the millions of calculations required to simulate the interaction of fluids and gases with the complex surfaces used in engineering. However, even with simplified equations and high-speed supercomputers, only approximate solutions can be achieved in many cases. More accurate software that can accurately and quickly simulate even complex scenarios such as transonic or turbulent flows are an ongoing area of research. Validation of such software is often performed using a wind tunnel.

In the present case, the aim is to use CFD tools to simulate high performance cooling system, used inside high power ion sources. In order to obtain high cooling capabilities, the fluid velocities in these system are considerably high, but the typical hydraulic diameters are quite small. The Reynolds numbers range between 5000 and 100000 (depending on the operating scenarios), hence the water flows can be in the fully developed turbulence zone, as well as in the transition zone (see the Moody

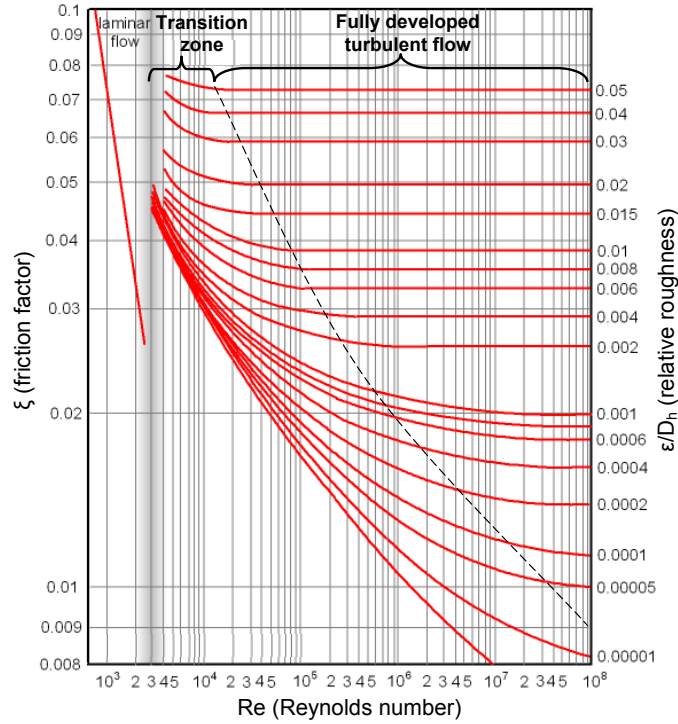


Figure 6.1: The Moody diagram.

diagram in Fig. 6.1). Hence, the most sophisticated turbulence models, able to fit a large amount of experimental data both in the transition and fully developed turbulence zones, are described in this chapter. A comparison between them and with the analytical formulae is carried out considering a simple model of a circular tube, with the typical flow regime (Reynolds number and velocity) considered for the cooling system of the ion source.

Once chosen the most suitable turbulence model, an integrated CFD-thermo-structural FEM technique, particularly developed for the simulation of high performance cooling systems, is introduced.

6.2 Modeling the turbulent flow

Turbulence consists of fluctuations in the flow field in time and space. It is a complex process, mainly because it is three dimensional, unsteady and consists of many scales. It can have a significant effect on the characteristics of the flow. Turbulence occurs when the inertia forces in the fluid become significant compared to viscous forces, and is characterized by a high Reynolds Number.

In principle, the Navier-Stokes equations describe both laminar and turbulent

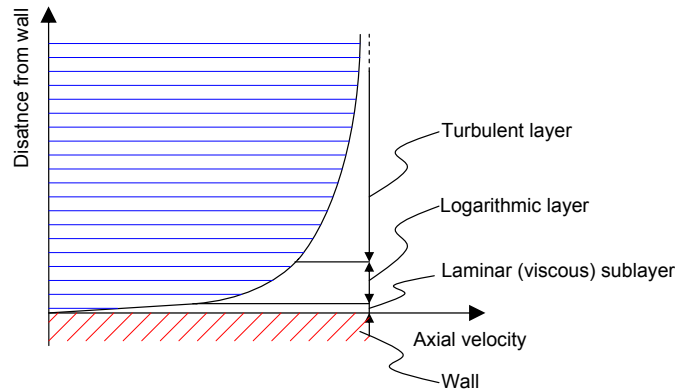


Figure 6.2: A typical velocity profile of a turbulent flow near the wall.

flows without the need for additional information. However, turbulent flows at realistic Reynolds numbers span a large range of turbulent length and time scales, and would generally involve length scales much smaller than the smallest finite volume mesh, which can be practically used in a numerical analysis. The Direct Numerical Simulation (DNS) of these flows would require computing power which is many orders of magnitude higher than available in the foreseeable future [1].

To enable the effects of turbulence to be predicted, a large amount of CFD research has concentrated on methods which make use of turbulence models. Turbulence models have been specifically developed to account for the effects of turbulence without recourse to a prohibitively fine mesh and direct numerical simulation.

It is an unfortunate fact that no single turbulence model is universally accepted as being superior for all classes of problems. The choice of turbulence model will depend on considerations such as the physics encompassed in the flow, the established practice for a specific class of problem, the level of accuracy required, the available computational resources, and the amount of time available for the simulation. To make the most appropriate choice of model for an application, it is needed to understand the capabilities and limitations of the various options [2].

The purpose of this section is to give an overview of issues related to the main turbulence models provided in the CFD software. While it is impossible to state categorically which model is best for a specific application, general guidelines are presented to help choosing the appropriate turbulence model for the flow to be modelled.

Very simple eddy viscosity models compute a global value for the turbulent viscosity from the mean velocity and a geometric length scale using an empirical formula. Because no additional transport equations are solved, these models are termed “zero equation”.

Two-equation turbulence models are very widely used, as they offer a good compromise between numerical effort and computational accuracy. They are much more sophisticated than the zero equation models [3]. Both the velocity and length scale are solved using separate transport equations (hence the term “two-equation”). In two-equation models, the turbulence velocity scale is computed from the turbulent kinetic energy, which is provided from the solution of its transport equation. The turbulent length scale is estimated from two properties of the turbulence field, usually the turbulent kinetic energy and its dissipation rate. The dissipation rate of the turbulent kinetic energy is provided from the solution of its transport equation [4].

The k - ε and k - ω two-equation models use the gradient diffusion hypothesis to relate the Reynolds stresses to the mean velocity gradients and the turbulent viscosity. The turbulent viscosity is modeled as the product of a turbulent velocity and turbulent length scale.

6.2.1 The Standard k - ε Model

One of the most prominent turbulence models, the k - ε model, has been implemented in most general purpose CFD codes and is considered the industry standard model. k is the kinetic energy of the turbulence, while ε is the dissipation per unit mass. It has proven to be stable and numerically robust and has a well established regime of predictive capability. For general purpose simulations, the k - ε model offers a good compromise in terms of accuracy and robustness. Within ANSYS CFX, the k - ε turbulence model uses the scalable wall-function approach to improve robustness and accuracy when the near-wall mesh is very fine. The scalable wall functions allow solution on arbitrarily fine near wall grids, which is a significant improvement over standard wall functions.

While standard two-equation models, such as the k - ε model, provide good predictions for many flows of engineering interest, there are applications for which these models may not be suitable. Among these are:

- Flows with boundary layer separation.
- Flows with sudden changes in the mean strain rate.
- Flows in rotating fluids.
- Flows over curved surfaces.

A Reynolds Stress model may be more appropriate for flows with sudden changes in strain rate or rotating flows, while the SST model may be more appropriate for separated flows.

The simplest “complete models” of turbulence are two-equation models in which the solution of two separate transport equations allows the turbulent velocity and length scales to be independently determined. The standard k - ε model in ANSYS CFX falls within this class of turbulence model and has become the workhorse of practical engineering flow calculations in the time since it was proposed by Launder and Spalding. k is the kinetic energy of the turbulence, while ε is the dissipation per unit mass. Robustness, economy, and reasonable accuracy for a wide range of turbulent flows explain its popularity in industrial flow and heat transfer simulations. It is a semi-empirical model, and the derivation of the model equations relies on phenomenological considerations and empiricism.

As the strengths and weaknesses of the standard k - ε model have become known, improvements have been made to the model to improve its performance.

6.2.2 The Shear-Stress Transport (SST) k - ω Model

One of the main problems in turbulence modeling is the accurate prediction of flow separation from a smooth surface. Standard two-equation turbulence models often fail to predict the onset and the amount of flow separation under adverse pressure gradient conditions. This is an important phenomenon in many technical applications, particularly for airplane aerodynamics since the stall characteristics of a plane are controlled by the flow separation from the wing. For this reason, the aerodynamic community has developed a number of advanced turbulence models for this application. In general, turbulence models based on the k - ε equation predict the onset of separation too late and under-predict the amount of separation later on. This is problematic, as this behavior gives an overly optimistic performance characteristic for an airfoil. The prediction is therefore not on the conservative side from an engineering stand-point. The models developed to solve this problem have shown a significantly more accurate prediction of separation in a number of test cases and in industrial applications. Separation prediction is important in many technical applications both for internal and external flows.

Currently, the most prominent two-equation models in this area are the k - ω based models of Menter [5], where ω parameter represents the dissipation per unit turbulence kinetic energy. The k - ω based Shear-Stress-Transport (SST) model was designed to give a highly accurate predictions of the onset and the amount of flow separation under adverse pressure gradients by the inclusion of transport effects into the formulation of the eddy-viscosity. This results in a major improvement in terms of flow separation predictions. The superior performance of this model has been demonstrated in a large number of validation studies [6]. The SST model is recommended for high accuracy boundary layer simulations. To benefit from this model,

a resolution of the boundary layer of more than 10 points is required. For details, see *Modelling Flow Near the Wall*.

For free shear flows, the SST model is identical to the $k-\varepsilon$ model. The SST model was developed to overcome deficiencies in the $k-\omega$ and BSL $k-\omega$ models. Therefore, using the SST model over these models is recommended.

One of the advantages of the $k-\omega$ formulation is the near wall treatment for low-Reynolds number computations where it is more accurate and more robust.

The convergence behaviour of the $k-\omega$ model is often similar to that of the $k-\varepsilon$ model. Since the zonal $k-\omega$ models (BSL and SST) include blending functions in the near wall region that are a function of wall distance, an additional equation is solved to compute the wall distance at the start of simulations (the first few iterations).

The standard $k-\omega$ model in ANSYS CFX is based on the Wilcox $k-\omega$ model [7, 1], which incorporates modifications for low-Reynolds-number effects, compressibility, and shear flow spreading. The parameter ω represent the dissipation per unit turbulence kinetic energy. The Wilcox model predicts free shear flow spreading rates that are in close agreement with measurements for far wakes, mixing layers, and plane, round, and radial jets, and is thus applicable to wall-bounded flows and free shear flows.

The shear-stress transport (SST) $k-\omega$ model is a variation of the standard $k-\omega$ model, developed by Menter [5] to effectively blend the robust and accurate formulation of the $k-\omega$ model in the near-wall region with the free-stream independence of the $k-\varepsilon$ model in the far field. To achieve this, the $k-\varepsilon$ model is converted into a $k-\omega$ formulation. The SST $k-\omega$ model is similar to the standard $k-\omega$ model, but includes the following refinements:

- The standard $k-\omega$ model and the transformed $k-\varepsilon$ model are both multiplied by a blending function and both models are added together.
- The blending function is designed to be one in the near-wall region, which activates the standard $k-\omega$ model, and zero away from the surface, which activates the transformed $k-\varepsilon$ model.
- The SST model incorporates a damped cross-diffusion derivative term in the ω equation.
- The definition of the turbulent viscosity is modified to account for the transport of the turbulent shear stress.
- The modeling constants are different.

These features make the SST $k-\omega$ model more accurate and reliable for a wider class of flows (e.g., adverse pressure gradient flows, airfoils, transonic shock waves) than the standard $k-\omega$ model.

6.2.3 The Reynolds Stress Model (RSM)

Two-equation turbulence models ($k-\varepsilon$ and $k-\omega$ based models) offer good predictions of the characteristics and physics of most flows of industrial relevance. In flows where the turbulent transport or non-equilibrium effects are important, the eddy-viscosity assumption is no longer valid and results of eddy-viscosity models might be inaccurate. Reynolds Stress (or Second Moment Closure (SMC)) models naturally include the effects of streamline curvature, sudden changes in the strain rate, secondary flows or buoyancy compared to turbulence models using the eddy-viscosity approximation. You may consider using a Reynolds Stress model in the following types of flow:

- Free shear flows with strong anisotropy, like a strong swirl component. This includes flows in rotating fluids.
- Flows with sudden changes in the mean strain rate.
- Flows where the strain fields are complex, and reproduce the anisotropic nature of turbulence itself.
- Flows with strong streamline curvature.
- Secondary flow.
- Buoyant flow.

Reynolds Stress models have shown superior predictive performance compared to eddy-viscosity models in these cases [8]. This is the major justification for Reynolds Stress models, which are based on transport equations for the individual components of the Reynolds stress tensor and the dissipation rate. These models are characterized by a higher degree of universality. The penalty for this flexibility is a high degree of complexity in the resulting mathematical system. The increased number of transport equations leads to reduced numerical robustness, requires increased computational effort and often prevents their usage in complex flows.[7]

Theoretically, Reynolds Stress models are more suited to complex flows, however, practice shows that they are often not superior to two-equation models. An example of this is for wall-bounded shear layers, where despite their (theoretically) higher degree of universality, Reynolds Stress models often prove inferior to two-equation

models. For wall-bounded flows try to use the SMC-BSL model. It is based on the ω -equation and automatic wall treatment.

Three varieties of the Reynolds Stress Model are available which use different model constants:

- Reynolds Stress Model (LRR-IP)
- QI Reynolds Stress Model (LRR-IQ)
- SSG Reynolds Stress Model (SSG)

In general, the SSG model is more accurate than the LRR versions for most flows. This is particularly true for swirling flows. The SSG model is therefore recommended over the other models, which are there for historic reasons and because they are standard models.

Compared to the k - ε model, the Reynolds Stresses model has six additional transport equations that are solved for each timestep or outer coefficient loop in the flow solver. The source terms in the Reynolds Stress equations are also more complex than those of the k - ε model. As a result of these factors, outer loop convergence may be slower for the Reynolds Stress model than for the k - ε model.

In principle, the same timestep can be used for all turbulence model variants, but pragmatically the timestep should be reduced for the Reynolds Stress Model due to the increased complexity of its equations and due to numerical approximations made at general grid interfaces (GGI) and rotational periodic boundary conditions. If convergence is difficult, it is recommended that a k - ε or k - ω based model solution be obtained first and then a Reynolds stress model solution can be attempted from the converged two-equation solution. It is frequently observed that Reynolds Stress models produce unsteady results, where two-equation models give steady state solutions. This can be correct from a physical standpoint, but requires the solution of the equations in transient mode.

The Reynolds Stress models may be used with isotropic or anisotropic turbulent diffusion terms in the Reynolds Stress and Epsilon transport equations. The difference is usually second order as there is often domination by the source terms and the effects of diffusion are small. An exception might be buoyant flows, which can be diffusion dominated. However, the model that uses isotropic turbulent diffusion terms is potentially more robust than the model that uses anisotropic turbulent diffusion terms.

The Reynolds stress model (RSM) is currently the most elaborate turbulence model provided by the main CFD codes. Abandoning the isotropic eddy-viscosity hypothesis, the RSM closes the Reynolds-averaged Navier-Stokes equations by solving transport equations for the Reynolds stresses, together with an equation for the

dissipation rate. This means that four additional transport equations are required in 2D flows and seven additional transport equations must be solved in 3D.

Since the RSM accounts for the effects of streamline curvature, swirl, rotation, and rapid changes in strain rate in a more rigorous manner than one-equation and two-equation models, it has greater potential to give accurate predictions for complex flows. However, the fidelity of RSM predictions is still limited by the closure assumptions employed to model various terms in the exact transport equations for the Reynolds stresses. The modeling of the pressure-strain and dissipation-rate terms is particularly challenging, and often considered to be responsible for compromising the accuracy of RSM predictions.

The RSM might not always yield results that are clearly superior to the simpler models in all classes of flows to warrant the additional computational expense. However, use of the RSM is a must when the flow features of interest are the result of anisotropy in the Reynolds stresses. Among the examples are cyclone flows, highly swirling flows in combustors, rotating flow passages, and the stress-induced secondary flows in ducts.

6.3 Comparison between the CFD models

A comparison between the three CFD models considered above is made, in order to decide which is the most suitable for the analyses on the cooling channels of the grid.

A very simple geometry is considered for the comparison, a circular channel with 10 mm diameter and 100 mm length, with a water velocity between 2 and 10 m/s. The circular channel is chosen because it is not only the industrial standard, but also the experimental standard. In fact, the most important and used analytic formulas for the calculation of the pressure drop and CHT coefficient were developed interpolating a big number of hydraulic experiments with this type of section. The size of the channel and the water velocity range are chosen similar to the ones of the Extraction Grid cooling system design. The mesh, shown in Fig. 6.3a, is optimized for CFD analyses: it features an inflated layer near the wall, where the elements are very dense in the direction normal to the surface. Fig. 6.3b, 6.3c and 6.3d show the most important outputs of the analysis: the velocity, the pressure and the CHT coefficients. The turbulence kinetic energy and the water velocity in function of the distance from the channel wall are shown in Figs. 6.4 and 6.5.

From Fig. 6.4 it is possible to appreciate the limit layer, where the turbulence kinetic energy assumes higher values than in the rest of the section. The axial velocity is lower in the limit layer, as visible from figure 6.5.

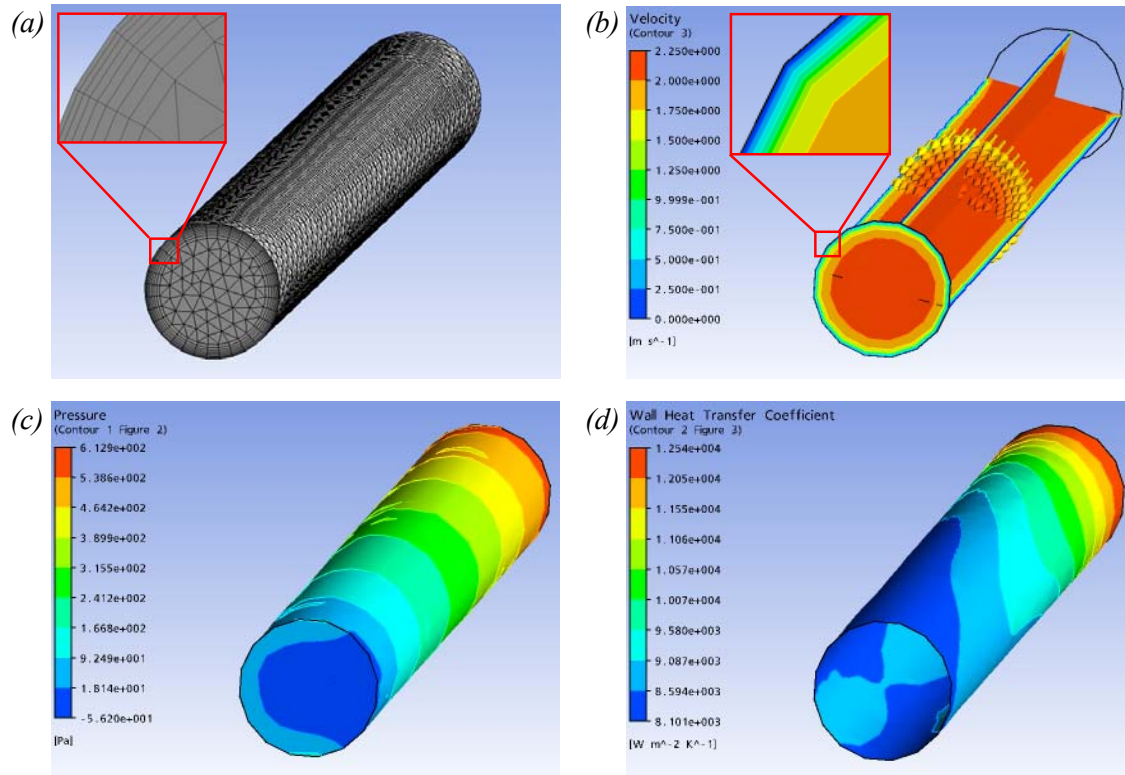


Figure 6.3: CFD analyses on a circular tube: (a) Mesh; (b) Axial velocity; (c) Pressure; (d) CHT coefficient at the wall. The results are calculated with the $k-\varepsilon$ model with 2 m/s velocity inside the channel.

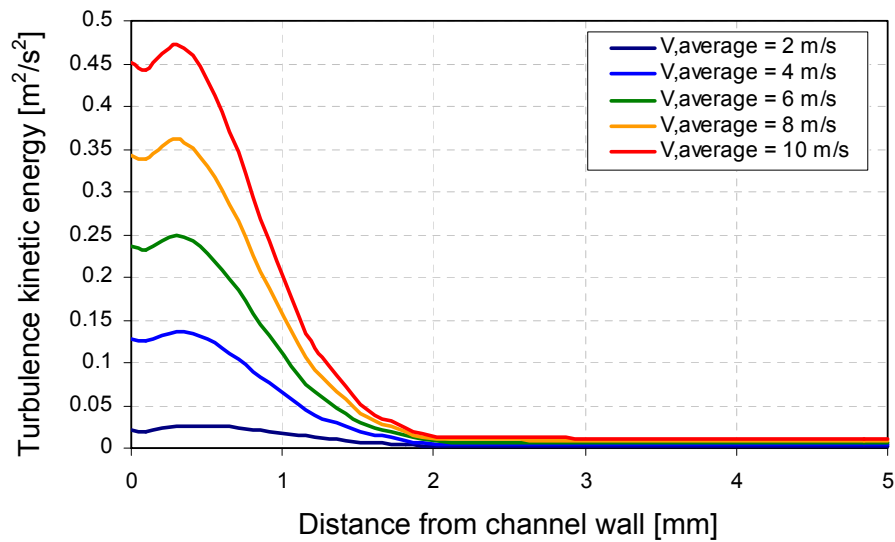


Figure 6.4: Turbulence kinetic energy along the radius.

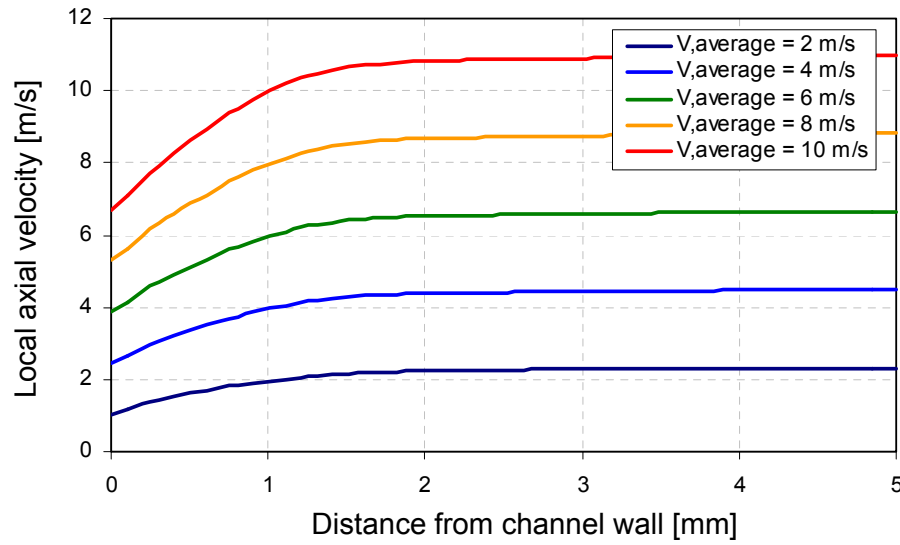


Figure 6.5: Local axial velocity along the radius.

The plot in Fig. 6.6 shows the pressure drop in function of the water velocity, calculated with the analytical formulas (Darcy-Weisbach and Colebrook-White) and with the three main turbulence models.

All the three models overestimate the pressure drop calculated with the analytical formula (which interpolates a large number of experimental data): the standard $k-\varepsilon$ has an average overestimation of 48%, the SST $k-\omega$ of 24%, the RSM of 22%. Hence, the RSM model has the best performance in estimating the pressure drop inside a tube.

A similar comparison is made for the CHT coefficients, taking the Sieder-Tate formula as benchmark (Fig. 6.7). In this case the standard $k-\varepsilon$ provides an average overestimation of 18%, the SST $k-\omega$ of 6%, while the RSM provides an average underestimation of 8%.

As an underestimation of the CHT coefficient can be considered a conservative factor in the thermo-structural analyses, also in this case the RSM method is to be considered the better one for the analyses. Hence, the RSM is chosen for the analyses on the Extraction Grid cooling system.

6.4 Description of a novel approach for the FEM analysis

A novel approach for the FEM analysis of high-performance cooling system is introduced, featuring the following three analyses made in series:

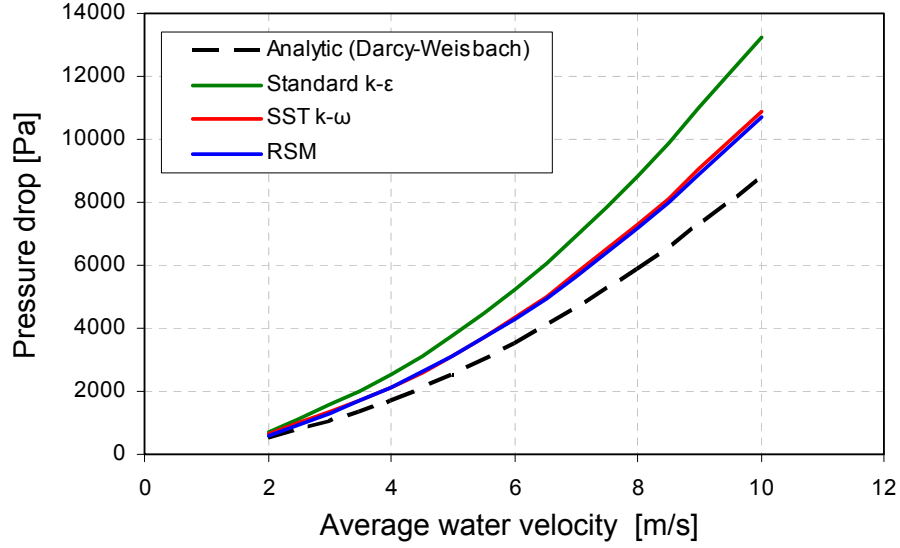


Figure 6.6: Pressure drop estimation: comparison between analytic and numerical results.

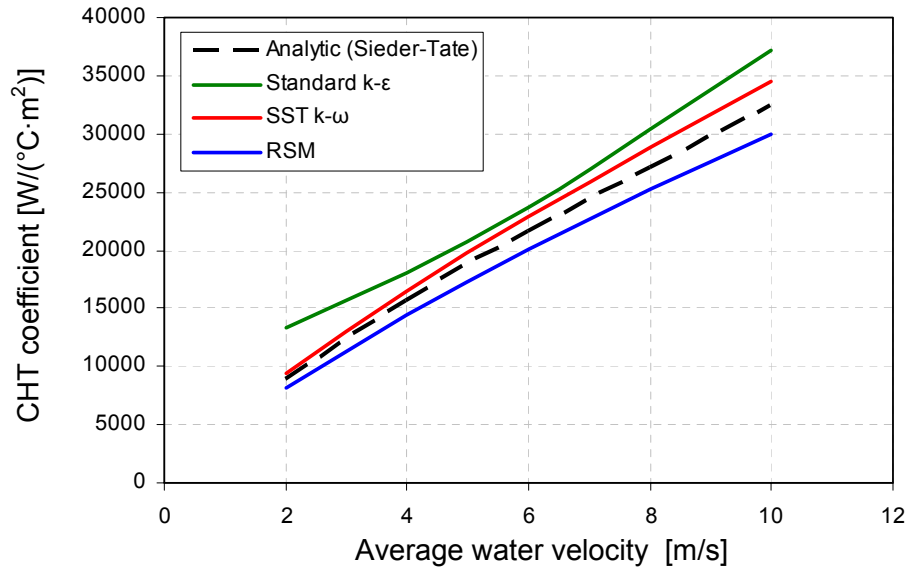


Figure 6.7: CHT coefficient estimation: comparison between analytic and numerical results.

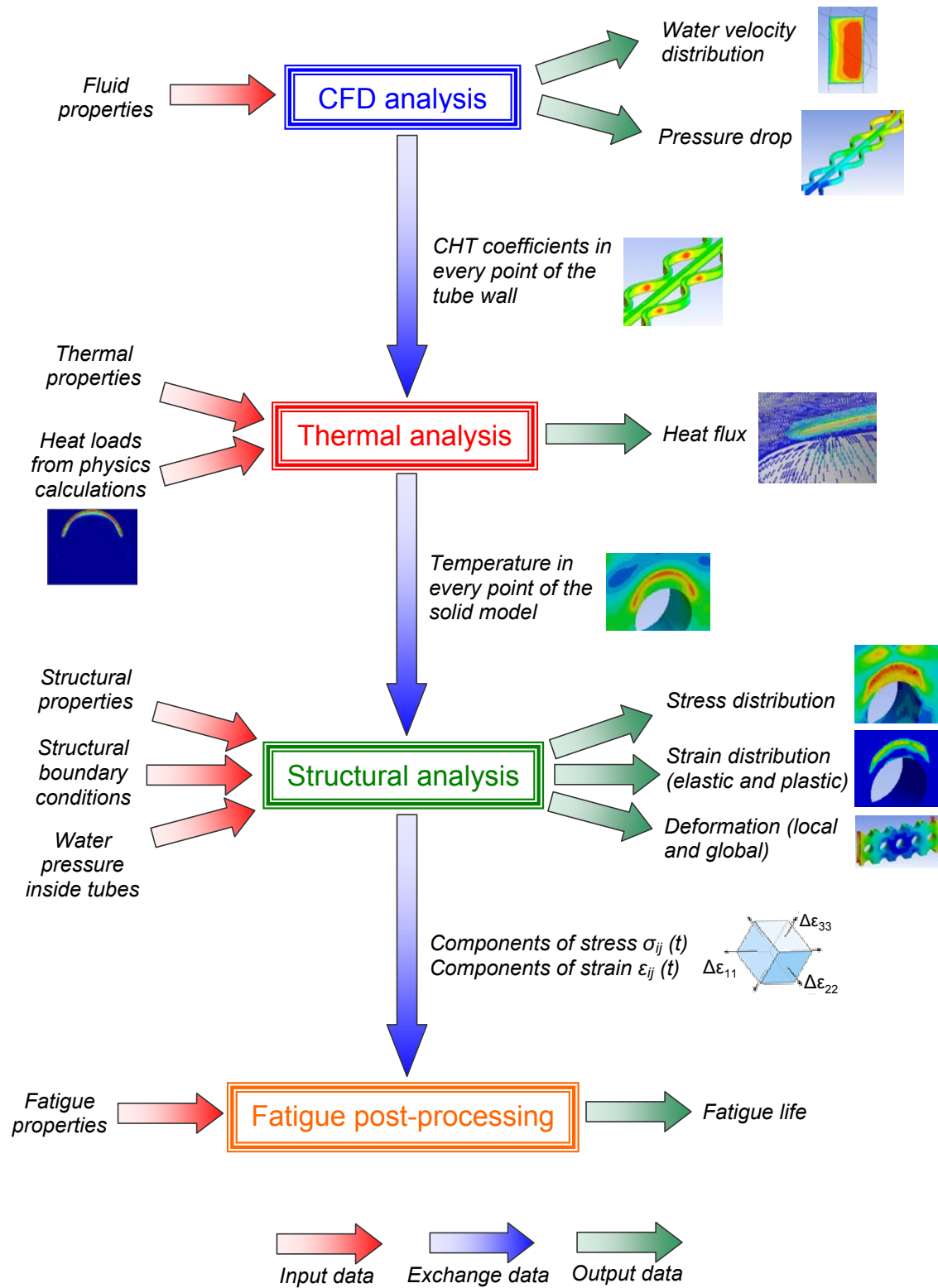


Figure 6.8: Scheme of the integrated FEM technique.

1. A Computational Fluid Dynamics (CFD) analysis which calculates the water parameters inside the tubes (velocity, pressure) and on the walls (CHT coefficient). These analysis must feature a proper turbulence model and a proper mesh (inflated near the boundary layer), as described in this chapter. This is fundamental in order to obtain meaningful results, not only by the CFD point of view but also for by the thermal and structural point of views.
2. A thermal analysis, which calculates the temperature in every point, using the output of the CFD analysis as a boundary condition. To improve the quality of the results in the case of thermal loads, this analysis should use a non-linear model for the material behavior, with the mechanical properties calculated in function of the local temperature.
3. A structural analysis, that calculates the stresses and strain in every point, using the output of the thermal analysis as a boundary condition. To improve the quality of the results in the case of thermal loads, this analysis should use a elasto-plastic model for the material behavior.

A scheme of the performed analyses is shown in Fig. 6.8, while an application is described in the next chapter.

Bibliography

- [1] D.C. Wilcox. *Multiscale model for turbulent flows*. AIAA 24th Aerospace Sciences Meeting. American Institute of Aeronautics and Astronautics (1986).
- [2] B.E. Launder and D.B. Spalding. *The numerical computation of turbulent flows*. Comp. Meth. Appl. Mech. Eng. (1974) 269-289.
- [3] H. Grotjans and F.R. Menter. *Wall functions for general application CFD codes*. Proceedings of the Fourth European Computational Fluid Dynamics Conference (1998) 1112-1117.
- [4] C.G. Speziale, S. Sarkar and T.B. Gatski. *Modelling the pressure-strain correlation of turbulence: an invariant dynamical systems approach*. J. Fluid Mechanics 277 (1991) 245-272.
- [5] F.R. Menter. *Two-equation eddy-viscosity turbulence models for engineering applications*. AIAA-Journal 32(8) (1994) 1598-1605.
- [6] J.E. Bardina, P.G. Huang and T.J. Coakley. *Turbulence Modeling Validation Testing and Development*. NASA Technical Memorandum 110446 (1997).

-
- [7] D. C. Wilcox. *Turbulence Modeling for CFD*. DCW Industries, Inc., 1994.
 - [8] B. E. Launder, G. J. Reece and W. Rodi. *Progress in the developments of a Reynolds-stress turbulence closure*. J. Fluid Mechanics 68 (1975) 537-566.

Chapter 7

Numerical methods applied to the accelerator design

SUMMARY - In many experimental campaigns – carried out in Europe, Japan and USA – the Extraction Grid (EG) was found to be one of the most critical components in the whole negative ion source. In fact, a considerable quantity of electrons is usually extracted together with the negative ions. These electrons are deflected in the extractor and mostly absorbed by the EG, causing high heat loads on the surface of the grid. If these loads exceed to tolerable values, the EG gets damaged, typically with a local melting and consequent water leakage in the vacuum environment, that is to be considered as a highly detrimental event. In this chapter, the novel technique based on integrated CFD-thermal-structural FEM analysis, introduced in the previous chapter, will be applied for the design optimization of this critical component.

7.1 Design optimization

Because of the central importance of the EG cooling system, a particular care was used to design the cooling system of this grid for the ELISE experiment, that has the main goal to reach a good extraction efficiency. According to the Pugh method [1], diverse proposals were introduced and evaluated, in an iterative process of optimization.

7.1.1 First optimization cycle

The ITER reference scheme for the EG cooling system, given by the DDD5.3 [2], is quite simple, and permits to have a good uniformity of the water flow inside the

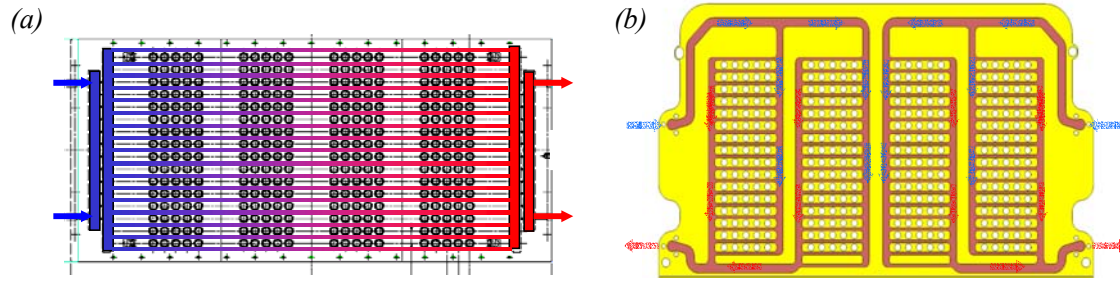


Figure 7.1: Two alternatives for the Extraction Grid cooling system: (a) reference solution (scheme A); (b) alternative solution (scheme B).

cooling channels. Nevertheless, the cooling channels are quite long and hence the total pressure drop of the circuit reaches high values, in the range of 7 bar and more. We will refer to this cooling scheme, shown in Fig. 7.1a, as to scheme A.

The scheme B, a first alternative proposed by IPP, is shown in Fig. ??b, features the vertical manifolds between the aperture groups and two separated cooling schemes.

As it is visible from Fig. 7.2, that compares the two schemes considering the same water velocity inside the cooling channels, the scheme B has some advantages:

- A 16% lower peak temperature on the grid heated surface.
- A 21% lower peak stress.
- A lower pressure drop inside the cooling channels, which are in parallel and not in series like in the previous scheme.
- A more symmetric distribution of the cooling water (with the coolest water coming from the middle of the segment).

Nevertheless, it is also affected by some problems:

- The velocity and pressure drop inside the manifolds are very high (in the range of the pressure drop and velocity inside the cooling channels). Consequently, also the total pressure drop is 15% higher.
- The water flow inside the cooling channels is highly un-uniform, with much more water flow in the channels at the edges of the grid and less in the centre.
- A 14% higher out of plane deformation.

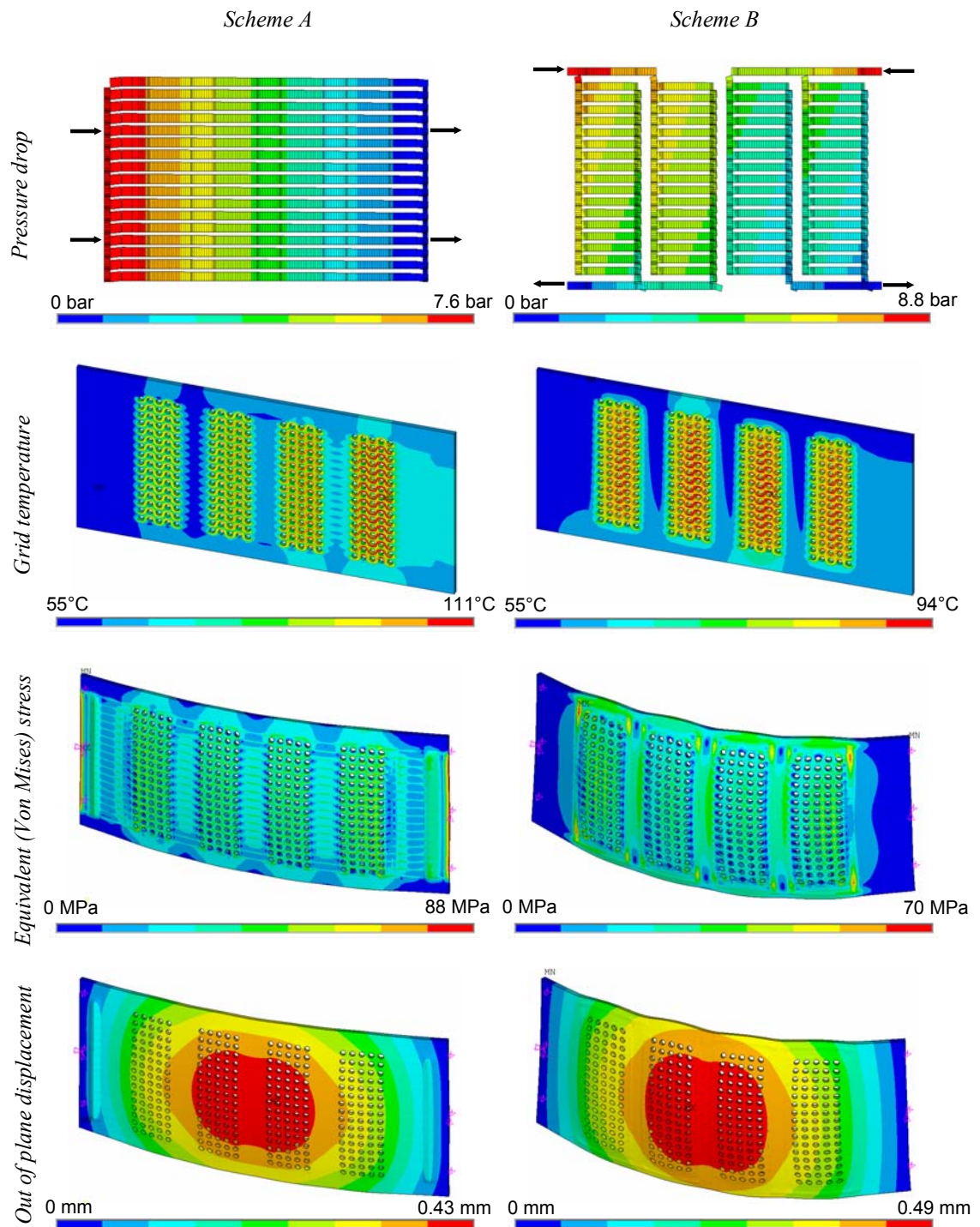


Figure 7.2: Extraction Grid: Comparison between scheme A and B.

		Concepts	
		Scheme A (datum)	Scheme B
CTQ Criteria	Pressure drop along cooling channels (lower is better)		+
	Water velocity along cooling channels (lower is better)		S
	Pressure drop along manifolds (lower is better)		-
	Water velocity along manifolds (lower is better)		-
	Temperature distribution (more uniform is better)		+
	Stress distribution (lower is better)		+
	Manufacturing (easier is better)		-
	Cost (cheaper is better)		-
	In plane deformation (lower is better)		S
	Out of plane deformation (lower is better)		-
	<i>Total positives</i>		3+
	<i>Total negatives</i>		5-

Table 7.1: Pugh matrix to compare cooling schemes A and B for the Extraction Grid.

7.1.2 Second optimization cycle

A Pugh matrix was introduced to point out and evaluate the pro and contra of the two schemes (see Tab. 7.1). In particular, this operation was done to draft a new scheme (C) that could maximize the positive aspects and minimize the negative aspects of both schemes A and B.

In particular the problems of scheme B could be solved by decreasing the pressure drop inside the manifolds and increasing the pressure drop inside the cooling channels. The first goal could be addressed by decreasing the number of vertical manifolds, with two apertures arrays in series, and by enlarging as much as possible the manifolds cross section (paying attention to the stress due to the water inside the circuit) and optimizing their shape. The second goal could be addressed by reducing the cooling channels cross section and optimizing their shape. The resulting scheme C, shown in Fig. 7.3, can be considered a mixture of schemes A and B, because it features a hybrid series/parallel cooling channels layout.

This scheme features:

1. Two separated circuits for the left and right part of every segment
2. Vertical manifolds with a decreasing cross section, to equilibrate the water flow inside the cooling channels.

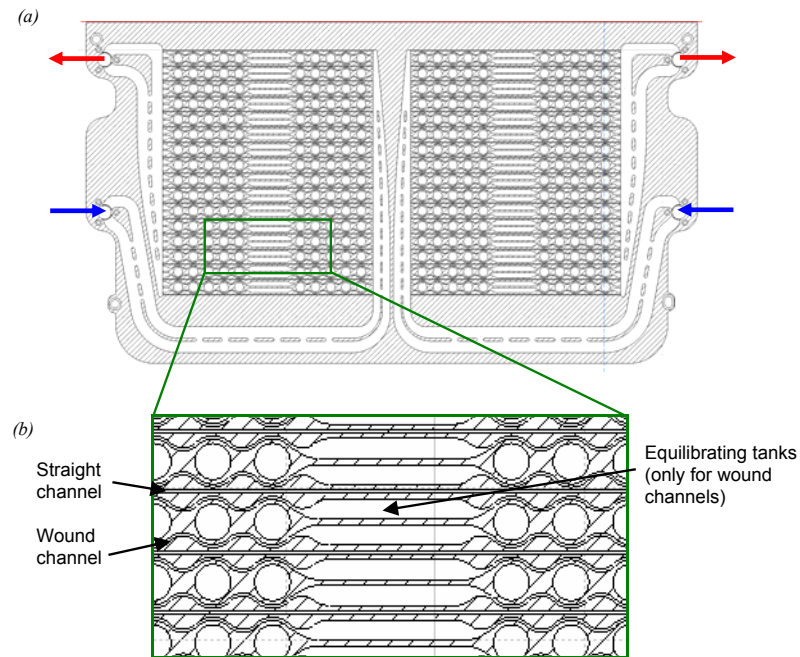


Figure 7.3: Cooling scheme C: (a) overview; (b) detail of the cooling channels.

3. Manifolds with an increased cross section, to limit the water velocity inside acceptable values.
4. 3 cooling channels between every aperture row, with a $2 \times 1 \text{ mm}^2$ cross section. While the central channel is straight, the external channels have a path with many curves, which follows the shape of the apertures. In this way, the pressure drop inside the channel (considering the same water flow) is higher, the coverage of the heated zone is better and also the interface area between copper and cooling water is increased of 37 %.
5. Equilibrating tanks between the aperture groups, in order to compensate the higher pressure drop in the wound cooling channels, compared to the one in the straight channels.

About the point 3, the limiting factor in enlarging the manifold is that the stress due to the water inside the circuit also increases with an increasing width of the manifold, and could exceed the allowable values. In fact the maximum stress is approximately proportional to the square of the channel width, as can be seen from the simplified model in Fig. 7.4.

The maximum bending moment and the maximum bending stress can be calcu-

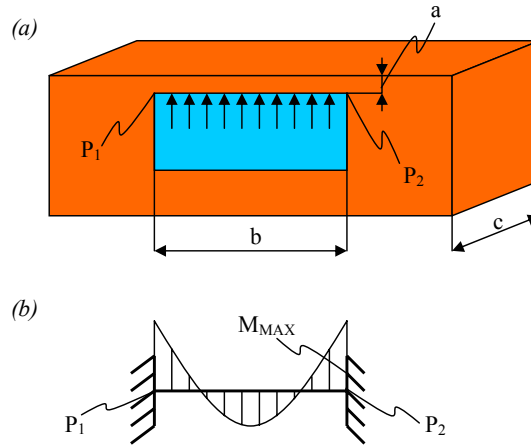


Figure 7.4: Simplified model to calculate the primary stress due to the water pressure inside a cooling channel: (a) Main dimensions and critical points; (b) Bending moment along the copper layer.

lated from the internal pressure and the geometrical parameters:

$$M_{MAX} = p \cdot c \cdot \frac{b^2}{12} \quad (7.1)$$

$$\begin{aligned} \sigma_{MAX} &= \frac{M_{MAX}}{J} \cdot \frac{a}{2} \\ &= \frac{p \cdot c \cdot \frac{b^2}{12}}{\frac{1}{12} \cdot a^3 \cdot c} \cdot \frac{a}{2} \\ &= \frac{1}{2} \cdot \frac{p \cdot b^2}{a^2} \end{aligned} \quad (7.2)$$

where p is the pressure inside the tube, a is the copper layer that covers the tube, b is the tube width, c is the tube length, M_{MAX} is the maximum momentum due to the internal pressure, σ_{MAX} is the maximum stress due to the internal pressure and J is the area moment of inertia.

The most critical points, where the stress due to the internal pressure is maximum, are P_1 and P_2 . In the case of the present cooling circuit (scheme C), in order to increase as much as possible the manifolds width, reinforcements are placed at the middle of the manifolds. With those reinforcements, the width is about half and the maximum stress is about one quarter.

About the point 4, one of the most important aspects of the cooling scheme C is that the pressure drop inside the cooling channels is not the limiting factor, as it was for scheme A. In other words, while with scheme A it was needed to enlarge the cooling channels cross section to maintain the pressure drop into acceptable values (and for this reason it was considered a $5 \times 2 \text{ mm}^2$ section, which is the maximum with the available space), for the scheme C there is no more this need, and instead a certain pressure drop inside the cooling channels is advisable in order to equilibrate

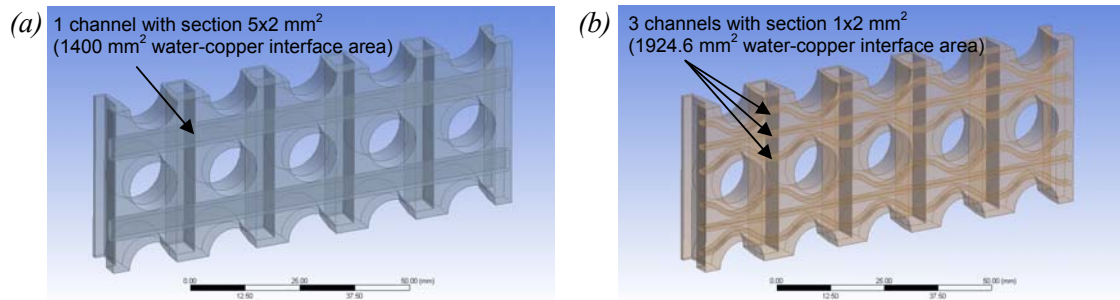


Figure 7.5: Local models of the Extraction Grid: (a) cooling scheme A; (b) cooling scheme C.

the water flow in the cooling channels (same flow in the center and at the edges of the segment).

For this reason, there is a certain freedom in the choice of the channels number and shape. In particular, the cooling channels can be optimized in order to reach all the heated zones in a better way. Then, by splitting the single channel that passes between every aperture row in the scheme A, into 3 channels with a smaller section, there is also the possibility to increase of 37% the surface able to exchange heat between water and copper, like visible in Figs. 7.5a and 7.5b.

7.2 Description of the analyses

7.2.1 Computational Fluid Dynamic analysis

The cooling schemes A and C for the Extraction Grid were compared by the point of views of:

- The Convective Heat Transfer (CHT) coefficient
- The cooling efficiency, in terms of capability to maintain the temperature and stress levels under the allowable limits
- The pressure drop inside the channels

As the cooling channels of the scheme C have a complicated shape, with many curves, the only way to properly model the water parameters inside the channels is to use Computational Fluid Dynamics. Another reason to use CFD is that with this type of analysis the water parameters are calculated in every point, which is important for a local analysis like this.

The CFD analysis is performed with the code ANSYS CFX. As the velocity

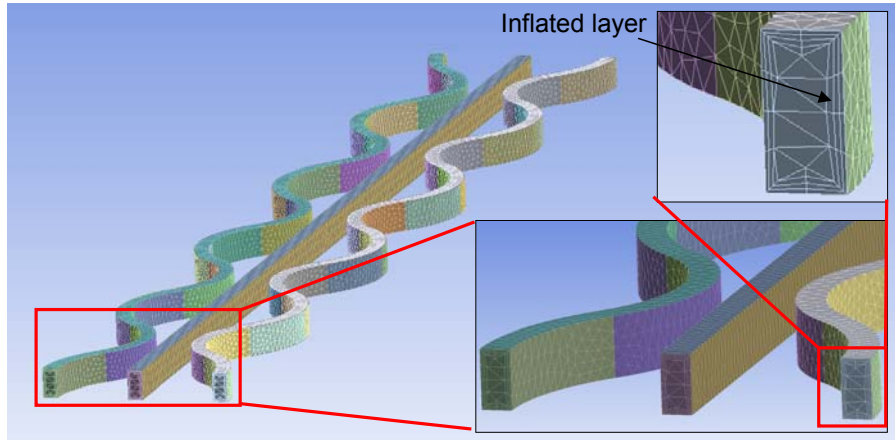


Figure 7.6: CFD mesh of the cooling channels with scheme C.

gradient is very high near the walls (in the boundary layer), the mesh of the fluid regions must be made with a particular technique. In particular, the mesh of the elements near the surface must be inflated, which means that it must be denser in the transversal direction (where the gradient is higher), as shown in Fig. 7.6.

7.2.2 Thermal analysis

The thermal analysis is a non-linear steady state analysis. The non linearity comes from the copper thermal properties, which are in function of the temperature. The Convective Heat Transfer (CHT) coefficients calculated by the CFD analysis are applied as boundary conditions to this analysis. In order to have a good approximation of the surface heat loads, the mesh is particularly dense on the surface exposed to the power. As the heat load is applied with a matrix 81x81 (0.25 mm distance between the points), also the average size of the surface elements is fixed to 0.25 mm.

7.2.3 Structural analysis

The structural analysis is a non-linear elasto-plastic analysis. The non linearity comes from the copper structural properties, which are in function of the temperature. The elasto-plastic model is the Multilinear Kinematic Hardening model (KINH), which takes into account the plastic behaviour and the work hardening of the material [3].

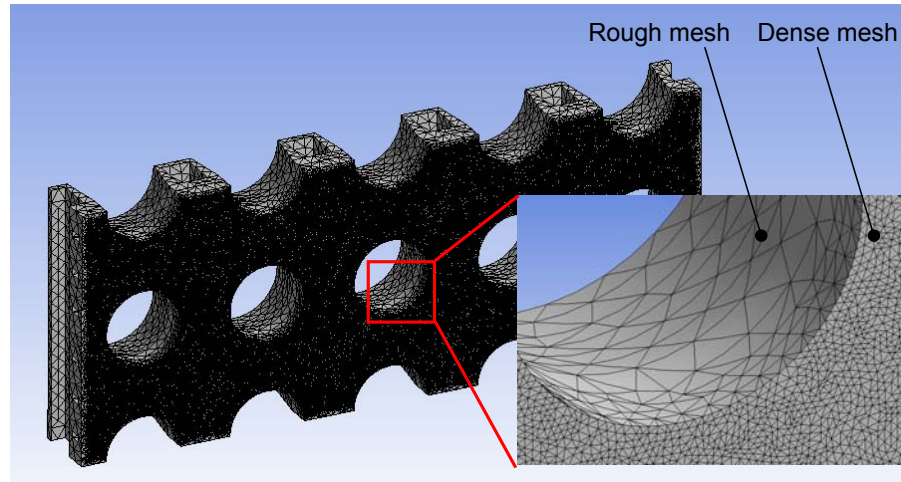


Figure 7.7: Solid mesh.

7.2.4 Material properties

7.2.4.1 Thermal and structural elastic properties

The same properties used for the global models, reported in paragraph 5.2.1, are utilized to simulate the elastic behaviour of the electrodeposited copper.

7.2.4.2 Plastic properties

The uniaxial deformation of engineering alloys subjected to cyclic loads is usually characterized by the Cyclic Stress-Strain (CSS) curve which is the locus of the tips of saturated cyclic stress-strain hysteresis loops. In both stress-controlled and strain-controlled fatigue loading, the respective strain amplitude and stress amplitude reach a stable saturation value after an initial shakedown period. This saturation state gives rise to stable hysteresis loops. Indeed, during fatigue shakedown, there is a continuous change in dislocation substructure until a stable configuration representative of the saturation state is reached. Beyond this point, the hysteresis loop remains essentially the same, cycle after cycle, over the life of the test specimen [4].

The cyclic stress-plastic strain curves for OFCH copper, taken from the ITER MPH [5], are here below reported. Fig. 7.8a shows the total strain-stress curve in a linear scale, while Fig. 7.8b shows the plastic strain-stress curves in a logarithmic scale. The data are obtained with different experimental campaign with room and high temperature in several laboratories.

The local stresses and strain ranges have to be transformed in engineering values, as the ANSYS code elaborates true stresses and strains.

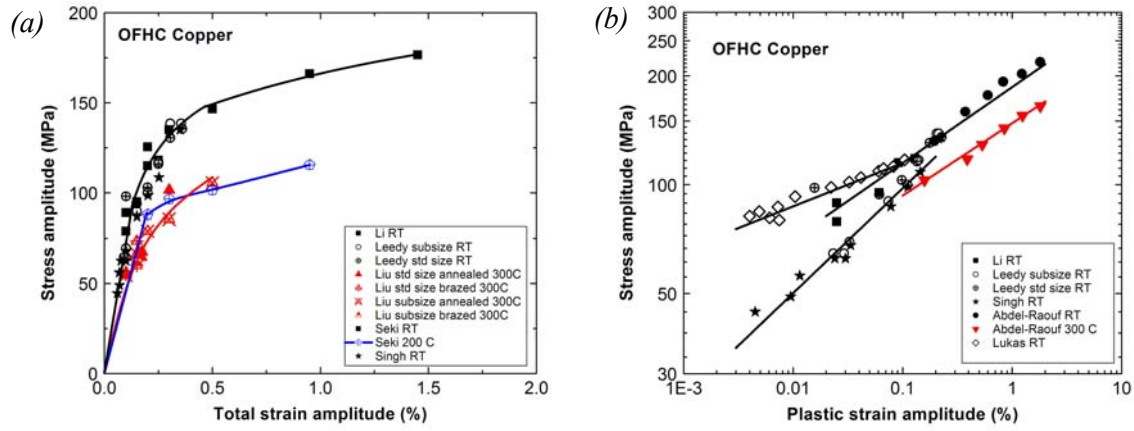


Figure 7.8: Cyclic stress-strain curves for OFHC copper: (a) Stress-total strain curves in linear scale; (b) Stress-plastic strain curves in logarithmic scale.

T = 22 °C		T = 300 °C	
σ_a [MPa]	$\epsilon_{a,pl}$ [%]	σ_a [MPa]	$\epsilon_{a,pl}$ [%]
60.5	0.00	47.3	0.00
94.8	$3.98 \cdot 10^{-2}$	77.8	$3.99 \cdot 10^{-2}$
110	$7.97 \cdot 10^{-2}$	89.4	$7.97 \cdot 10^{-2}$
118	$1.12 \cdot 10^{-1}$	95.7	$1.12 \cdot 10^{-1}$
133	$1.99 \cdot 10^{-1}$	108	$1.99 \cdot 10^{-1}$
151	$3.54 \cdot 10^{-1}$	121	$3.54 \cdot 10^{-1}$
170	$6.28 \cdot 10^{-1}$	136	$6.29 \cdot 10^{-1}$
193	1.11	153	1.11
220	1.97	174	1.97
252	3.48	198	3.48
292	6.10	228	6.11
345	10.6	268	10.6

Table 7.2: Stress-plastic strain values implemented in the FEM model.

The true values are calculated from the engineering values using the formulae:

$$\epsilon_{a,true} = \ln(1 + \epsilon_{a,eng}) \quad (7.3)$$

$$\sigma_{a,true} = \sigma \cdot (1 + \epsilon_{a,true}) \quad (7.4)$$

Fig. 7.9 shows the CSS curves with engineering and true values for the considered temperatures of 22°C and 300°C. It can be seen that for pure OFHC copper the plastic behaviour begins at 60 MPa at 22°C, and 50 MPa at 300°C.

Tab. 7.2 reports the values for the plastic strain-stress curve implemented in the FEM model.

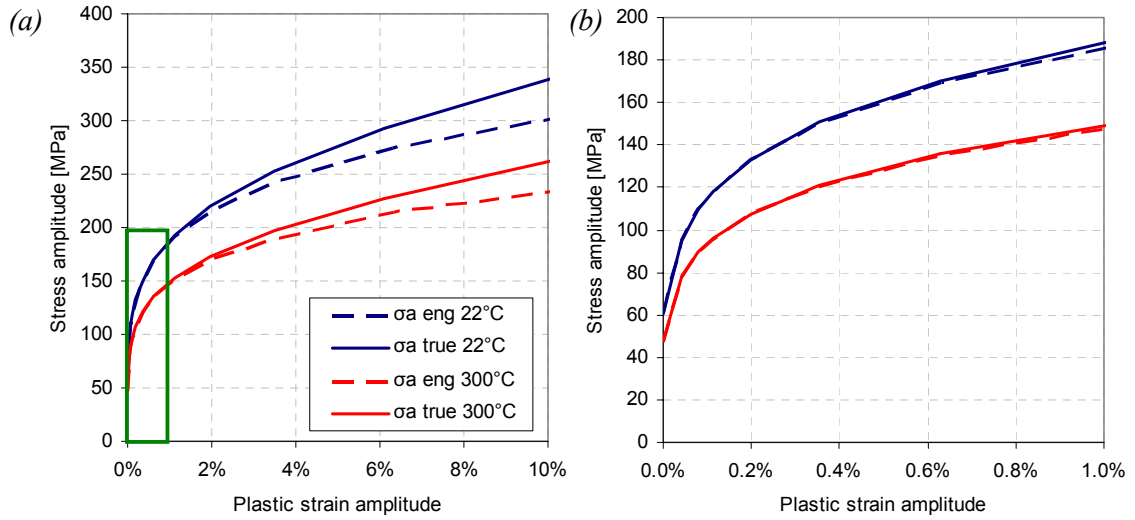


Figure 7.9: Cyclic stress-strain curves for OFCH copper, with engineering and true values: (a) Whole curve implemented in the FEM code; (b) Focus on the most important region (green rectangle in (a)).

7.2.5 Heat loads

The heating power loads on the Extraction Grid are given almost exclusively by the co-extracted electrons impinging on localized areas on the upstream surface of this grid. The following power is deposited on the EG [6]:

- 314 W/aperture to the front surface
- 59 W/aperture to the inside of the aperture
- 11 W/aperture leaks through (and is subsequently accelerated)

The power densities on the cylindrical surface (maximum value 0.8 MW/m²) are negligible by the thermo-structural point of view, as well as the ones on the downstream side of the grid. Hence, only the loads on the front surface can be considered for the thermo-mechanical analyses. The heat load from the electrons depends on many factors:

- The operating scenario (conditioning, half power, full power etc.).
- The electron-to-ion ratio. This could range in a wide domain (from 0.8 up to 10 and more) depending on the conditions of the RF chamber (walls temperature, PG temperature, plasma temperature, magnetic configuration, cesium coverage, RF power injected, gas injected)

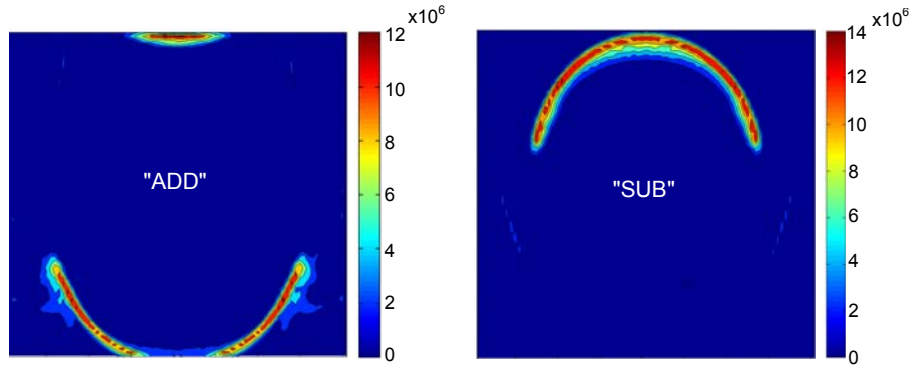


Figure 7.10: “ADD” and “SUB” heat loads (nominal power) [W/m^2].

Hence, the design must be a “robust design”, able to withstand a heat load also very different from the nominal conditions. For this reason, beside the nominal heat load calculated for the ideal conditions (shown in Fig. 7.10), also a scenario with a doubled heat load will be considered. There are two types of power densities applied on the apertures, depending on the magnetic filter field configuration and the polarities of the magnets:

- ADD configuration: added PG filter and permanent magnets magnetic field
- SUB configuration: subtracted PG filter and permanent magnets magnetic field.

Hence, the following two different scenarios are considered to compare the cooling efficiency of scheme A and C:

- **Scenario 1 (Nominal power).** The heat loads are the ones calculated by the physicists in the optimal conditions, which means for a electrons-to-ions ratio of 1, 9.6 kV extraction voltage, $26 \text{ mA}/\text{cm}^2$ ion current density and 40 mA ion current per aperture
- **Scenario 2 (Double power).** The heat loads are the double of the ones of scenario 1, in every point. This scenario should conservatively consider a deviation from the local nominal power due to one or more of these cases: bad conditioning, non-uniform cesium distribution, non-uniform PG temperature, effect of non-considered magnetic fields, operations with a power exceeding the nominal value.

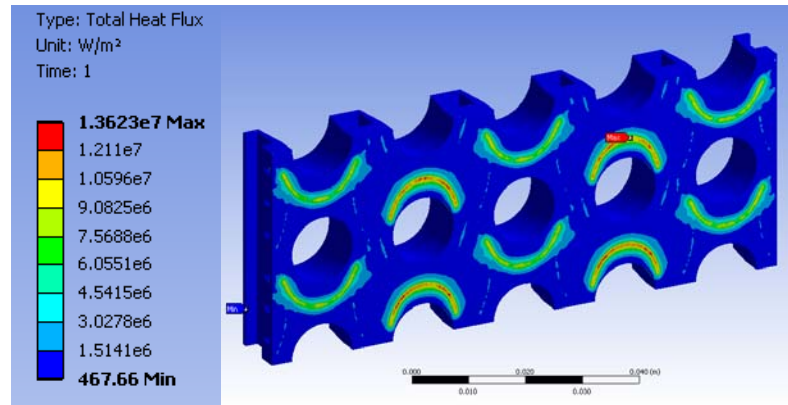


Figure 7.11: Heat load with scenario 1 (nominal power).

7.2.6 Boundary conditions

Tab. 7.3 summarizes the boundary conditions applied to the fluid dynamic, thermal and structural models.

Fluid dynamic analysis	• Average water velocity at the inlet (2 ± 16 m/s) • No slip condition for the channels walls, with rough surface ($10\ \mu\text{m}$) • Water pressure at the outlet (0 MPa)
Thermal analysis	• Convection heat transfer coefficient on the channels walls calculated by fluid dynamic analysis • Periodical boundary conditions in the vertical direction (same temperature at correspondent positions on the upper and lower border)
Structural analysis	• Temperature distribution on copper calculated by thermal analysis • Isostatic constraints

Table 7.3: Boundary conditions summary.

7.3 Fluid dynamic results

In this paragraph, the principal fluid dynamic results are calculated using the optimized Computational Fluid Dynamics model described in the previous Chapter. The two cooling schemes are compared taking for both an average velocity of 12 m/s inside the cooling tubes, value that was found to be a good choice in the global models of the Extraction Grid (see Fig. 7.2). Moreover, a comparison between the new numerical approach and the conventional approach is made.

7.3.1 Water velocity distribution and limit layer

In the case of the scheme A (see Fig. 7.12), the velocity inside the tube is not uniform, being higher in the center of the tube and null at the walls. The velocity is higher in the central point of the section (25% higher than the average value of 12 m/s). Moreover, the velocity gradient near the short sides of the tube (2 mm) is lower than the one near the long sides (5 mm). The fluid near the corners, in particular is the slowest. The velocity distribution is always axial-symmetric, as there are no curves of the channel.

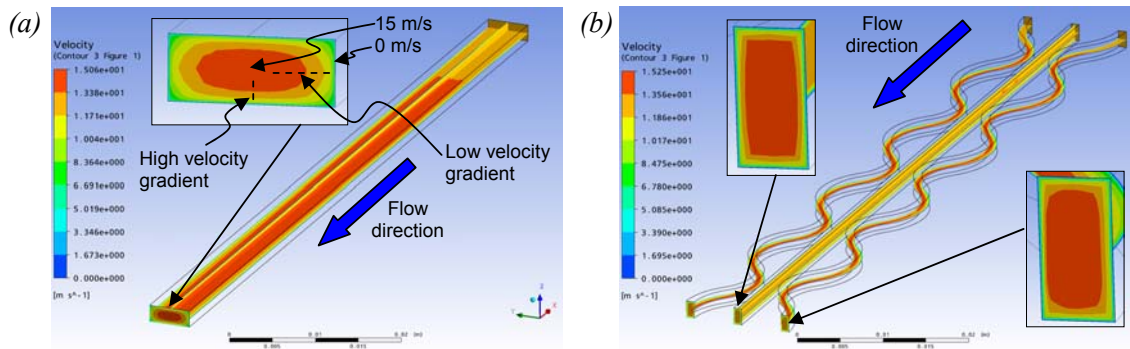


Figure 7.12: Water velocity calculation with numerical methods: (a) Scheme A; (b) Scheme C.

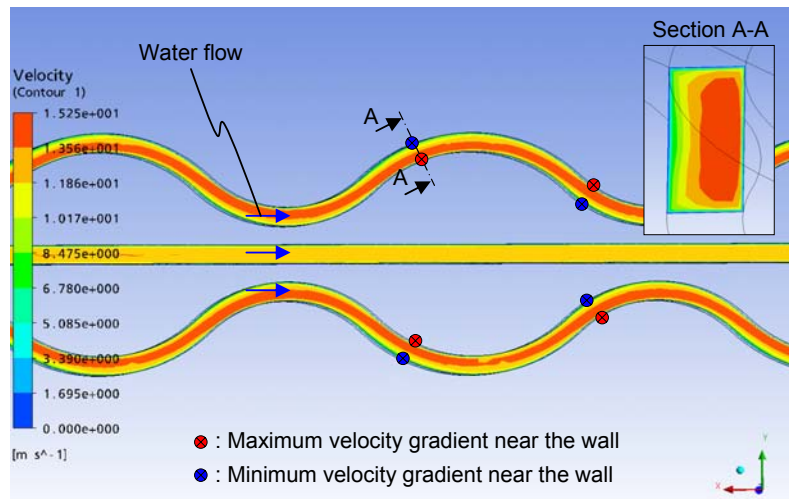


Figure 7.13: Water velocity un-homogeneity along the tubes with scheme C.

In the case of the scheme C, the water velocity has a more complex distribution. While for the central tube the situation is similar to the scheme A, with an axially

symmetrical distribution of velocity, for the curved tubes there are points where the water flow is concentrated at one side of the tubes, as shown in Fig. 7.13. As the limit layer has a different gradient of velocity in the different zones, the CHT coefficients (that are almost proportional to this gradient) are not uniform on the water-copper interface.

7.3.2 Water pressure drop calculation and comparison with analytical approach

The water pressure calculated with numerical methods (i.e. CFD) is compared with the values calculated with the analytical formulae.

For the scheme A, the CFD gives a pressure drop of about 0.076 MPa, as it is visible from Fig. 7.14a.

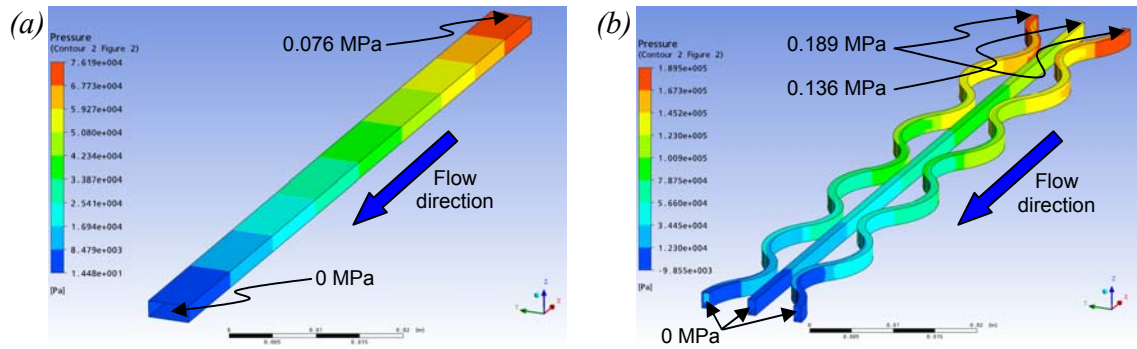


Figure 7.14: Pressure drop calculation with numerical methods: (a) Scheme A; (b) Scheme C.

The value for pressure drop calculated with CFD is in this case in very good agreement to the one calculated with the analytical method (see Tab. 7.4, scheme A).

The relations used to calculate the values in Tab. 7.4 are:

$$\nu = \frac{\mu}{\rho} \quad (7.5)$$

$$Re = \frac{v \cdot D_h}{\nu} \quad (7.6)$$

$$dp = \xi \cdot \frac{\rho v^2}{2} \cdot \frac{dL}{D_h} \quad (7.7)$$

		Scheme A	Scheme C
T	Temperature [°C]	25	25
ρ	Density [kg/m³]	996.9	996.9
μ	Dynamic viscosity [Pa·s]	$9.02 \cdot 10^{-4}$	$9.02 \cdot 10^{-4}$
ν	Cinematic viscosity [m²/s]	$9.05 \cdot 10^{-7}$	$9.05 \cdot 10^{-7}$
V	Velocity [m/s]	12	12
a	Channel width [m]	$5 \cdot 10^{-3}$	$1 \cdot 10^{-3}$
b	Channel height [m]	$2 \cdot 10^{-3}$	$2 \cdot 10^{-3}$
D_h	Hydraulic diameter [m]	$2.86 \cdot 10^{-3}$	$1.3 \cdot 10^{-3}$
ε	Absolute roughness [m]	$1 \cdot 10^{-5}$	$1 \cdot 10^{-5}$
Re	Reynolds number	37902	17688
ξ	Friction factor	0.0301	0.038
Δl	Channel lenght [m]	0.1	0.1
Δp	Pressure drop [Pa]	75826	205200

Table 7.4: Pressure drop calculation with analytical formulae.

$$\frac{1}{\sqrt{\xi}} = -2 \cdot \left[\frac{2.51}{Ra \cdot \sqrt{\xi}} + \frac{\varepsilon}{3.71 \cdot D_h} \right] \quad (7.8)$$

while the density and dynamic viscosity of water in function of the temperature can be found in many manuals.

In the case of scheme C (see Fig. 7.14b), the water pressure calculated with CFD in a 100 mm long region is about 0.136 MPa for the straight tube and 0.189 MPa for the curved tubes. This difference should be equilibrated by the larger section in the part of channels between the aperture arrays, like it is visible in Fig. 7.3. In this case the value calculated with the analytical formula (see Tab. 7.4, scheme C) is 47% higher than the one calculated with the CFD model for the central tube. This overestimation could be due to the fact that the considered analytical formulae are better estimating pressure drops when the water flow is highly turbulent, while in this case it is in the laminar/turbulent transition zone.

7.3.3 Convection heat transfer coefficient calculation and comparison with analytical approach

In the case of scheme A, the convection heat transfer (CHT) coefficient reaches its highest value in the central part of the longest side, where the gradient of velocity is the highest, as shown in Fig. 7.15a. The lowest CHT coefficient is reached near the corners of the channel, where the gradient of velocity is the lowest. In this case, the lowest CHT coefficient ($60000 \text{ W}/(^{\circ}\text{C} \cdot \text{m}^2)$) is about 30% lower than the highest one ($85000 \text{ W}/(^{\circ}\text{C} \cdot \text{m}^2)$).

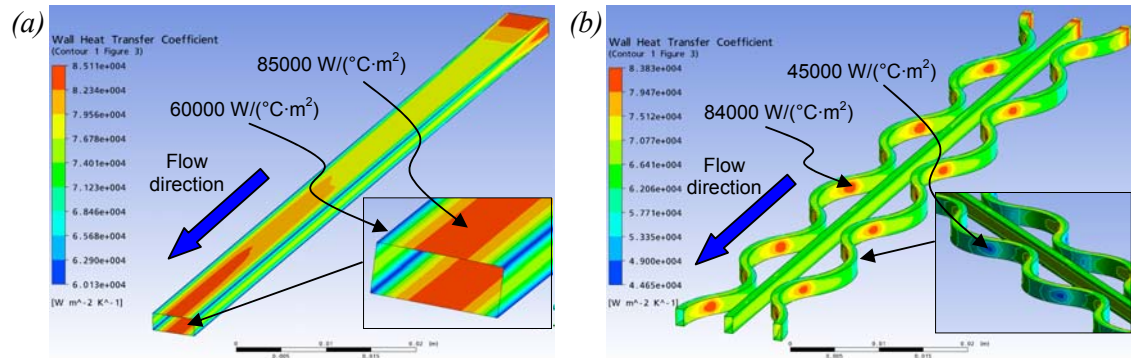


Figure 7.15: CHT coefficient calculation with numerical methods: (a) Scheme A; (b) Scheme C.

Also in this case, a comparison with the analytical calculations can be done. The Seider-Tate formula, which is at the moment the most used for analytical evaluations of the CHT coefficient in literature, will be used.

		Scheme A	Scheme C
T	Temperature [°C]	25	25
ρ	Density [kg/m³]	996.9	996.9
μ	Dynamic viscosity [Pa·s]	$9.02 \cdot 10^{-4}$	$9.02 \cdot 10^{-4}$
ν	Cinematic viscosity [m²/s]	$9.05 \cdot 10^{-7}$	$9.05 \cdot 10^{-7}$
λ	Thermal conductivity [W/(m·°C)]	0.607	0.607
C_p	Specific heat [J/(kg·°C)]	4187	4187
V	Velocity [m/s]	12	12
a	Channel width [m]	$5 \cdot 10^{-3}$	$1 \cdot 10^{-3}$
b	Channel height [m]	$2 \cdot 10^{-3}$	$2 \cdot 10^{-3}$
D_h	Hydraulic diameter [m]	$2.86 \cdot 10^{-3}$	$1.33 \cdot 10^{-3}$
Re	Reynolds number	37902	17688
Pr	Prandtl number	6.212	6.212
Nu	Nusselt number	227	123
CHT	Convection heat transfer coefficient [W/(m²·°C)]	48305	56259

Table 7.5: CHT coefficients calculation with analytical formulae.

The relations used to calculate the values in Tab. 7.5 are:

$$\nu = \frac{\mu}{\rho} \quad (7.9)$$

$$D_h = \frac{2 \cdot a \cdot b}{a + b} \quad (7.10)$$

$$Re = \frac{v \cdot D_h}{\nu} \quad (7.11)$$

$$Pr = \frac{C_p \cdot \mu}{\lambda} \quad (7.12)$$

$$Nu = 0.027 \cdot Re^{0.8} \cdot Pr^{\frac{1}{3}} \quad (7.13)$$

while the density, dynamic viscosity, thermal conductivity and specific heat of water in function of the temperature can be found in many manuals.

The value for the CHT coefficient calculated with the analytical formulae (7.5, scheme A) is about 20% lower than the average value calculated with the CFD model. In general, it seems that for rectangular channels the analytical formulae are normally more conservative than the CFD approach. In fact, as the analytical formulae were obtained from many experiments on circular section tubes, they could be even too conservative when they are applied to cross sections very different from the circular one (like in this case). This could be due to the fact that the hydraulic diameter to be considered for the analytical formulae becomes very small compared with the channel dimensions, when the cross section has a very elongated shape (with a dimension much bigger than the other).

In the case of scheme C, the convection heat transfer (CHT) coefficient reaches its highest value in the point where the water flow impinges on the channel wall, as visible from Fig. 7.15b; also in this case, the maximum is reached where the gradient of velocity is the highest. The lowest CHT coefficient is at the opposite side of the channel, where the gradient of velocity is the lowest. In this case, the lowest CHT coefficient is about 50% lower than the highest one. On the side of the channel near the surface exposed to the heat load, the CHT coefficient is almost uniform, around $65000 \text{ W}/(\text{m}^2 \cdot ^\circ\text{C})$. The value for the CHT coefficient calculated with the analytical formulae (7.5, scheme C) is in moderate agreement with the average value calculated with the CFD model, being 12% lower.

7.4 Thermal and structural results

In this paragraph the thermal and structural results with the two cooling schemes are compared. The fixed boundary conditions are the water velocity, fixed to 12 m/s, and the power load scenario 2 (double power). The water velocity of 12 m/s is chosen because this value was found to be a good choice also for the alignment analyses.

The double power load scenario is chosen because it is the most conservative case considered for the operations. Nevertheless, a sensitivity analysis with different values of the water velocity and power load will be addressed in the paragraph 5.

For an easier comparison, the scales of the figure are the same in each of the next paragraphs.

The maximum temperature is 16% lower with cooling scheme C.

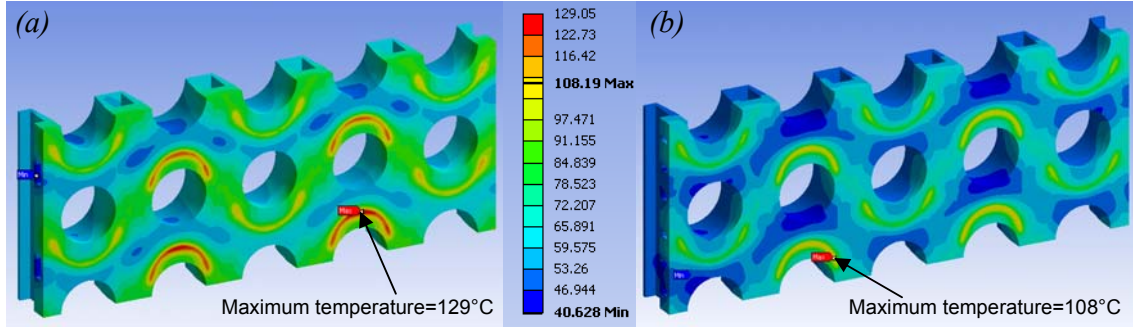


Figure 7.16: Temperature contour plot with 12 m/s water velocity and double power.

The maximum equivalent (Von Mises) stress is 5% lower with cooling scheme C.

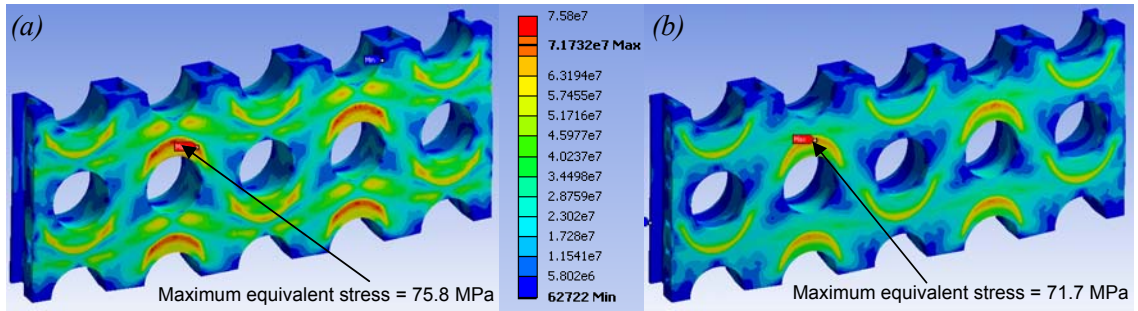


Figure 7.17: Equivalent (Von Mises) stress contour plot with 12 m/s water velocity and double power.

The maximum equivalent (Von Mises) strain is 6% lower with cooling scheme C.

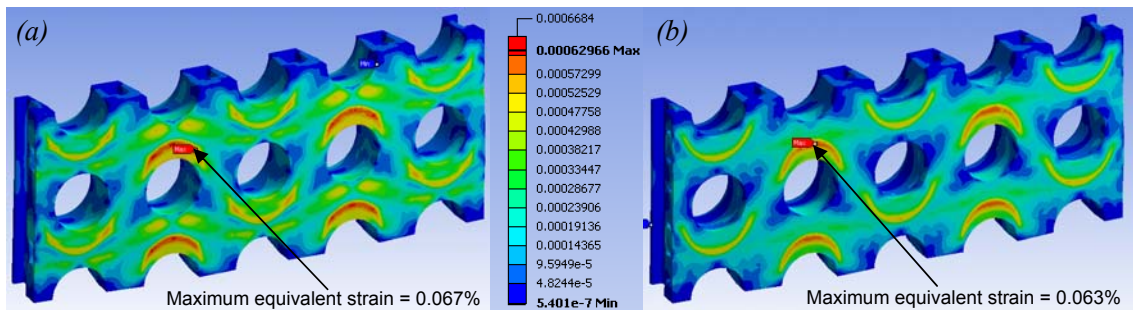


Figure 7.18: Equivalent (Von Mises) elastic strain contour plot with 12 m/s water velocity and double power.

The maximum equivalent plastic strain is 22% lower with cooling scheme C.

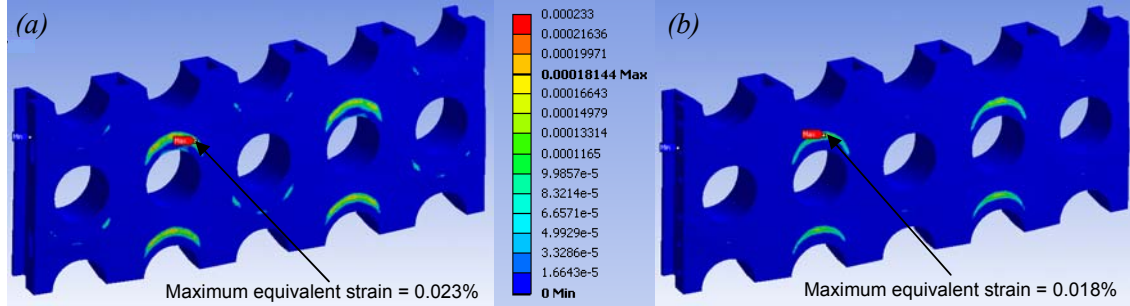


Figure 7.19: Equivalent plastic strain contour plot with 12 m/s water velocity and double power.

The maximum out of plane displacement is 30% lower with cooling scheme C.

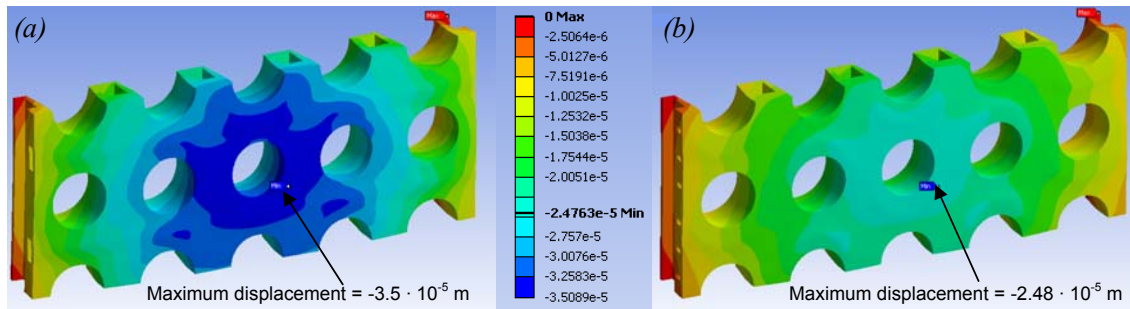


Figure 7.20: Local out of plane displacement contour plot with 12 m/s water velocity and double power.

7.5 Sensitivity analyses

In this paragraph, the main result with two cooling schemes (A and C) are compared, considering the two power load scenario (nominal and double) and the water velocity ranging from 2 to 16 m/s.

The global out of plane displacement is obtained by multiplying the local out of plane displacement times a correlation factor. This factor was evaluated by comparing a local and a global analysis performed with corresponding boundary conditions (see Fig. 7.21).

Two hypotheses are assumed to make this correlation:

1. The out of plane deformation is generated only in the apertures group areas, where there is a gradient between upstream and downstream sides of the grid.

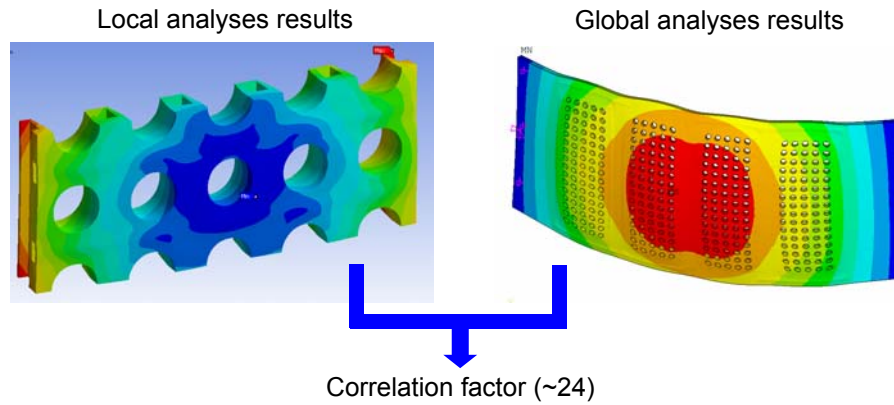


Figure 7.21: Evaluation of a correlation factor between local and global displacement.

2. A negligible influence is given by the cooling manifolds on the global out of plane deformation

7.6 Surface cooling efficiency evaluation

The models used to calculate the heat loads amount and distributions rely on assumptions that approximate the reality with some uncertainties. For this reason, the actual heat loads could be different from the calculated ones. In particular, the shape and position of the most heated areas, that are foreseen to be a kind of half moons around the apertures, could be different from the foreseen ones.

In order to obtain a design flexible to possible variations on the heat load distribution, the surface cooling efficiency should be sufficiently good not only on the areas that are foreseen to be the most heated, but also on the rest of the surface among the apertures.

To compare the two cooling schemes also by this point view, a thermal analysis has been done, where a uniform heat load has been applied on the heated surface. The total heat flux is here the same of the nominal power scenario, but with a uniform distribution.

We can estimate a surface cooling efficiency index (ψ) by dividing the temperature increase between surface temperature and water temperature to the water temperature.

In the case of the cooling scheme A (shown in Fig. 7.24a), we obtain for this

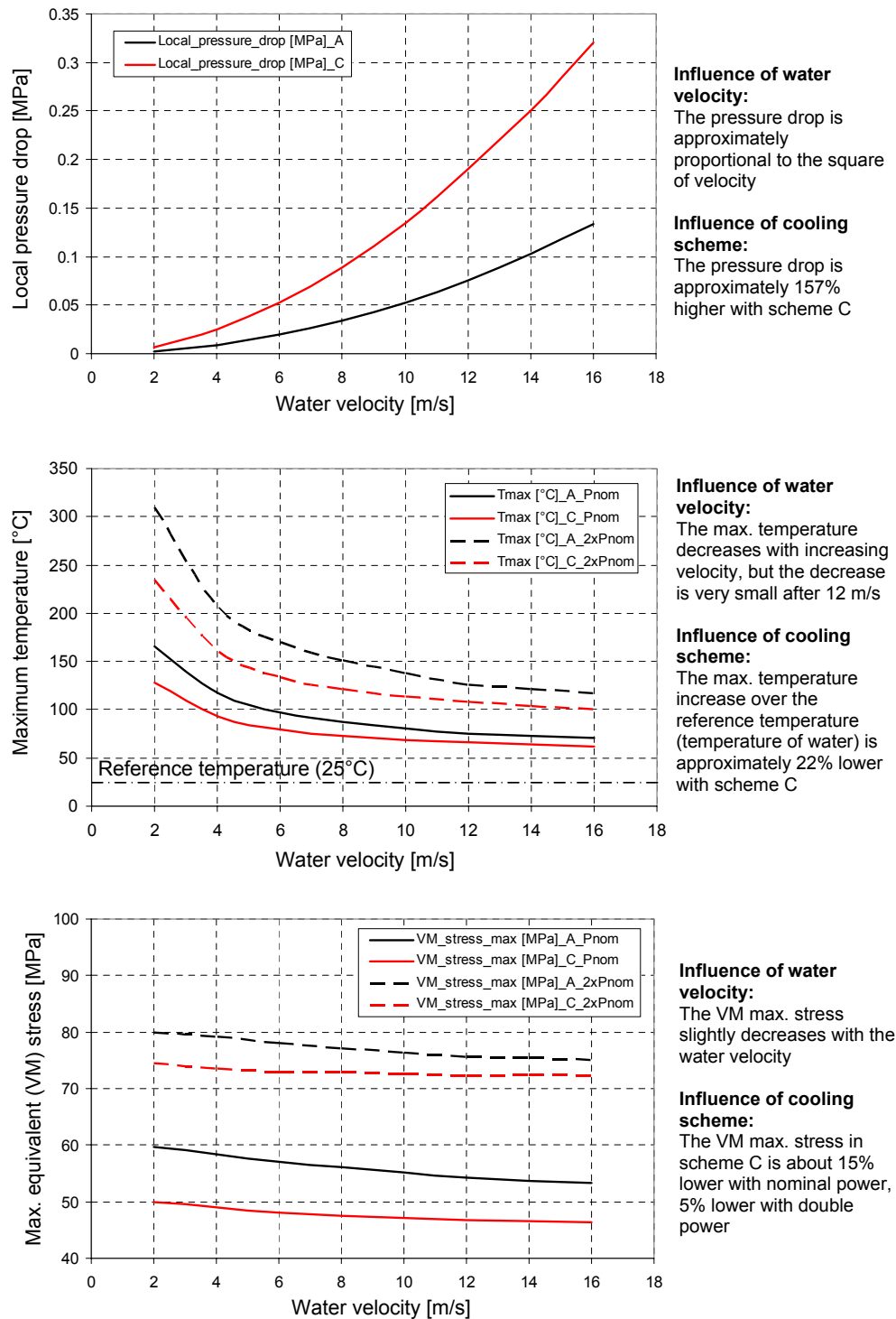


Figure 7.22: Sensitivity analysis of pressure drop, temperature and equivalent stress to water velocity and cooling scheme.

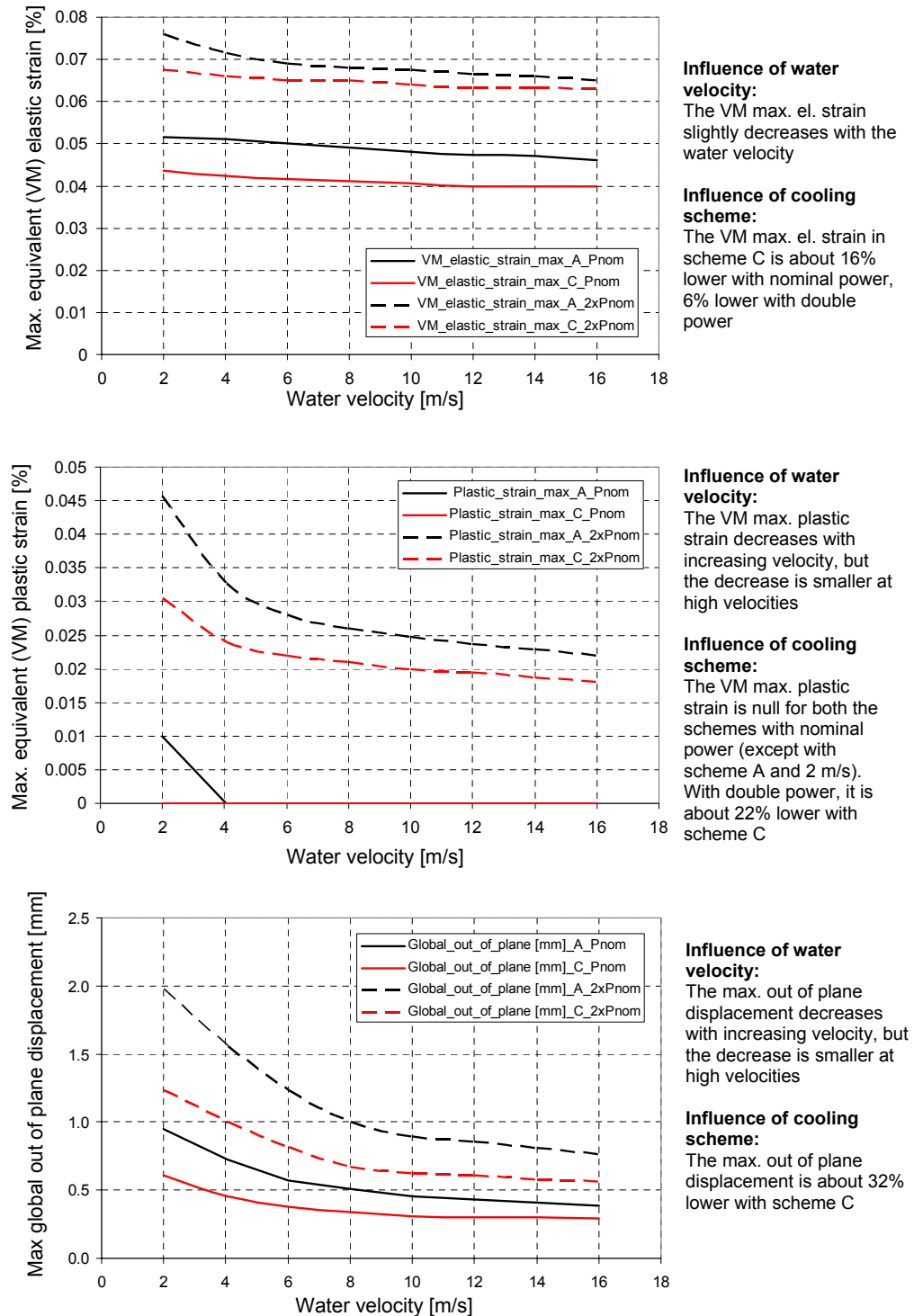


Figure 7.23: Sensitivity analysis of elastic strain, plastic strain and out of plane displacement to water velocity and cooling scheme.

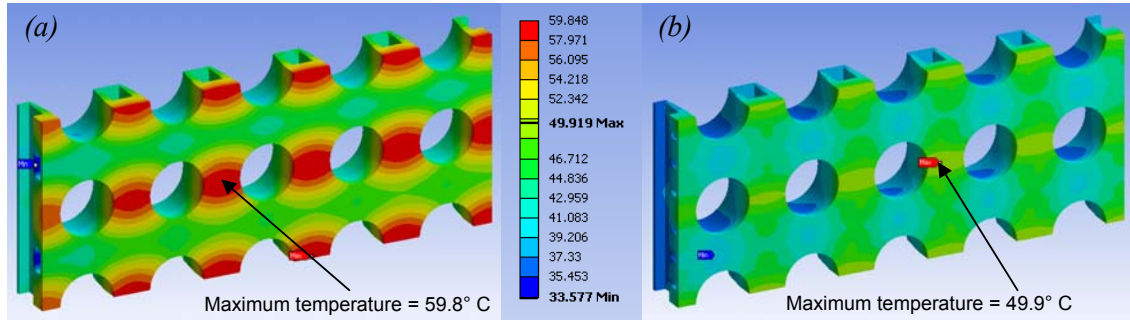


Figure 7.24: Temperature distribution with uniform load: (a) Scheme A; (b) Scheme C.

index the two extreme values:

$$\begin{aligned}
 \psi_{A,MAX} &= \frac{T_{MAX,surf} - T_{bulk,water}}{T_{bulk,water}} \\
 &= \frac{59.8 - 25}{25} \\
 &= 1.39
 \end{aligned} \tag{7.14}$$

$$\begin{aligned}
 \psi_{A,min} &= \frac{T_{min,surf} - T_{bulk,water}}{T_{bulk,water}} \\
 &= \frac{42.2 - 25}{25} \\
 &= 0.69
 \end{aligned} \tag{7.15}$$

A surface cooling uniformity index (μ) can be calculated by the ratio between the minimum and maximum cooling efficiency index:

$$\begin{aligned}
 \mu_A &= \frac{\psi_{A,min}}{\psi_{A,MAX}} \\
 &= \frac{0.69}{1.39} \\
 &= 0.49
 \end{aligned} \tag{7.16}$$

Hence, we can say that the cooling efficiency in the most critical points (red areas in Fig. 7.24a) is about a half of the cooling efficiency in the less critical points. A value of the unity for the uniformity index means a perfect uniformity of the cooling effect on the surface.

In the case of scheme C (shown in Fig. 7.24b) the values of the cooling efficiency index are:

$$\psi_{C,MAX} = 0.99 \tag{7.17}$$

$$\psi_{C,min} = 0.60 \tag{7.18}$$

And the uniformity index is:

$$\begin{aligned}
 \mu_A &= \frac{\psi_{A,min}}{\psi_{A,MAX}} \\
 &= \frac{0.60}{0.99} \\
 &= 0.61
 \end{aligned} \tag{7.19}$$

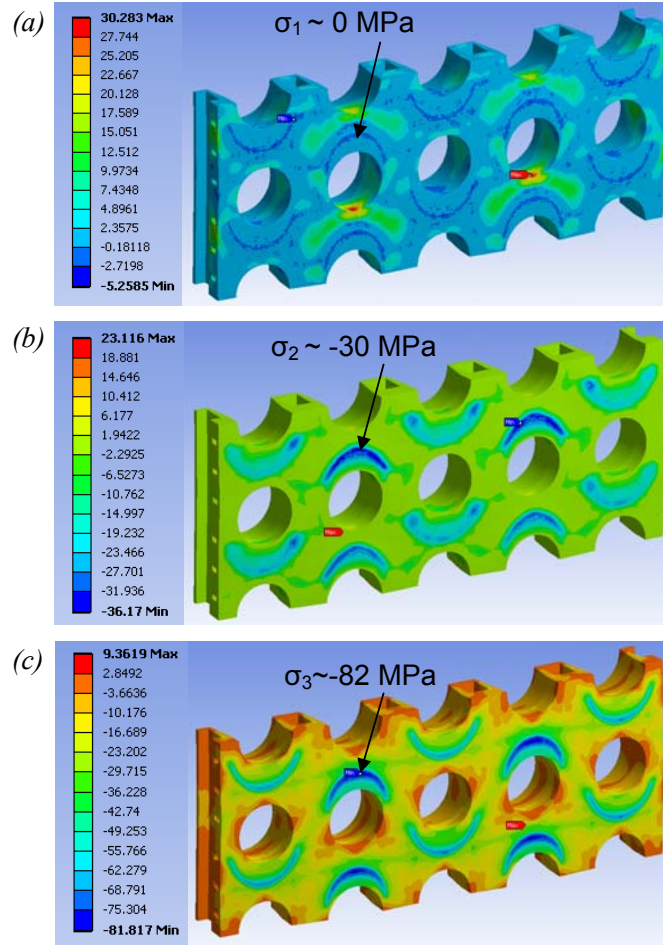


Figure 7.25: Principal stress contour plot: (a) Maximum principal stress; (b) Middle principal stress; (c) Minimum principal stress. The labels are referred to the most critical point for fatigue.

As the uniformity index is 24% higher with scheme C, this scheme can be considered better for the cooling uniformity.

7.7 Fatigue life evaluation

From Fig. 7.25, it can be observed that the strain cycles in the most critical point are almost completely monotonic compressive. We will consider the critical point for fatigue life, the one that has the higher value for the total equivalent strain. This point is in the middle of the most heated area. The values of the three principal stresses on the critical point are: $\sigma_1 \sim 0$ MPa, $\sigma_2 \sim -30$ MPa, $\sigma_3 \sim -82$ MPa. So

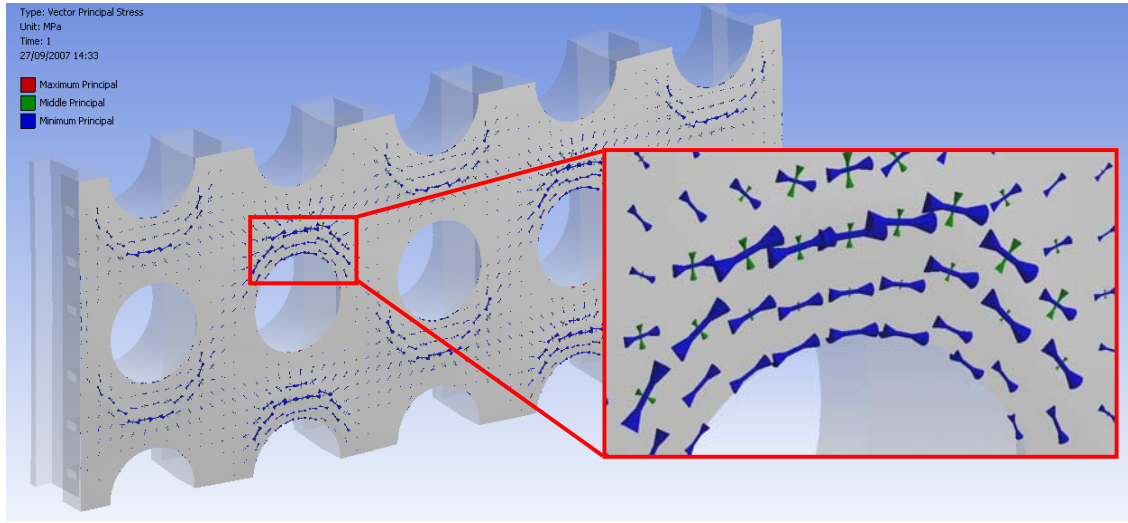


Figure 7.26: Vector plot of principal stresses.

the stress in the critical point is nearly of pure compression. This is confirmed by the vector plot of principal stresses (Fig. 7.26), from which it is clear that the third principal stress (of pure compression on the surface along the tangential direction) prevails over the first (null, normal to the surface) and second (of pure compression on the surface, along the radial direction). The Mohr's circles stress diagram for the most critical point is shown in Fig. 7.27.

About the effect of the mean strain, the following considerations taken from section C3320.5 [7] Appendix C can be applied in the present case: “For low, steady primary (load-controlled, e.g., pressure, self-weight, etc.) stresses and larger cyclic thermal stresses, the induced mean stresses can be relaxed by cyclic plasticity and it is more realistic to correct for mean strain rather than mean stress. The abundant literature on this subject concludes that the effect of mean strain on endurance is negligible as long as it is small compared to the ductility of the material. For a highly ductile material [...] the mean strain is always negligible compared to the ductility and does not need to be taken into account in defining the permissible limits.”

On the extraction grid, the cyclic stress is due to thermal loading and the material is very ductile, so the effects of mean stress and corresponding mean strain are not taken into account, and the damage given by the monotonic compressive strain is considered equal to the one given by fully reversed strain (with the same total strain range).

Hence, a simplified evaluation of the fatigue life, based on the total equivalent (Von Mises) strain and fully reversed fatigue curves is considered in this paragraph to estimate the difference between the schemes A and C.

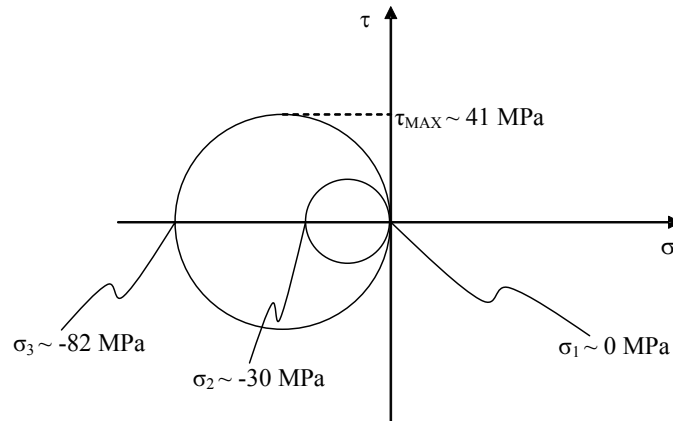


Figure 7.27: Mohr's circles in correspondence of the maximum Von Mises stress.

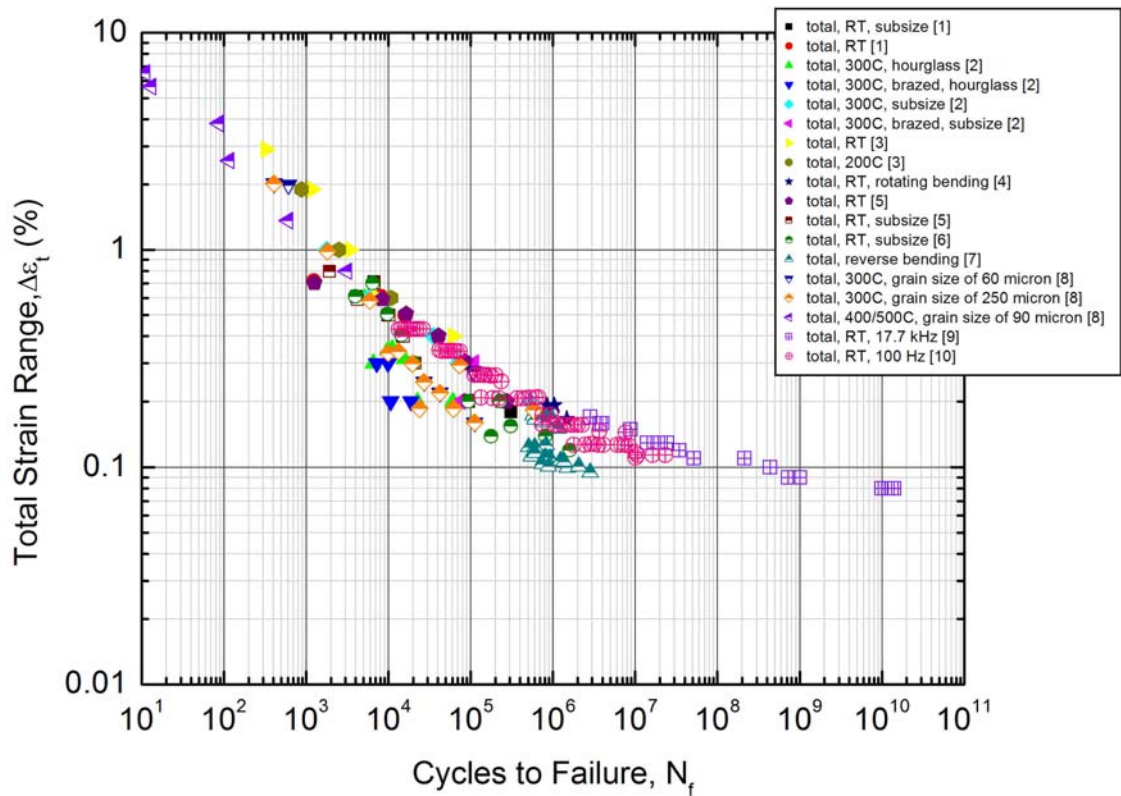


Figure 7.28: Fatigue data for unirradiated pure annealed copper.

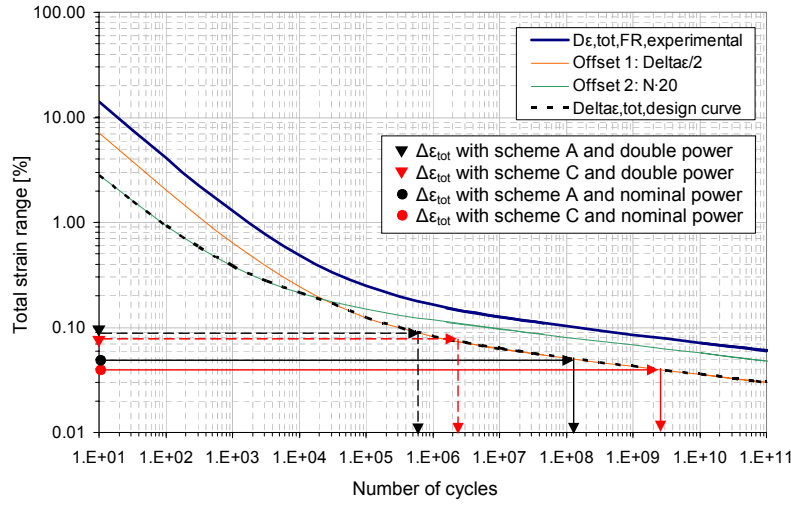


Figure 7.29: Estimation of the Extraction Grid fatigue life, with 12 m/s water velocity in the cooling channels, with the two cooling schemes and two power scenarios

Fig. 7.28 summarizes the available fatigue data for unirradiated copper from the ITER database [5]. Most of the data were obtained from the fully-reversed ($R = -1$) strain-controlled fatigue tests, and the cycles to failure are presented as a function of the total strain range.

The design fatigue curve for unirradiated pure annealed copper was deduced by offsetting of the fitted strain-life curve by a factor of 20 in number of cycles to failure or by a factor of 2 in total strain range, whichever is the more conservative (following the rules in section A.GEN.5.5 Appendix A [5]).

The curve equation fitting the experimental data of Fig. 7.28 is:

$$\Delta\varepsilon = 49.89 \cdot N^{-0.57} + 0.40 \cdot N^{-0.075} \quad (7.20)$$

as reported in [5]. Hence the fatigue design curve equation is:

$$\Delta\varepsilon_{design} = \min \left[\frac{49.89 \cdot N^{-0.57} + 0.40 \cdot N^{-0.075}}{2}; 49.89 \cdot (N \cdot 20)^{-0.57} + 0.40 \cdot (N \cdot 20)^{-0.075} \right] \quad (7.21)$$

Fig. represents the fitting curve, the two offset curves and the fatigue design curve. As it is visible in Tab. 7.6, the fatigue life foreseen with scheme C is 20 times longer in the case of nominal power beam on/off cycles, 4 times longer in the case of double power beam on/off cycles.

	Scheme A	Scheme C	Ratio
Nominal power scenario	$1 \cdot 10^8$	$2 \cdot 10^9$	20
Double power scenario	$5 \cdot 10^5$	$2 \cdot 10^6$	4

Table 7.6: Fatigue evaluations summary, with 12 m/s water velocity.

7.8 Discussion

The Pugh matrix that compares the cooling schemes A and C is reported in table 7.7. It can be noted that the number of plus and minus is the same. Nevertheless, some points are to be considered more important than others. In particular, the pressure drop on manifolds is found to be a critical point in the design of the ITER NBI, and so an optimized version of the cooling scheme A is chosen for this facility. On the other side, the boundary conditions of ELISE are different, and an optimized version of the cooling scheme C is chosen for this facility.

		Concepts	
		Scheme A (datum)	Scheme C
CTQ Criteria	Pressure drop along cooling channels (lower is better)		+
	Water velocity along cooling channels (lower is better)		S
	Pressure drop along manifolds (lower is better)		–
	Water velocity along manifolds (lower is better)		–
	Temperature distribution (more uniform is better)		+
	Stress distribution (lower is better)		+
	Manufacturing (easier is better)		–
	Cost (cheaper is better)		–
	In plane deformation (lower is better)		S
	Out of plane deformation (lower is better)		+
	<i>Total positives</i>		<i>4+</i>
	<i>Total negatives</i>		<i>4–</i>

Table 7.7: Pugh matrix to compare scheme A and C.

Bibliography

- [1] D. Ginn and E. Varner. *The Design for Six Sigma Memory Jogger: Tools and*

- Methods for Robust Processes and Products*. GOAL/QPC 2004.
- [2] Various Authors. *Design Description Document (DDD 5.3), Neutral Beam Heating and Current Drive (NBH&CD) System*. ITER Document No. 53 DDD 29 01-07-03 R 0.1.
- [3] Various Authors. *Ansys 11.0 guide*. Ansys Inc.
- [4] M. Dalla Palma. *Multiaxial Fatigue Design Criteria for High Heat Flux Components Actively Cooled in Nucleate Boiling Conditions*. Ph.D. Thesis, Università degli Studi di Padova, Consorzio RFX - Associazione Euratom-ENEA sulla Fusione, 2006.
- [5] Various Authors. *ITER MPH. Material Properties Handbook*. ITER Document No. G 74 MA 16.
- [6] H. P. L. de Esch. *TW4-THHN-IITF2 Final Report- Deliverable 3.1.4.4 Electrons in SINGAP*. EFDA report 2006.
- [7] Various Authors. *ITER SDC-IC, Structural Design Criteria for ITER In-Vessel Components*. July 2004.

Chapter 8

Methods for the analysis and verification of vacuum vessels

SUMMARY - The pressure vessels are closed, rigid containers designed to hold gases or liquids at a pressure different from the ambient pressure. They have many applications, like the storage of gases, distillation towers, nuclear reactors, autoclaves, petrochemical plants etc. A vessel that is inadequately designed to handle a high pressure constitutes a very significant safety hazard, because of the great amount of potential energy that can be stored in the structure. This has caused many explosions in the early times of pressure vessels, usually with great damages and sometimes also lost of human lives. For this reason, a particular care must be taken in the design of a pressure vessel, and norms dedicated to the pressure vessel design have been developed. In the case of vacuum vessel, where the pressure inside the vessel is lower than the atmospheric one, a failure event could cause an implosion, which would be a highly detrimental event for the experiment itself. For this reason, also in the case of a vacuum vessel, proper norms must be utilized for the design, as exemplified in this chapter for the vacuum vessel of the ELISE experiment at IPP.

8.1 Design criteria

8.1.1 Generalities

Pressure vessel design has been historically based on Design-by-Formula (DBF) route: standard vessel configurations were sized using a series of simple formulae and charts.

In addition to the DBF route, many national codes and standards for pressure

vessel and boiler design do provide rules for a Design-by-Analysis (DBA) route, where the admissibility of a design is checked, or proven, via a detailed investigation of the structure's behaviour under the expected loads to be considered. Considering the nature of the geometry and the peculiarity of the design loads, the DBA route is more appropriate for fusion experimental devices.

The DBA procedure is intended to guard against possible pressure vessel failure modes by performing a detailed stress analysis of the vessel. The failure modes to be considered are:

1. Excessive elastic deformation affecting functional adequacy.
2. Excessive plastic deformation (immediate plastic collapse).
3. Progressive deformation (ratcheting)
4. Fatigue and crack propagation
5. Structural instability (buckling)

Other failure modes (as stress corrosion and creep) and their interaction effects are normally not a concern for the vacuum vessels. The ASME Code are nowadays the most regarded and used norms in this field. Also the document "ITER Structural Design Criteria - General Section" [1] suggests to adopt these design criteria, in particular:

- **ASME Section VIII Division 2** [2]: Always, except for: Category III and IV events, buckling verifications
- **ASME Section III-NC** [3]: for Category III and IV events, and for buckling verifications

The categories of events in ITER are ordered from I to IV according to their probability to happen: operational loading (I), likely loading (II), unlikely loading (III), extremely unlikely loading (IV).

The ASME Code offers two routes to design by analysis, the elastic route and the inelastic route. The elastic route is suggested to perform static verifications, because it is more consolidated and conservative.

The two main critical phases, that arise following the elastic route, are stress linearization and categorization. The linearization procedure is required to decompose the calculated stresses into membrane, bending and non-linear, the categorization to distinguish primary from secondary stresses. In this processes various difficulties arise especially where there are discontinuities or complicated geometries. Interpretations of the elastic analysis results and comparisons with allowable values may be

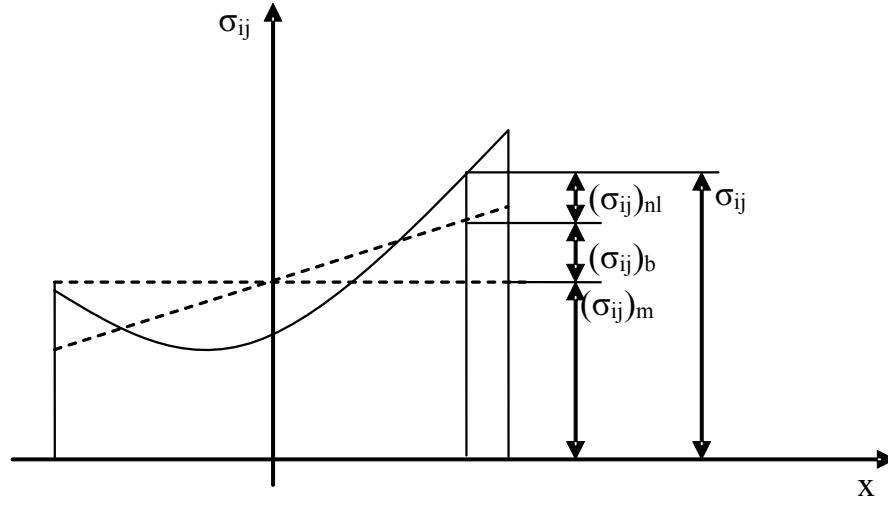


Figure 8.1: Stress linearization.

problematical and, to a large extent, depends on the analyst's judgement.

8.1.2 Stress linearization

With regard to linearization, the stress components σ_{ij} in any point of the structure can be always divided into three addenda:

$$\sigma_{ij} = (\sigma_{ij})_m + (\sigma_{ij})_b + (\sigma_{ij})_{nl} \quad (8.1)$$

where:

- $(\sigma_{ij})_m$ is the mean value of stress across the cross section
- $(\sigma_{ij})_b$ is the part of stress which varies linearly on the cross section and which, when integrated, have the same bending moment as the total stress.
- $(\sigma_{ij})_{nl}$ is the non-linearly distributed stress, i.e. the difference between the total stress σ_{ij} and the linearly distributed stress $[(\sigma_{ij})_m + (\sigma_{ij})_b]$

More details on linearization can be found on paragraph IC 2512 of SDC-IC [4]. Fig. 8.1 represents the stress linearization (x is the coordinate on a generic supporting line segment that lies on the cross section).

8.1.3 Stress categorization

With regard to stress categorization, the stress components can be divided in primary and secondary, defined as follows.

8.1.3.1 Primary stress

A normal stress or a shear stress developed by the imposed loading, which is necessary to satisfy the simple laws of equilibrium of external and internal forces and moments. The basic characteristic of a primary stress is that it is not self-limiting. Primary stresses which considerably exceed the yield strength will result in failure or at least in gross distortion. A thermal stress is not classified as a primary stress. Primary membrane stress is divided into general and local categories. A general primary membrane stress is one which is so distributed in the structure that no redistribution of load occurs as a result of yielding.

Examples of primary stress are:

- general membrane stress in a circular cylindrical or a spherical shell due to internal pressure or to distributed live loads;
- bending stress in the central portion of a flat head due to pressure.

8.1.3.2 Secondary stress

A normal stress or a shear stress developed by the constraint of adjacent parts or by self-constraint of a structure. The basic characteristic of a secondary stress is that it is self-limiting. Local yielding and minor distortions can satisfy the conditions which cause the stress to occur and failure from one application of the stress is not to be expected.

Examples of secondary stress are:

- general thermal stress;
- bending stress at a gross structural discontinuity.

8.1.4 Stress classification according to ASME

The ASME VIII rules provides a stress classification that takes account of both linearization (dividing the stress into membrane, bending and non-linear components) and categorization (dividing the stress into primary and secondary components). The total stress classification follow the rules:

- $\sigma_{tot} = P_m + P_b + Q + F$ if the membrane stress is not intensified by means of a discontinuity of geometry
- $\sigma_{tot} = P_m + P_b + Q + F$ if the membrane stress is intensified by means of a discontinuity of geometry

Tab. 8.2, taken from Appendix 4 of ASME VIII Div. 2, gives a definition of the different categories of stress (P_m , P_L , P_b , Q and F), and also summarizes the verifications to be made. Tab. 8.1, taken from Appendix 4 of ASME VIII Div. 2, can help the designer to classify the stresses, in cases where there could be some uncertainty.

8.1.5 Static verifications for vacuum vessels

In order to prevent the failure modes:

1. Excessive elastic deformation affecting functional adequacy.
2. Excessive plastic deformation (immediate plastic collapse)

the ASME design code define the following limits:

$$P_m < k \cdot S_m$$

$$P_L < 1.5 \cdot k \cdot S_m$$

$$P_L + P_b < 1.5 \cdot k \cdot S_m$$

where

- k is the stress intensity factor, which depends on the service level.
- S_m is the allowable stress, which is defined as the least of 2/3 the material yield strength and 1/3 the material ultimate strength.

Tab. summarizes the service levels and the stress intensity k factors corresponding to the ITER loading event categories.

The static verifications are performed by comparing in every point some significant combinations of stress categories (P_m , P_L or $P_L + P_b$) with a quantity (S_m) that gives a measure of the material strength.

Stress category	Primary			Secondary Membrane plus bending	Peak
	General Membrane	Local Membrane	Bending		
Description	Average primary stress across solid section. Excludes discontinuities and concentrations. Produced only by mechanical loads.	Average stress across any solid section. Considers discontinuities but not concentrations. Produced only by mechanical loads.	Component of primary stress proportional to distance from centroid of solid section. Excludes discontinuities and concentrations. Produced only by mechanical loads.	Self-equilibrating stress necessary to satisfy continuity of structure. Occurs at structural discontinuities. Can be caused by mechanical load or by differential thermal expansion. Excludes local stress concentrations.	(1) Increment added to primary or secondary stress by a concentration (notch) (2) Certain thermal stresses which may cause fatigue but not distortion of vessel shape
Symbol	P_m	P_L	P_b	Q	F
Combination of stress components and allowable limits of stress intensities	<pre> graph TD Pm[Pm] --> Km((k * Sm)) PL[PL] --> K15Sm1((1.5 * k * Sm)) Pb[Pb] --> K15Sm2((1.5 * k * Sm)) Q[Q] --> K3Sm((3 * Sm)) F[F] --> Sa((Sa)) Pm -.-> Km PL -.-> K15Sm1 Pb -.-> K15Sm2 Q -.-> K3Sm F -.-> Sa Km --> K15Sm1 K15Sm1 --> K15Sm2 K15Sm2 --> K3Sm K3Sm --> Sa Pm -.-> K15Sm1 PL -.-> K15Sm1 Pb -.-> K15Sm1 K15Sm1 --> K3Sm K3Sm --> Sa Pm -.-> K3Sm K3Sm --> Sa Pm -.-> Sa PL -.-> Sa Pb -.-> Sa Q -.-> Sa F -.-> Sa </pre> ————— Use design loads - - - - - Use operating loads				

Table 8.1: Stress categories and limits of stress intensities.

Vessel component	Location	Origin of stress	Type of stress	Classification
Cylindrical or spherical shell	Shell plate remote from discontinuities	Internal pressure	General membrane Gradient through plate thickness	P_m Q
		Axial thermal gradient	Membrane Bending	Q Q
	Junction with head or flange	Internal pressure	Membrane Bending	P_L Q
Any shell or head	Any section across entire vessel	External load or moment, or internal pressure	General membrane averaged across full section. Stress component perpendicular to cross section	P_m
		External load or moment	Bending across full section. Stress component perpendicular to cross section	P_m
	Near nozzle or other opening	External load moment, or internal pressure	Local membrane Bending Peak (fillet or corner)	P_L Q F
	Any location	Temp. diff. between shell and head	Membrane Bending	Q Q
Dished head or conical head	Crown	Internal pressure	Membrane Bending	P_m P_b
	Knuckle or junction to shell	Internal pressure	Membrane Bending	P_L Q
Flat head	Center region	Internal pressure	Membrane Bending	P_m P_b
	Junction to shell	Internal pressure	Membrane Bending	P_L Q
Perforated head or shell	Typical ligament in a uniform pattern	Pressure	Membrane Bending Peak	P_m P_b F
	Isolated or atypical ligament	Pressure	Membrane Bending Peak	Q F F
Nozzle	Cross section perpendicular to nozzle axis	Internal pressure or external load or moment	General membrane. Stress component perpendicular to cross section.	P_m
		External load or moment	Bending across nozzle section	P_m
	Nozzle wall	Internal pressure	General membrane Local membrane Bending Peak	P_m P_L Q F
		Differential expansion	Membrane Bending Peak	Q Q F
Cladding	Any	Radial temperature distribution	Equivalent linear stress Nonlinear portion of stress distribution	Q F
Any	Any	Any	Stress concentration (notch effect)	F

Table 8.2: Classification of stresses for some typical cases.

ITER Loading Event Category	Service Level (per ASME code)	Damage Limit	Stress Intensity “k” Factor
I Operational	A	Normal	1.0
II Likely	A	Normal	1.0
III Unlikely	C	Emergency	1.2
IV Extremely unlikely	D	Faulted	2.0

Table 8.3: Stress intensity factors according ASME, Section VIII, Division 2.

8.1.6 Ratcheting verification for vacuum vessels

In order to prevent the progressive deformation (ratcheting) failure mode the ASME design code define the following limit:

$$(P_m + P_b + Q)_{range} < 3.0 \cdot S_m$$

Ratcheting is defined as a progressive incremental inelastic deformation which can occur in a component that is subjected to variations of mechanical stress, thermal stress, or both.

The ratcheting verifications are performed by comparing in every point the range (during cyclic loading) of a significant combination of stress categories ($P_m + P_b + Q$) with a quantity ($3 \cdot S_m$) that gives a measure of the material capability to withstand progressive deformation.

8.1.7 Fatigue verification for vacuum vessels

In order to prevent the fatigue and crack propagation failure mode the ASME design code define the following limit:

$$(P_m + P_b + Q + F)_{range} < S_a$$

where S_a depends on the material and on the number of cycles. The fatigue verifications are performed by comparing in every point the range (during cyclic loading) of a significant combination of stress categories ($P_m + P_b + Q + F$) with a quantity (S_a) that gives a measure of the material capability to resist to a certain number of load cycles.

8.1.8 Buckling verification for vacuum vessels

In order to prevent the structural instability (buckling) failure mode the ASME design code define the following limits:

- Load factor < Load factor limit (Load controlled buckling)
- Strain factor < Strain factor limit (Strain controlled buckling)

where the factors and factors limits are defined as follows.

8.1.8.1 Load controlled and strain controlled buckling

Details of the analysis and the limits depend on whether the buckling is load-controlled or strain-controlled:

1. **Load controlled buckling.** Buckling is said to be load controlled if it is the result of externally imposed live loads, which are not reduced by the deformation associated with the buckling. It is characterized by continued application of an applied load in the post-buckling regime, leading to failure;
2. **Strain controlled buckling.** It is characterized by the immediate reduction of strain induced load upon initiation of buckling, and by the self-limiting nature of the resulting deformations.

Considering that for vacuum vessel buckling would be the result of mechanical loads (like internal vacuum and component weights), which are not reduced by the deformation associated with the buckling itself, load controlled buckling is to be considered.

8.1.8.2 ASME rules for buckling prevention

ASME VIII div.2 and ASME III NC codes contain minimal rules to prevent the two types of buckling. Both the codes contain DBF rules for vessel loaded by external pressure, which are used for calculating the required thickness of simple axisymmetric shells. These formulas contain safety factors equal to 3 and 4 for cylindrical and spherical shells respectively. The DBF approach is not considered appropriate for a complicated structure like the BSV. Hence DBA approach, which is present only in the ASME III NC code, will be preferred. Table 6.4 reports the Load Factor limits for load controlled buckling, as reported in ASME VII Div. 2 and ASME III NH codes. Calculated load factors shall be equal or exceed the values given in the table. The Load Factor for buckling is calculated as:

$$\text{Load Factor} = \frac{\text{Load that would cause buckling}}{\text{Load that occurs in service conditions}}$$

	ASME VIII Div. 2 (not considered)		ASME III NH (considered)	
ITER Criteria	Load Combinations	Load factor limit	Service limits	Load factor limit
Level A	Design A (normal)	3.0 – 4.0 ⁽¹⁾	Level A	3.0
			Level B	3.0
Level C	N/A		Level C	2.5
Level D	N/A		Level D	1.5

Notes: ⁽¹⁾ Limits are implicit in formulas for the definition of the minimum required thickness to prevent buckling (DBF approach). Load factors 3 and 4 are used for vessel of cylindrical and spherical shapes respectively

Table 8.4: Time-independent load factor limits to prevent buckling failure.

8.2 Comparison between analytical and numerical models for buckling analysis

One of the main issues, regarding the design of vacuum vessels, is the ability of the vessel to fulfil the buckling (structural instability) requirements. In fact, the vessel must be dimensioned in order not to go under structural instability (with a certain safety margin) when the vacuum is created inside it.

The Buckling Load Factor (BLF) is the main parameter in evaluating if the structure is safe, with regard to buckling requirements.

In the following, the BLF of a cylinder will be calculated with different approaches, considering the data in Tab. 8.5. The dimensions are similar to the ones of the vacuum vessel of the ITER NBI, and the material data are the ones of stainless steel.

Utilizing always these parameters, a certain number of calculations were performed:

- Analytical calculations, with the Flügge formula [5].
- Numerical (FEM) calculations, with the ANSYS CLASSIC code. With this software the cylinder was modelled using 5 types of shell elements and 2 types of solid elements.
- Numerical (FEM) calculations, with ANSYS WORKBENCH code. With this software the cylinder was modelled using 2 types of solid elements.

E	Young's modulus	194 GPa
ν	Poisson's modulus	0.3
r	Average radius	2222.5 mm
t	Shell thickness	45 mm
L	Length	3500 mm
p	External pressure	0.1013 MPa

Table 8.5: Parameters used for all the calculations.

This work has the following goals:

- To compare the buckling results calculated for a cylinder with different elements types and meshes. In particular it is interesting to investigate how much the buckling results are modified if the mesh is irregular (for example, a solid mesh made by tetrahedra), and to assess if the buckling results calculated for the ELISE vacuum vessel are to be considered as consistent. In that case, the model is made with “3-D 10-Node Quadratic Tetrahedron” elements.
- To investigate the accuracy of the various models also in stress and deformation calculations. Therefore, to confirm also stress and deformation results (and verifications) made for the ELISE vacuum vessel.

8.2.1 Analytical calculations for buckling

For a straight slender column, the Euler formula [6, 7] is represented by:

$$F_{cr} = \frac{\pi^2 \cdot E \cdot I}{L^2} \quad (8.2)$$

And the equation for hoop stress in cylinder is:

$$\sigma_{cr} = \frac{P \cdot d}{2 \cdot t} \quad (8.3)$$

where P is the externally applied pressure, d is the cylinder diameter and t the cylinder thickness. Considering the quarter symmetry of two-lobe buckling and neglecting the arc of a 90° segment [8], applying Euler's equation to the shell yields:

$$\frac{pd}{2} = \frac{\pi^2 Et^3}{12 \left(\frac{\pi d}{4} \right)} \quad (8.4)$$

Solving for the pressure load, we get:

$$p_{cr} = \frac{8E}{3} \left(\frac{t}{d} \right) \quad (8.5)$$

Eigenmodes m	Buckling critical pressure p_{cr}	Buckling Load Factor BLF
2	328.21	3239.94
3	49.50	488.66
4	14.01	138.32
5	7.91	78.04
6	7.51	74.16
7	8.76	86.47
8	10.71	105.75
9	13.11	129.37
10	15.85	156.49

Table 8.6: Buckling load factor calculated with the Flügge formula.

The equation above shows the important effect of shell thickness and diameter on buckling capability.

For finite length cylinders, an eigenvalue solution for the critical pressure for buckling is given by Flügge [9]:

$$p_{cr} = \frac{Et}{r(1-\nu^2)} \left\{ \frac{(1-\nu^2)\lambda^2 + k[(\lambda^2 + m^2)^4 - 2(\nu\lambda^2 m^4 + m^6)] + 2(2-\nu)\lambda^2 m^2 + m^4}{m^2(\lambda^2 + m^2)^2 - m^2(3\lambda^2 + m^2)} \right\} \quad (8.6)$$

where:

$$\lambda = \frac{\pi r}{l}$$

$$k = \frac{t^2}{12r^2}$$

And m is the number of the lobes in the buckled shape. The Buckling Load Factor can be calculated as:

$$BLF = \frac{p_{cr}}{p} \quad (8.7)$$

Where p is the operational external pressure and p_{cr} is the critical pressure for buckling.

In our case, the values in Tab. 8.5 are utilised for the analytical calculation of the BLF. The Flügge formula gives the following results, with different values for m :

Hence, the main results of buckling analytical calculations are the following:

- The principal eigenmode is 6, which yields the smallest load factor. This means that the buckled shape of the structure would be with 6 lobes.
- The BLF is 74.16. This means that the cylinder can tolerate an external pressure that is 74.16 times larger than the operational one (0.1013 MPa)

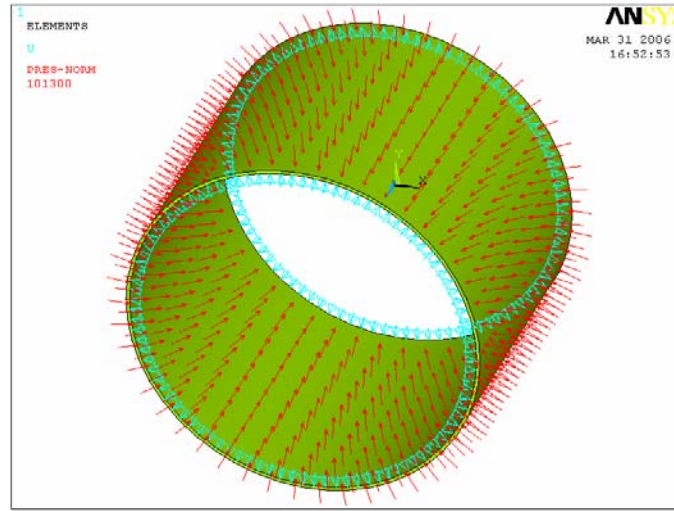


Figure 8.2: Loads and boundary conditions of all the numerical models.

8.2.2 Numerical calculations for buckling with regular mesh

Fig. 8.2 shows the loads and boundary conditions applied to all the numerical models:

- Loads: An atmospheric external pressure (101300 Pa) is applied to the external face of the cylinder.
- Boundary conditions: The edges are fixed in the radial direction.

The following Tab. 8.7 reports the main results of the numerical analyses performed with regular mesh, i.e. quadrilateral shells or hexahedral solids [10].

8.2.2.1 Observations

- The BLFs calculated using a regular mesh are all quite similar, and also the coherence with the analytical result is good.
- The shell models give a slightly bigger value for the buckling load factor, compared to the solid models.
- The fact that the shell elements are linear (without midside nodes) or quadratic (with midside nodes) doesn't affect the results (stress, deformation and buckling) sensibly.
- The equivalent stress and BLF calculated using linear solid elements (6÷8) are slightly higher than the ones calculated using quadratic solid elements (9÷11). However, the gap is very small (a few percentage points).

Ansys Classic/Workbench	#	Element type (and specifications)	Linear/ quadratic	Element description	Element number	Eigenmode number	Maximum stress [Mpa]	Maximum radial deformation [mm]	Buckling Load Factor
C	1	SHELL43	L	4-Node Plastic Large Strain Shell	2304	6	5.75	0.062	76.7
	2	SHELL63	L	4-Node Elastic Shell	2304	6	5.73	0.062	75.4
	3	SHELL93	Q	8-Node Structural Shell	2304	6	5.75	0.061	75.2
	4	SHELL143	L	4-Node Plastic Small Strain Shell	2304	6	5.75	0.062	76.7
	5	SHELL181	L	4-Node Finite Strain Shell	2304	6	5.63	0.062	74.8
	6	SOLID45 (1 element along thickness)	L	3D 8-Node Structural Hexahedron	2304	6	5.75	0.062	75.2
	7	SOLID45 (2 elements along thickness)	L	3D 8-Node Structural Hexahedron	4608	6	5.76	0.062	75.2
	8	SOLID45 (3 elements along thickness)	L	3D 8-Node Structural Hexahedron	6912	6	5.77	0.063	75.2
	9	SOLID95 (1 element along thickness)	Q	3D 20-Node Quadratic Hexahedron	2304	6	5.71	0.061	73.7
	10	SOLID95 (2 element along thickness)	Q	3D 20-Node Quadratic Hexahedron	4608	6	5.7	0.061	73.7
	11	SOLID95 (3 element along thickness)	Q	3D 20-Node Quadratic Hexahedron	6912	6	5.7	0.061	73.7
W	12	Automatic sizing/ hexahedron	Q	3D 20-Node Quadratic Hexahedron	2304	6	5.73	0.061	73.2

Table 8.7: FEM calculation results obtained with regular mesh.

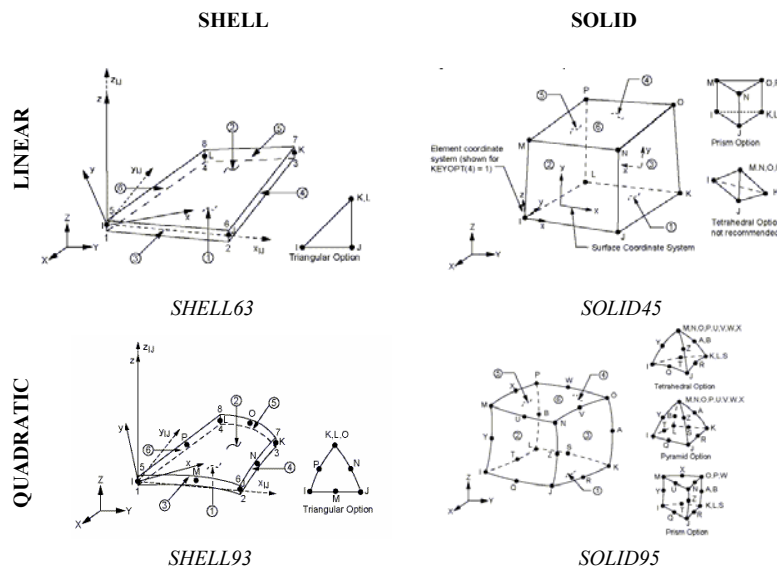


Figure 8.3: Linear and quadratic elements used for the numerical calculations.

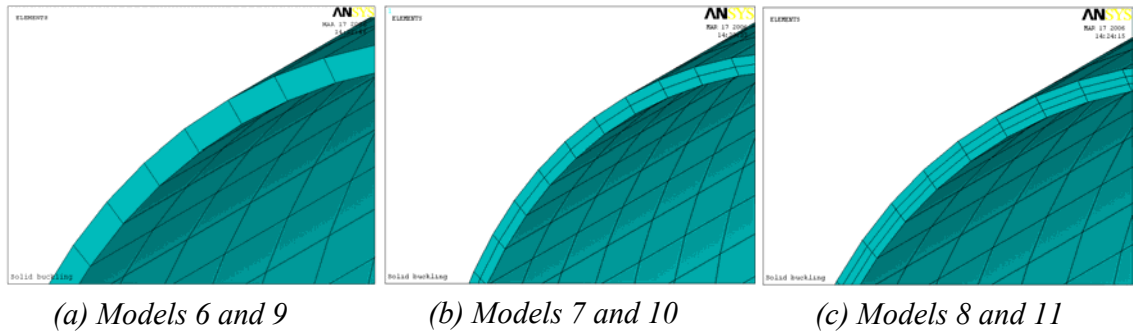


Figure 8.4: Elements along thickness direction in solid models.

- Comparing models 6,7,8 and 9,10,11: the number of elements along the thickness direction (1,2 or 3) for solid elements doesn't affect the results (stress, deformation and buckling) sensibly.
- Comparing models 9,12: the software (Classic or Workbench) doesn't sensibly affect the results (stress, deformation and buckling). It can be noted that the mesh and element type are the same in these two models.

In the following paragraphs, three models will be illustrated, in order to give a sample of the structural analyses performed. For every model the following figures will be reported:

1. The mesh
2. The Equivalent (Von Mises) stress contour plot
3. The radial deformation contour plot
4. The first buckling mode contour plot

8.2.2.2 Sample of analysis with shell elements in Ansys Classic

Fig. 8.5 show the model and the most important results of the analysis performed using SHELL181 elements in Ansys Classic (model 5). The maximum equivalent (Von Mises) stress - of about 5.6 MPa - is reached on two symmetric cylindrical regions. The maximum radial deformation - of about 62 mm - is reached on the central part of the cylinder. The buckling analysis gives 6 periodical deformation patterns (6 identical inner lobes alternated to 6 outer lobes) and a BLF of 74.8. This is in good agreement to what calculated analytically with the Flügge formula (a minimum BLF of 74.16 calculated with the eigenvalue 6).

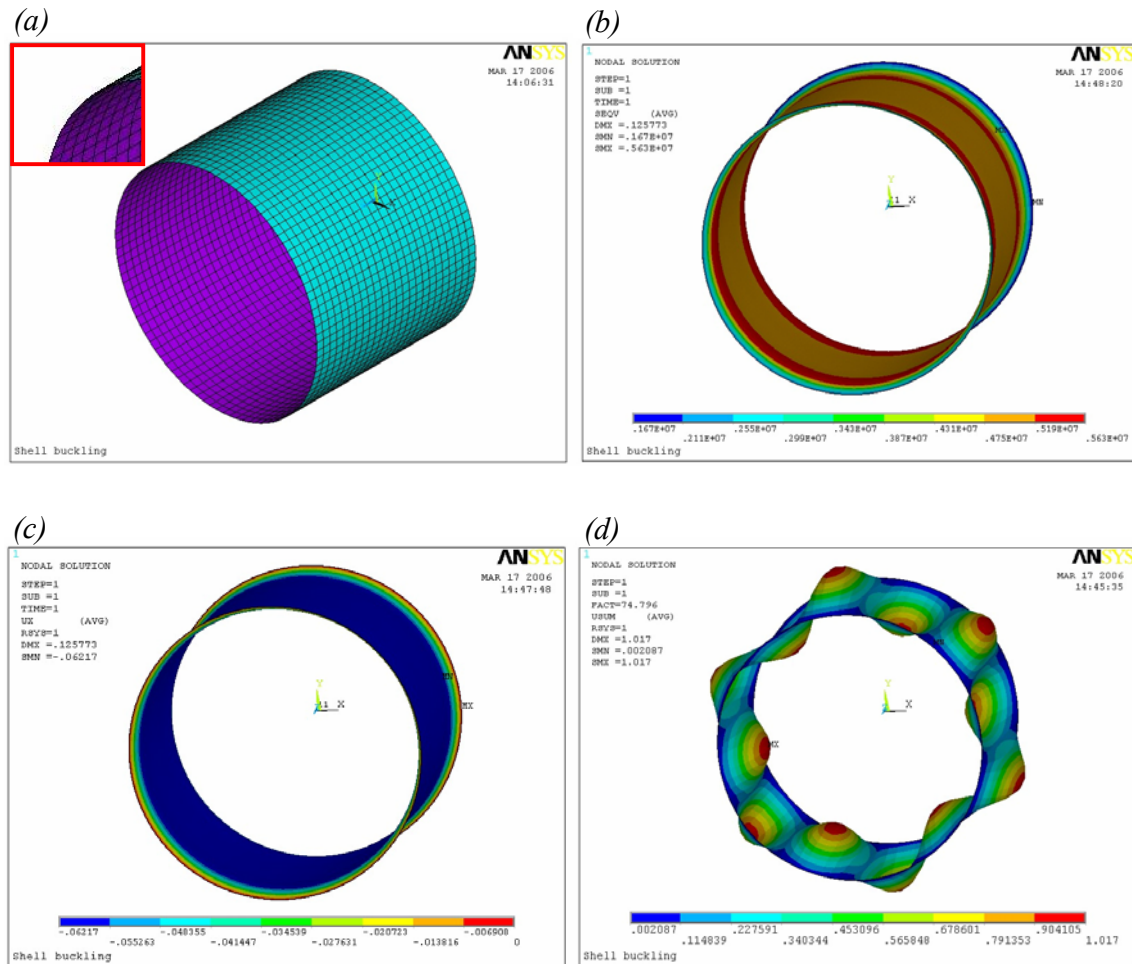


Figure 8.5: Results with a Shell model in ANSYS classic (model 5): (a) Mesh; (b) Equivalent (Von Mises) stress contour plot; (c) Radial deformation contour plot; (d) First buckling mode contour plot.

8.2.2.3 Sample of analysis with solid elements in Ansys Classic

Also the results obtained with a solid model in ANSYS classic, with a regular and sufficiently fine mesh, are in a good agreement with the analytical formula. Fig. 8.6 shows in particular the model and principal results of the analysis performed using SOLID95 elements with one element along thickness (model 9). The most important outputs (maximum equivalent stress, maximum radial deformation, buckling load factor) are in fact less then 2% different from the shell analysis in the previous paragraph, while the buckling periodicity is the same (6).

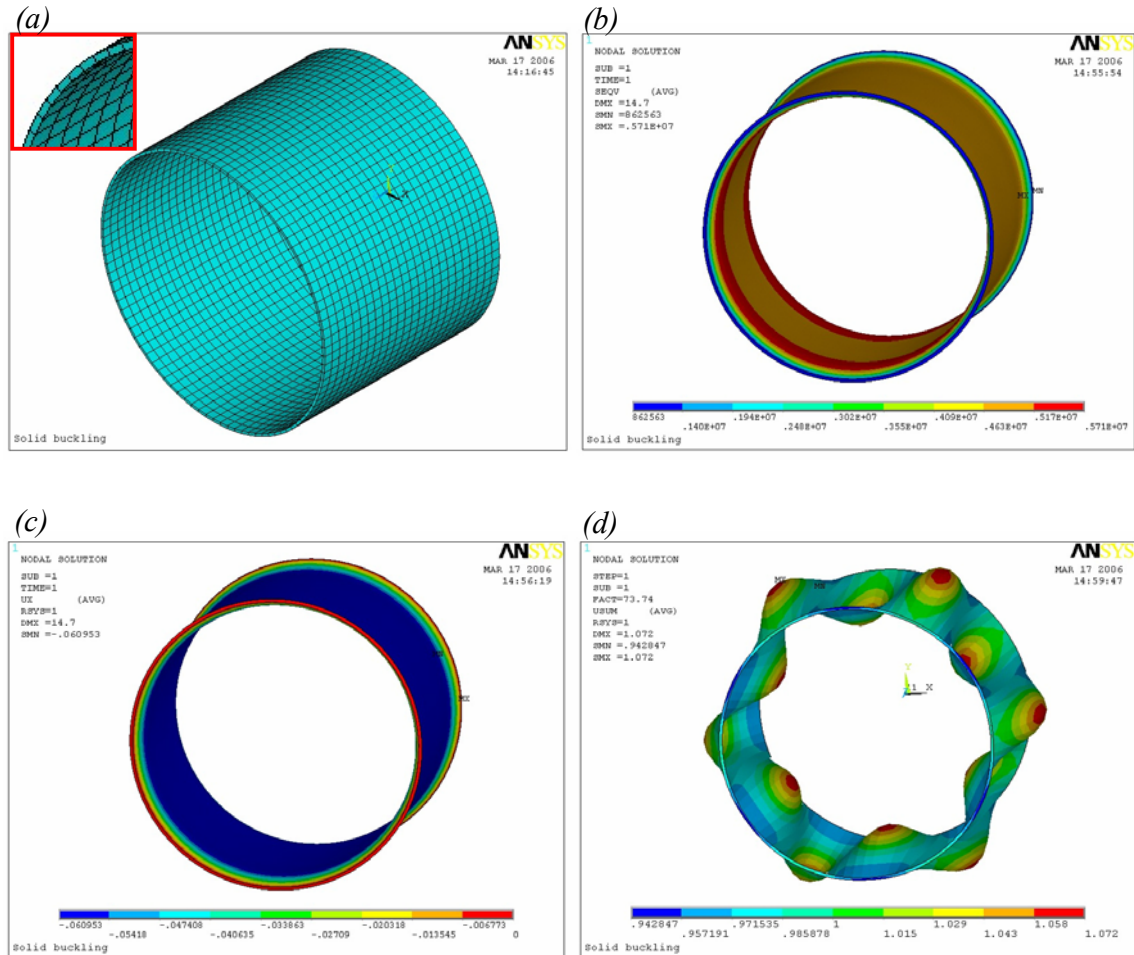


Figure 8.6: Results with a solid model in ANSYS classic (model 9): (a) Mesh; (b) Equivalent (Von Mises) stress contour plot; (c) Radial deformation contour plot; (d) First buckling mode contour plot.

8.2.2.4 Sample of analysis with solid elements in Ansys Workbench

Fig. 8.7 shows the model and the most important results of the analysis performed using Ansys Workbench with a regular mesh (model 12). Also in this case, the results are coherent with the analytical formula and the numerical analysis done with shell elements.

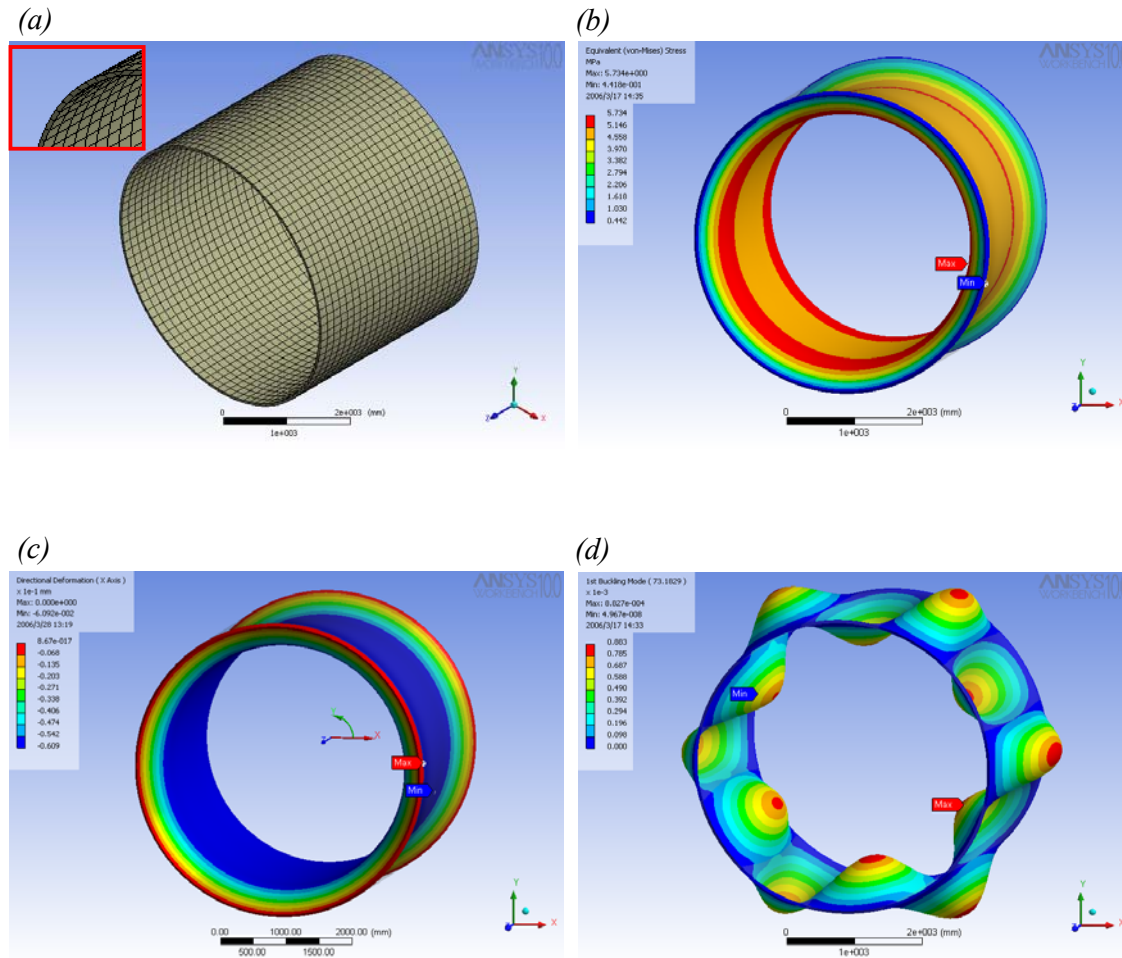


Figure 8.7: Results with a solid model in ANSYS Workbench (model 12): (a) Mesh; (b) Equivalent (Von Mises) stress contour plot; (c) Radial deformation contour plot; (d) First buckling mode contour plot.

8.3 Numerical calculations with irregular mesh

Tab. 8.8 reports the main results of the numerical analyses performed with irregular mesh, i.e. tetrahedra elements. Model 12, made with hexahedra elements, is here

Ansys Classic/Workbench	#	Element type (and specifications)	Linear/ quadratic	Element description	Element number	Eigenmode number	Maximum stress [Mpa]	Maximum radial deformation [mm]	Buckling Load Factor
W	12	Automatic/ hexahedra	Q	3D 20-Node Quadratic Hexahedron	2304	6	5.73	0.061	73.2
	13	Element Sizing 200/ tetrahedra	Q	3-D 10-Node Quadratic Tetrahedron	5001	6	6.00	0.063	90.6
	14	Element Sizing 180/ tetrahedra	Q	3-D 10-Node Quadratic Tetrahedron	5385	6	6.05	0.062	87.3
	15	Element Sizing 160/ tetrahedra	Q	3-D 10-Node Quadratic Tetrahedron	6424	6	6.11	0.063	84.2
	16	Element Sizing 140/ tetrahedra	Q	3-D 10-Node Quadratic Tetrahedron	8402	6	6.02	0.062	79.6
	17	Element Sizing 120/ tetrahedra	Q	3-D 10-Node Quadratic Tetrahedron	11195	6	5.90	0.061	76.5
	18	Element Sizing 100/ tetrahedra	Q	3-D 10-Node Quadratic Tetrahedron	15878	6	5.82	0.061	74.8
	19	Element Sizing 80/ tetrahedra	Q	3-D 10-Node Quadratic Tetrahedron	24728	6	5.80	0.061	74.0
	20	Element Sizing 60/ tetrahedra	Q	3-D 10-Node Quadratic Tetrahedron	43596	6	5.73	0.061	73.7

Table 8.8: FEM calculation results with regular (12) and irregular (13÷20) solid models.

reported only for comparison. The element sizing, that is an input to the Ansys mesher tool, is decreased going from model 13 to 20. As a consequence, the mesh becomes finer and the number of elements increases.

A view of the models is reported in Fig. 8.8.

8.3.1 Comparison between buckling models

In this section, a comparison is made between the different models, considering the equivalent stress, the deformation and the buckling load factor as parameters.

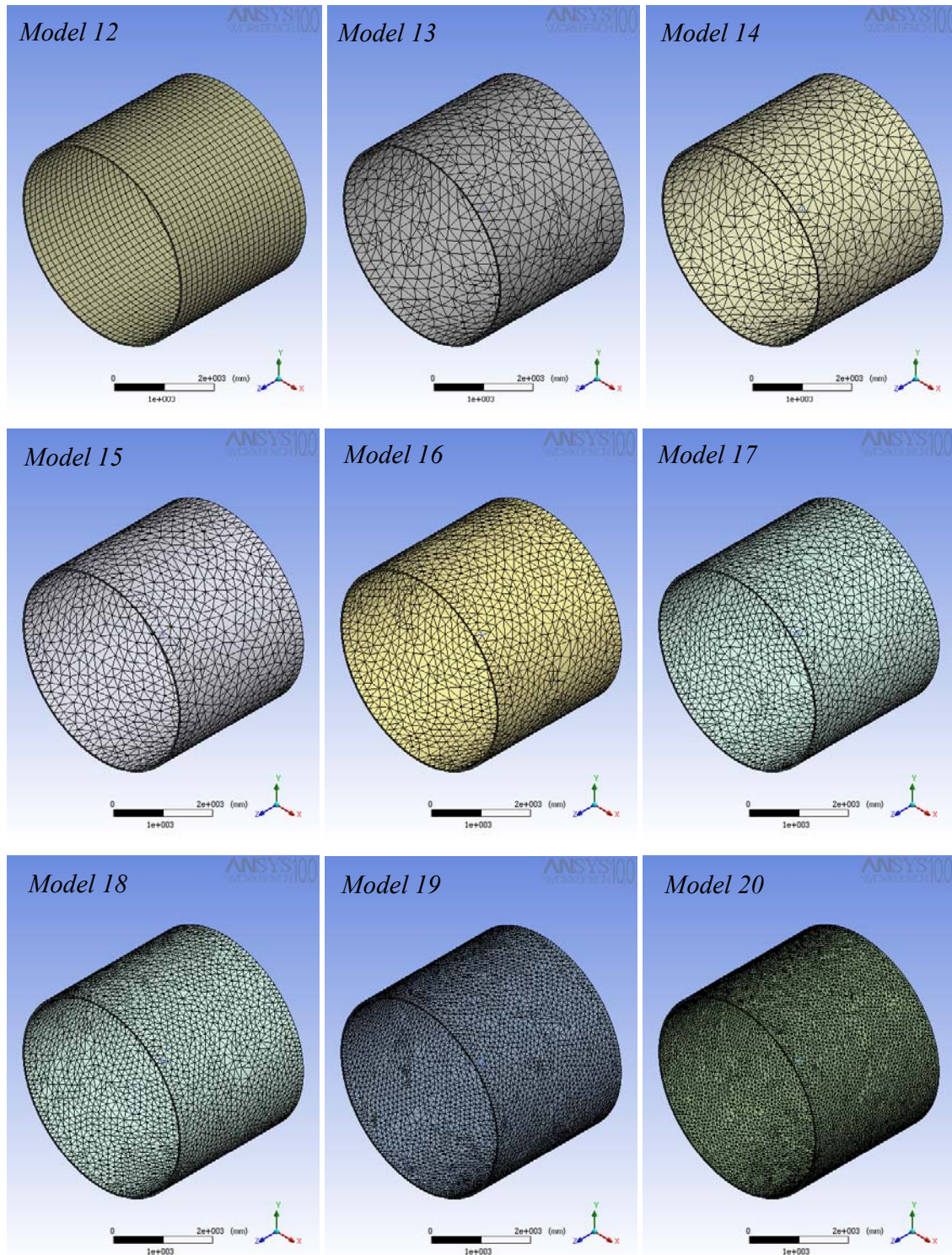


Figure 8.8: Regular (12) and irregular (13÷20) solid models.

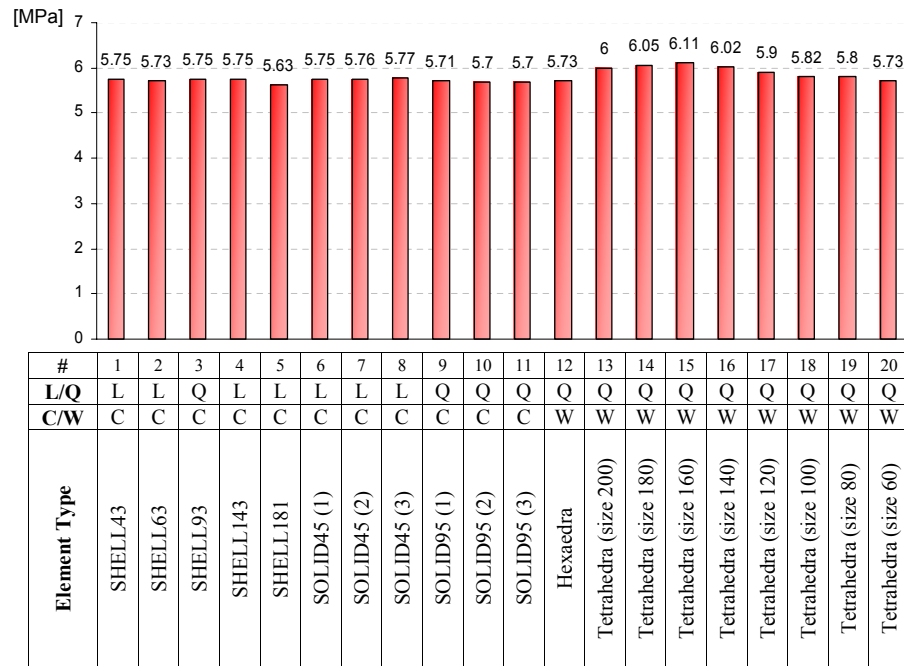


Figure 8.9: Maximum equivalent (Von Mises) stress calculated using various models.

8.3.1.1 Comparison on Equivalent (Von Mises) stress

The calculated maximum equivalent (Von Mises) stress is higher if the mesh is made with tetrahedra. This is particularly true when the tetrahedral mesh is coarse, like in model 13, 14 and 15.

However, with the tetrahedral mesh becoming progressively finer (models from 15 to 20) the corresponding values of the maximum equivalent stress become progressively closer to ones calculated using a regular mesh.

So, it is possible to argue that a coarse tetrahedral mesh could generally overestimate the stress results, but if the mesh is sufficiently fine the results are to be considered reliable. This happens when an asymptotic convergence of the solution is reached, i.e. when the results (in this case, the maximum equivalent stress) are not affected consistently if a smaller element size is imposed.

8.3.1.2 Comparison on deformations

The radial deformation results are not influenced by the model type in sensible way, as it is shown in the following histogram.

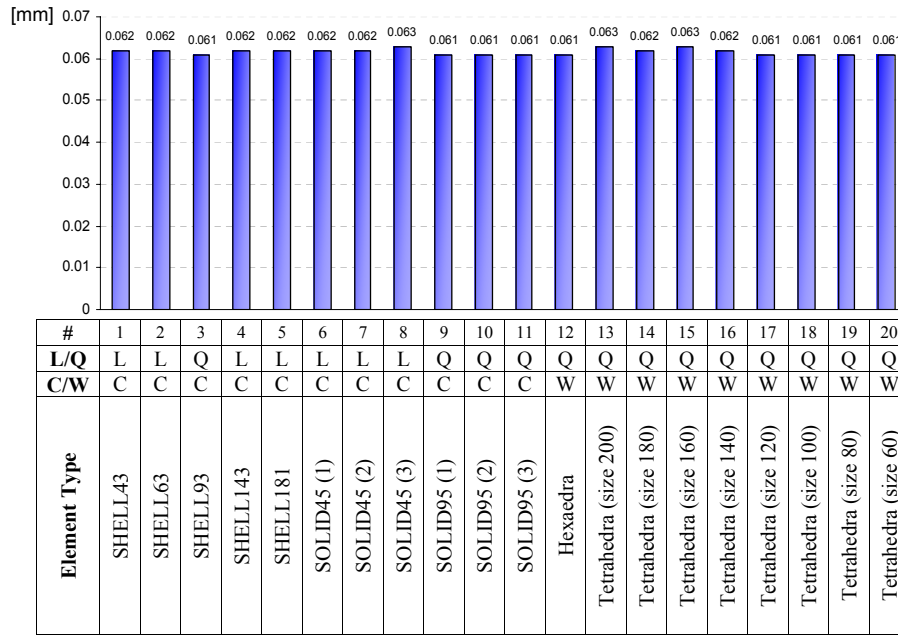


Figure 8.10: Maximum radial deformation calculated using various models.

8.3.1.3 Comparison on Buckling Load Factor

The buckling eigenmode numbers, calculated with all the methods (analytical and numerical), are all the same: 6, that corresponds to 6 lobes in the buckled shape. The corresponding load factors vary from 73.2 (minimum value, calculated using a mesh made of 2304 Quadratic Hexahedra) to 90.6 (maximum value, calculated using a mesh made of Quadratic Tetrahedra, meanly sized 200mm).

A reference value for the BLF can be identified as the one calculated with the analytical formula, equal to 74.2. The values calculated with the models with a regular mesh (1÷12) are quite similar to the reference value.

On the other hand, the BLF calculated with a very coarse tetrahedral mesh (model 13) is sensibly higher than the reference value (+22.1%).

However, with the tetrahedral mesh becoming progressively finer (models from 13 to 20) the corresponding values of the BLF become progressively closer to the reference value. So, it is possible to deduce that a coarse tetrahedral mesh could generally overestimate the BLF, but if the mesh is sufficiently fine the results are to be considered reliable. This happens when an asymptotic convergence of the solution is reached, i.e. when the results (in this case, the BLF) are not affected consistently if a smaller element size is imposed.

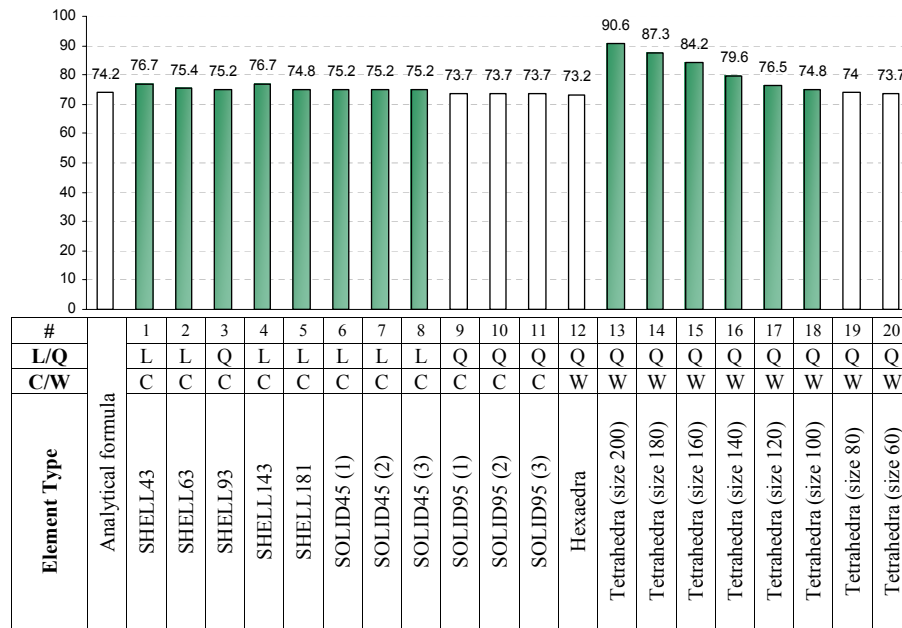


Figure 8.11: Buckling Load Factor calculated using various models.

8.4 Application: Design analyses and verifications of the ELISE vacuum vessel

8.4.1 General description

The design of the ELISE vacuum vessel was developed following the DBA (design by analysis) route. In particular, the iterative chart of Fig. 8.12 was considered.

The structural analyses were performed using the FEM software ANSYS Workbench 11.0. This software was preferred instead of Ansys Classic, because it contains a better CAD importer and it is optimized to perform FEM analyses starting from CAD models.

8.4.2 Material

The material chosen for the vacuum vessel is SS 316L [11]. The characteristics of SS 316L at 20°C are reported in Tab. 8.9. The allowable primary membrane stress was calculated as: $S_m = \min(2/3 \cdot S_y; 1/3 \cdot S_u)$.

8.4.3 Model description

The model contains:

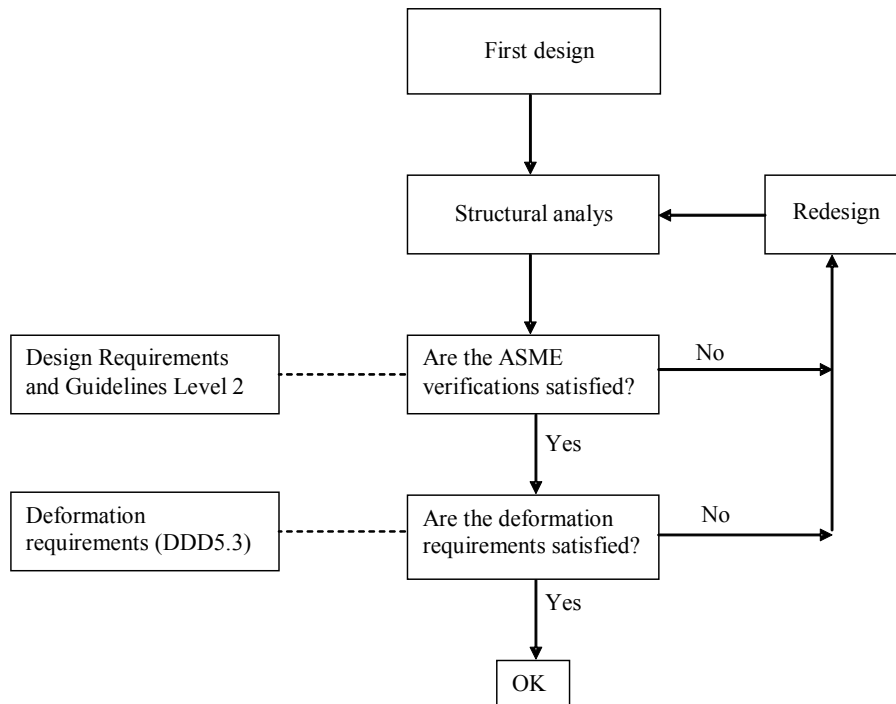


Figure 8.12: Considered design flow chart (DBA route).

Name	Symbol	Value
Young's modulus	E	197 GPa
Poisson's Ratio	ν	0.3
Yield stress	S_y	220 MPa
Ultimate tensile strenght	S_u	520 MPa
Allowable primary membrane stress	S_m	146 MPa

Table 8.9: Characteristics of SS 316L (Sheet/plate products).

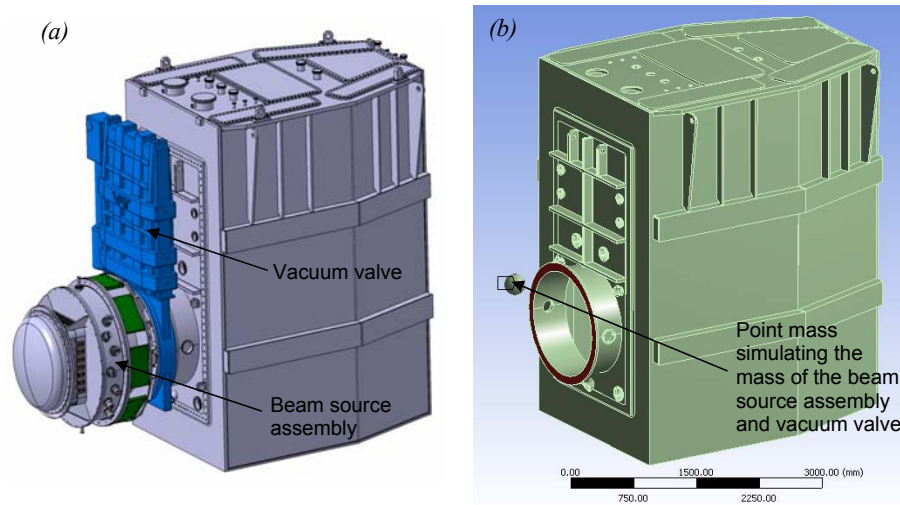


Figure 8.13: ELISE vacuum vessel: (a) CAD model, with ion source and vacuum valve; (b) FEM model.

- The ELISE vacuum vessel, which have a weight of approximately 22.6 tons.
- A point mass of 5.4 tons, which simulate the mass of the beam source assembly plus the vacuum valve. This mass is located at the barycentre of this two assembly, and it is linked to the main flange (red area in Fig. 8.13b).

As the original CAD geometry (Fig. 8.13a) - containing many holes, fillets and chamfers - was quite complicated, the Ansys Workbench mesher encountered some difficulties to create the mesh. In order to solve these problems, a simplified model was created in Catia, without all the details that did not influence the structural behaviour of the structure, and then imported to Ansys Workbench (Fig. 8.13b).

The FEM model is formed by 691495 nodes and 346051 elements. Fig. shows an overview of the mesh.

8.4.4 Loads and boundary conditions

The structure, as visible from Fig. 8.15, is subjected to the following loads:

- Inertial loads: Standard Earth Gravity (9.8 m/s^2)
- Pressure load: External pressure of 1.013 bar. The pressure on the main flange is higher (2.87 bar).

With regard to the structural boundary conditions, it is constraint as follows on the three supports:

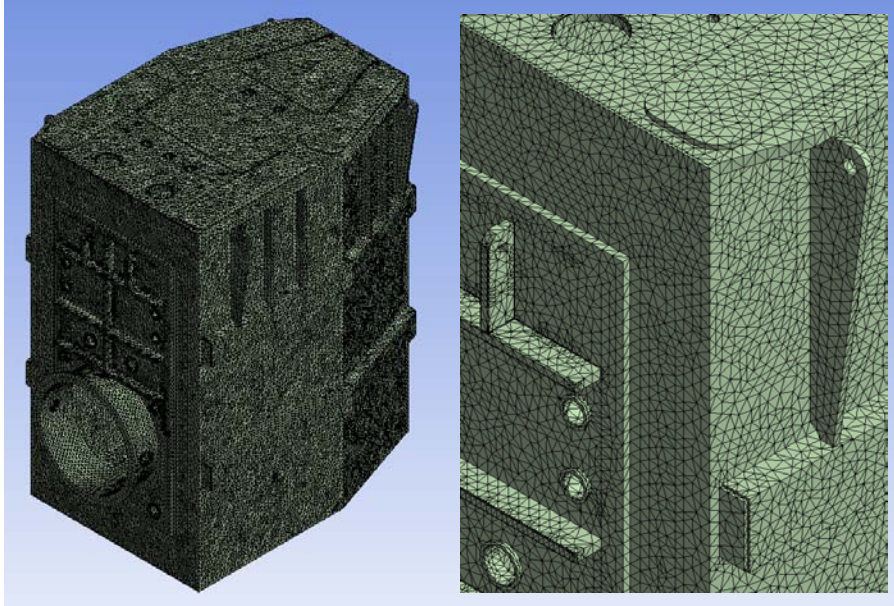


Figure 8.14: Mesh of the ELISE model.

- Constrained in x, y and z directions in D
- Constrained in x and z directions in E
- Constrained in z direction in F

In this way, the structure is isostatically constrained, and there are no localized stress given by the supports.

If, for instance, also the constraints in E and F were in the three directions x, y and z, there would be also high and localized stress near the supports, because the model consider them as infinitely rigid. This would be not a good approximation of the reality, as the rigidity of the supports is not infinite, and the localized near the supports are normally negligible in a vacuum vessel.

8.4.5 Static results and verifications

The P_m , P_L and P_b stress categories were evaluated, using the equivalent (Von Mises) stress, on every point and for every load case. The equivalent (Von Mises) stress contour plots are reported in Fig. 8.16.

The maximum value of the stress significant combinations (P_m and $P_L + P_b$) were evaluated considering a proper main section and considering the average value of the equivalent stress, as explicated in paragraph 1. The maximum local primary

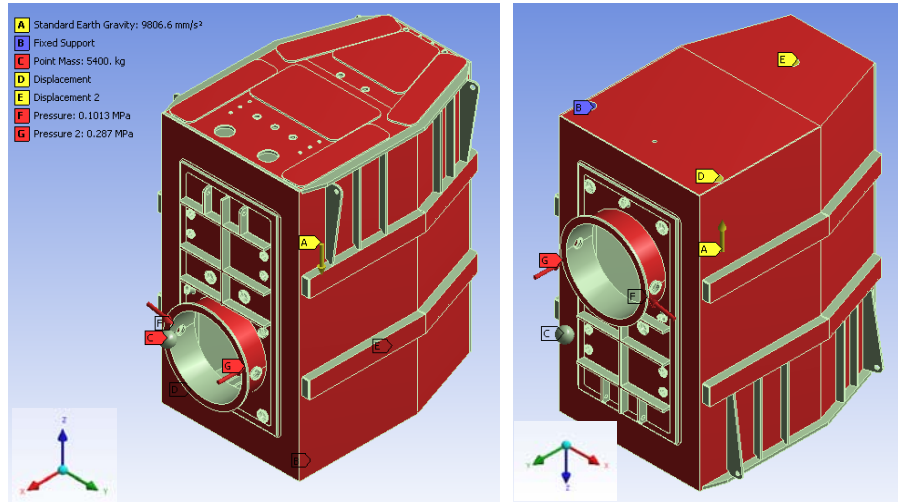


Figure 8.15: Loads and boundary conditions.

membrane stress (P_L) has always been considered coincident with the general primary membrane (P_m), because in this geometry no zones were found where the primary stress is amplified by a structural discontinuity (cross section reduction, nozzles etc.).

The maximum equivalent stress, of 165 MPa (see Fig. 8.16c) is located in a small area, near the edge of the internal stiffening structure. The stress distribution is neither constant nor linear along the part section, so the stress in that point is to be evaluated as a peak stress (F). Considering only the ELISE flange, the maximum stress (of 90 MPa) is also localized in a small area, and is also to be evaluated as a peak stress (F).

The maximum primary membrane stresses (P_m and P_L), of about 39 MPa, is located on the lateral plate (in this area the stress is almost constant on the cross section).

The maximum primary membrane+bending stress ($P_L + P_b$), of about 80 MPa, is located at the border between the back and lateral plate. In this area the stress varies almost linearly through the vessel thickness. Tab. 8.10 reports the structural static verifications, according to ASME VIII Div. 2 code.

As all the conditions are satisfied, the structure is statically verified.

8.4.6 Ratcheting results and verifications

The ASME design code defines the following limit for ratcheting:

$$(P_m + P_b + Q)_{range} < 3.0 \cdot S_m$$

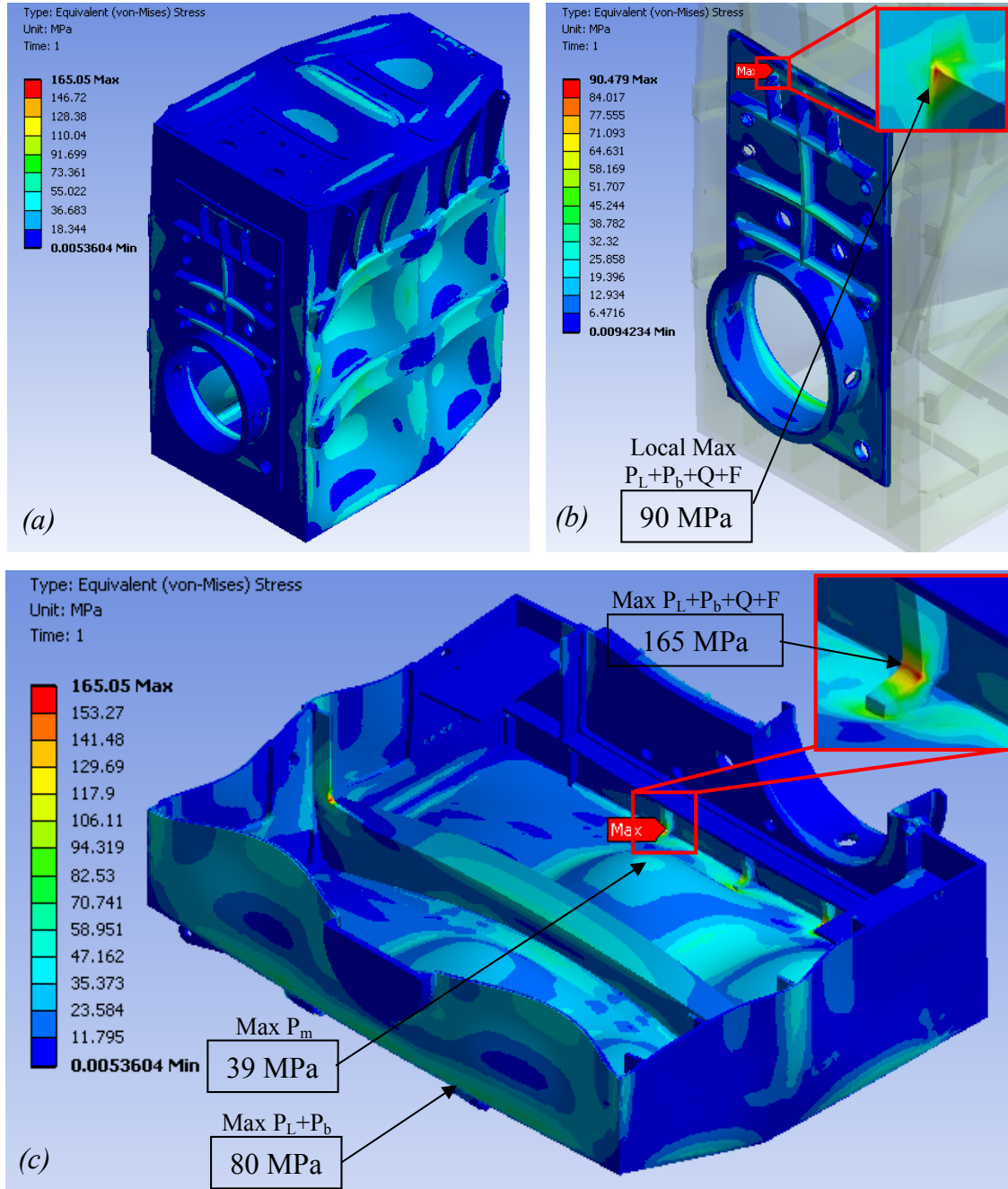


Figure 8.16: Equivalent (Von Mises) stress contour plots: (a) Overall view; (b) ELISE flange; (c) Vertical section, with maximum peak, membrane and bending stress.

Stress type	Maximum value [MPa]	Allowable value [MPa]	Verified
General primary membrane	39	146	Yes
Local primary membrane	39	146	Yes
Primary membrane + bending	80	220	Yes

Table 8.10: Static verifications summary.

The structure is subjected to a certain number of evacuation/pressurisation cycles. The results with (vacuum+weight) and (only weight), can be considered as extreme configurations of an evacuation/pressurisation cycle, in order to calculate the total stress range.

Fig. 8.17 shows the equivalent stress contour plots in the two cases. In order to make a comparison between vacuum loaded/unloaded contour plots, the plot legend is kept the same in the two cases. As it can be seen in Fig. 8.17 c and d, the maximum value of the $(P_m + P_b + Q)_{range}$ range on an evacuation/pressurisation cycle, reached on the lateral walls:

$$(P_m + P_b + Q)_{range,max} = 80 - 0 = 80 \text{ MPa}$$

As this value is lower than $3.0 \cdot S_m$ (348), the ratcheting verification is satisfied.

8.4.7 Fatigue results and verifications

The ASME design code defines the following limit for fatigue:

$$(P_m + P_b + Q + F)_{range} < S_a$$

The total stress range shall be considered for fatigue verification, which contains all the stress categories:

$$\Delta\sigma_{tot} = (P_m + P_b + Q + F)_{range}$$

The maximum value of the total stress range is (see Fig. 8.17e and 8.17f):

$$\Delta\sigma_{tot} = 165 - 15 = 150 \text{ MPa}$$

This value is lower than the critical values for fatigue, as can be seen in Fig. 8.18, that represent a design fatigue curve for nickel-chromium-iron alloys (source: ASME VIII div. 2 appendix 5 Figure 6.5-110.2.1)[2]. Hence, the fatigue verification is satisfied.

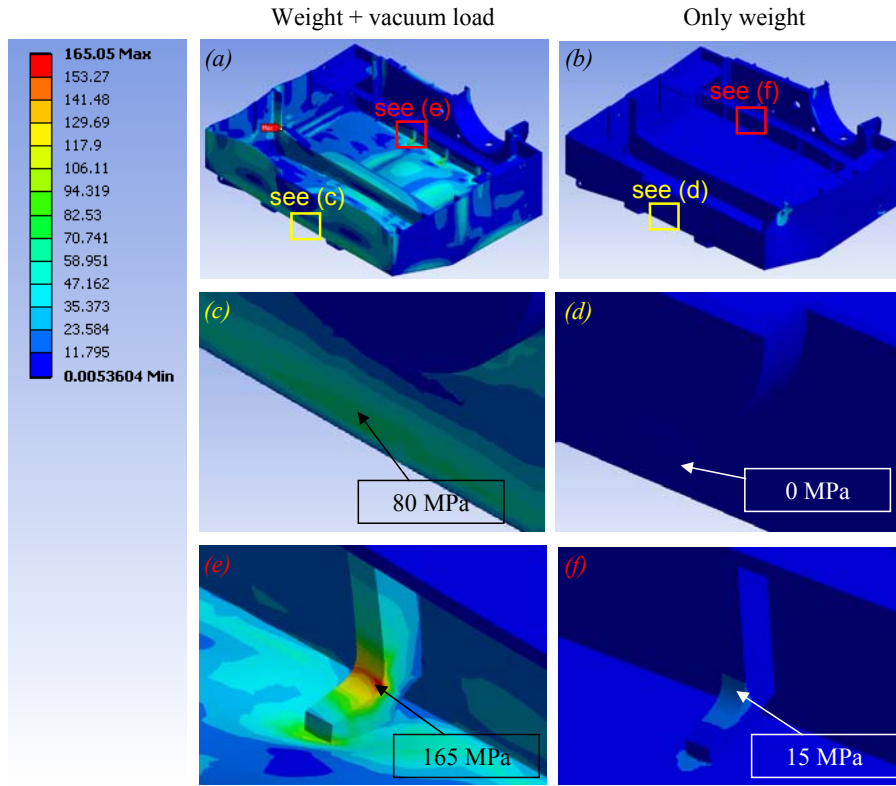


Figure 8.17: Equivalent (Von Mises) stress contour plots in the two extreme configurations of an evacuation/pressurisation cycle: (a) (b): Global views; (c) (d): Detail view of the most critical point for ratcheting; (e) (f): Detail view of the most critical point for fatigue.

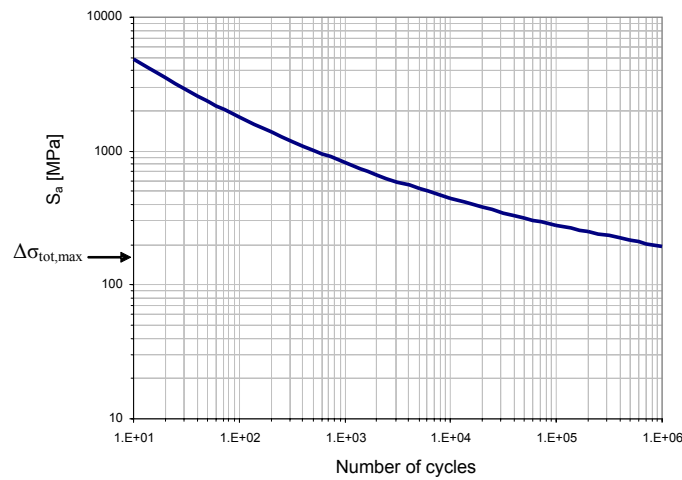


Figure 8.18: Design fatigue curve for nickel-chromium-iron alloys (from ASME).

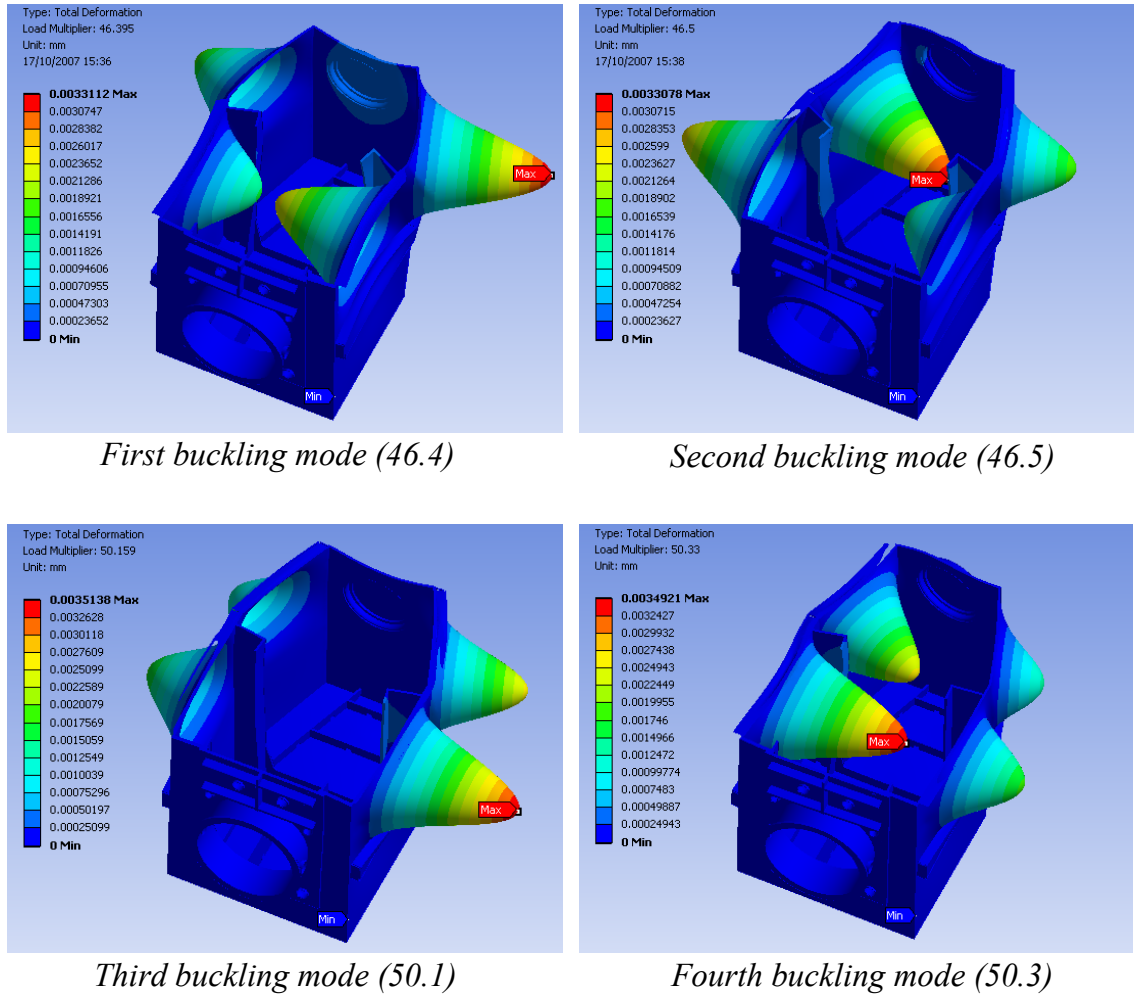


Figure 8.19: First four buckling modes.

8.4.8 Buckling results and verifications

The ASME design code defines the following limits for load controlled buckling verifications:

$$Loadfactor < Loadfactorlimit$$

The load factor limit to be taken is 3, according to ASME III NH code and considering service level A (operation normal loads). The calculated load factor (~ 46.4) for the first buckling mode is largely higher than the load factor limit (3). Hence the buckling verification is satisfied.

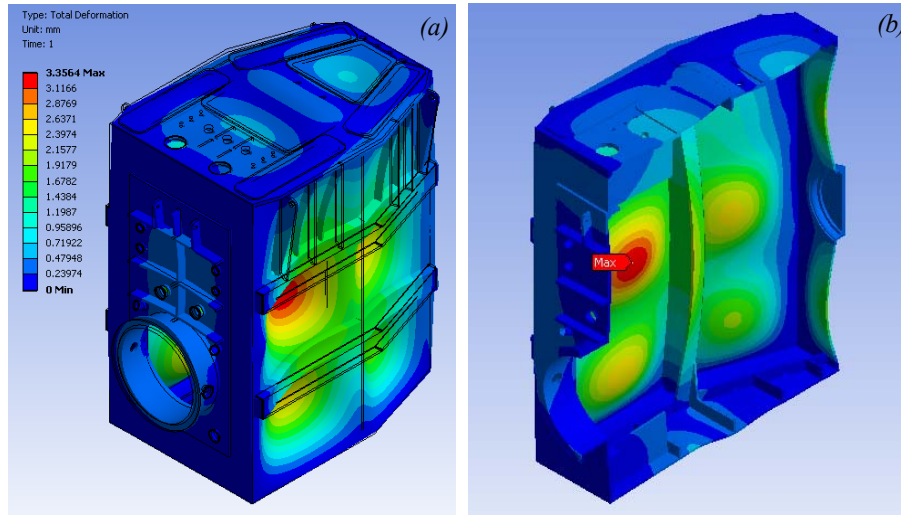


Figure 8.20: Total deformation contour plots: (a): Overall views; (b): Section showing the maximum.

8.4.9 Deformation results and verifications

In order to guarantee a correct operating behaviour, the structure is required to react with small deformations in any operating condition. Hence an additional verification, based on deformation values, was performed. The ITER limits are considered for deformations, taken from DDD5.3 [12, p. 68]:

1. Maximum deformation at any location: < 5 mm
2. Maximum deformation at the beam source supports: < 1 mm

Fig. 8.20 represents the total deformation of the structure when subjected to the vacuum and weight loads. The maximum deformation is foreseen to be located on the laterals walls of the vacuum vessel, with a value of about 3.3 mm, due to the external pressure. This deformation can be considered acceptable according to the ITER criteria.

8.5 Conclusions

The static, fatigue and deformation verifications methods according to the “ASME Boiler and Pressure Vessel Code” have been described. The methods have been applied to the design of the ELISE vacuum vessel, that satisfies all the verifications.

As a validation of the numerical methods used, a comparison have been done between different analytical and numerical approaches for the calculation of the

buckling modes of a structure subjected to external pressure. The general result is that, if the mesh is sufficiently fine, the buckling analysis gives consistent results, also in the case of an irregular mesh.

Bibliography

- [1] Various Authors. *Structural Design Criteria - General Section*. ITER Document No. G 74 MA 6 R0.1, 2004.
- [2] Various Authors. *ASME Boiler and Pressure Vessel Code, 2001 Edition, Section VIII Division 2, Alternative rules. (Rules for Construction of Pressure Vessels)*. The American Society of Mechanical Engineers, New York, 2001.
- [3] Various Authors. *ASME Boiler and Pressure Vessel Code, 2001 Edition, Section III Division 1 Subsection NC - Class 2 Components*. The American Society of Mechanical Engineers, New York, 2001.
- [4] Various Authors. *ITER SDC-IC, Structural Design Criteria for ITER In-Vessel Components*. July 2004.
- [5] W. Young. *Roark's Formulas for Stress and Strain*. McGraw-Hill, 1989.
- [6] Various Authors. *Manuale dell'Ingegnere*. 81esima edizione, Ulrico Hoepli Editore, Milano 1985.
- [7] A. P. Boresi, R. J. Schmidt and O. M. Sidebottom. *Advanced Mechanics of Materials*. 5th Edition, John Wiley & Sons, 1993.
- [8] C.H. Roche. *Comparing ANSYS Shell Elements for Buckling Analysis*. ANSYS Solutions magazine, Summer 2005.
- [9] W. Flügge. *Stresses in Shells*. Springer-Verlag, Berlin, 1960.
- [10] Various Authors. *Ansys 11.0 guide*. Ansys Inc.
- [11] Various Authors. *Metals Handbook*. 10th edition, Volume 1, ASM International Handbook Committee, ASM International, Materials Park, OH, 1990.
- [12] Various Authors. *Design Description Document (DDD 5.3), Neutral Beam Heating and Current Drive (NBH&CD) System*. ITER Document No. 53 DDD 29 01-07-03 R 0.1.

Conclusions

The present Doctorate Thesis has regarded some of the thermo-mechanical analysis and design of high power ion beam sources. The main original contributions have been:

- A new method for the post-processing of the calorimetric measurement on an ion source, able to precisely evaluate the power loads on the various components of the ion source. (Chapter 2)
- A new method for the evaluation of the damages given by sputtering inside an ion source. (Chapter 3)
- A new FEM technique especially developed for high performance cooling systems, that integrates non-linear CFD, thermal and structural analysis. (Chapter 6)

Moreover, different methods for the thermo-mechanical design have been considered, compared and applied to the high power ion beam sources analysis and design. In particular, the following themes have been investigated:

- A comparison between the main Decision Making methods, with an application on the accelerator cooling system design. (Chapters 4 and 5)
- A comparison between the most sophisticated approaches for Computational Fluid Dynamics, analytical formulas and experimental data, with an application on the ITER Neutral Beam Injector accelerator cooling system design. (Chapters 6 and 7)
- A comparison between different analytical and numerical approaches for the calculation of the buckling modes of a structure subjected to external pressure, with an application to ELISE vacuum vessel. (Chapter 8)

The methods described on this Doctorate Thesis were developed during the design activities for the next ion beam source to be built in Padova (Italy) and Garching (Germany). Nevertheless, most of these methods can be used also for the analysis and design of generic high heat flux components. In particular, they could be suitable for the ITER experiment, for the demonstrative reactor DEMO and for the future commercial fusion reactors.

Index

- acceleration physics, 21
- Accelerator design, 85
- Accelerator design optimization, 127
- AHP method applied on the accelerator design, 104
- Alignment analysis on accelerator, 99
- Alignment optimization on accelerator, 102
- Analytic Hierarchy Process, 78
- Analytical calculations for buckling, 167
- Application of the methods for vacuum vessel design and verifications, 179
- Arc driven ion sources, 15
- ASME Codes, 157
- ASME rules for buckling prevention, 165
- Back plate cooling system, 58
- Back-streaming ions, 55
- BATMAN ion source, 21, 33
- Beam optics calculations, 17
- Beamlets Steering, 18
- Bohdansky formula, 62
- Boundary conditions on Extraction Grid, 139
- Boundary conditions on Plasma Grid, 95
- Buckling, 164
- Buckling verification for vacuum vessels, 164
- Cesium seeding, 16
- CHT coefficient calculations on Extraction Grid, 142
- Colebrook-White formula, 91, 121
- Comparison between analytical and numerical models for buckling analysis, 166
- Comparison between buckling models, 175
- Comparison between the CFD models, 119
- Computational Fluid Dynamic analysis, 133
- Convection Heat Transfer (CHT) coefficient, 39, 95, 101, 119, 121, 133
- Convection heat transfer calculation, 90
- copper, 64
- Cyclic Stress-Strain (CSS), 135, 136
- Darcy-Weisbach formula, 91, 97, 121
- Deformation results, 99
- Design by Analysis, 157
- Design by Formula, 157
- Divergence, 18
- Electrodeposited copper, 86, 89, 135, 154
- ELISE test facility, 21, 27, 53
- ELISE vacuum vessel, 179
- Example of the Analytic Hierarchy Process, 80
- Example of the Pugh method, 77
- extraction physics, 21
- Failure modes, 158
- Faraday shields coils, 43
- Fatigue, 135, 151, 154, 164
- Fatigue life evaluation, 151

-
- Fatigue verification for vacuum vessels, 164
 - FEM models description, 89
 - Fluid dynamic results on Extraction Grid, 139
 - Fusion power, 3
 - Heat loads on Extraction Grid, 137
 - Heat loads on Plasma Grid, 94
 - Heating and Current Drive systems, 3
 - Hydraulic results, 95
 - Ion sources based on surface production, 12
 - Ion sources based on volume production, 13
 - IPP, 16, 27, 33
 - ITER, 2, 4, 33
 - Langmuir-Child law, 18
 - Large Helical Device, 55
 - Load controlled buckling, 165
 - MAMuG, 24
 - MANITU ion source, 33
 - Material properties, 89
 - Modeling of the plastic behaviour of the materials, 135
 - Modeling the turbulent flow, 112
 - molybdenum, 64
 - Moody diagram, 112
 - Negative ion acceleration, 19
 - Negative ion extraction, 19
 - Negative ion neutralization, 26
 - Negative ions production, 11
 - Neutral beam generation, 9
 - Neutral Beam Injector for ITER, 4, 28, 33, 53
 - New method for the post-processing of calorimetric measurements, 36
 - Novel approach for the FEM analysis of high-performance cooling system, 121
 - Nuclear fusion, 1
 - Numerical analyses on Faraday shields, 46
 - Numerical calculations for buckling with irregular mesh, 174
 - Numerical calculations for buckling with regular mesh, 169
 - Perveance, 18
 - Plasma Grid analyses on segment models, 93
 - Positive and negative ions, 11
 - Post-processing of the calorimetric measurements, 43, 45
 - Power load estimations, 56
 - Pressure drop calculation, 91
 - Primary stress, 160
 - Pugh matrix, 75
 - Pugh method, 74
 - Pugh method applied on the accelerator design, 104, 130
 - RADI ion source, 16, 33, 41
 - Radio Frequency coils, 45
 - Radio Frequency ion sources, 15
 - Ratcheting verification for vacuum vessels, 164
 - Reynolds Stress Model (RSM), 117
 - Robust design, 50
 - Secondary stress, 160
 - Segment models, 92
 - Sensitivity analyses on Extraction Grid, 146
 - Shear-Stress Transport (SST) $k-\omega$ Model, 115
 - Sieder-Tate formula, 90, 101, 121

- SINGAP, 24
- Sputtered layers thickness, 65
- Sputtering analysis, 61
- Sputtering yield, 64
- Standard k- ϵ Model, 114
- Static verifications for vacuum vessels, 161
- Strain controlled buckling, 165
- Stray electron suppression, 23
- Stress categorization, 160
- Stress classification according to ASME,
160
- Stress linearization, 159
- Stripping losses, 24
- Structural analysis, 134
- Structural results, 97
- Surface cooling efficiency evaluation, 147
- Surface production, 12

- Thermal analysis, 134
- Thermal results, 97
- tungsten, 64

- Volume production, 13

- Water pressure calculations on Extrac-
tion Grid, 141