JAERI-Conf 2001-002

ICANS-XV
15th Meeting of the International Collaboration on Advanced Neutron Sources
November 6-9, 2000
Tsukuba, Japan

23.14
Thermal-Hydraulic Study on Cross-Flow Mercury Target

Atsuhiko TERADA*, Masanori KAMINAGA, Katsuhiro HAGA, Hidetaka KINOSHITA and Ryutaro HINO

Japan Atomic Energy Research Institute, Tokai 319-1195, Japan
*E-mail:terada@cat.tokai.jaeri.go.jp

Abstract

In order to remove the high heat density generated in the mercury target effectively under the 1MW proton beam operation, we have proposed the Cross Flow Type (CFT) target using bladed flow distributors. From three dimensional numerical simulations using the general purpose computational fluid dynamics (CFD) code (STAR-CD), it was found that the maximum local temperature rise could be suppressed less than 58.2K under a mercury flow rate of 40m^3/h. This paper presents the current CFD analytical results of the 1MW CFT mercury target.

1.Introduction

The Japan Atomic Energy Research Institute (JAERI) and the High Energy Accelerator Research Organization (KEK) have planned to construct a MW-class neutron scattering facility connected with a high intensity proton accelerator under the High Intensity Accelerator Project. In designing the neutron scattering facility, it is necessary to establish a concept of spallation target working as a MW-class neutron source. We have proposed a cross-flow type (CFT) mercury target with a safety defense container (safety hull).

To optimize the CFT target concept from the viewpoint of thermal-hydraulic performance, three dimensional (3D) numerical simulations have been carried out by using a general purpose computational fluid dynamics (CFD) code. This paper presents the current CFD analytical results of a 1MW CFT mercury target.

2.CFT Target and Analytical Flow

Figure 1 shows a cutaway view of the CFT target for the 1MW proton beam operation. The CFT target consists of an inner vessel (target container) and an outer vessel (safety hull) which will be cooled by heavy water. Between the inner and the outer vessels will be Helium (He) gas. Mercury flows across the proton beam through bladed flow distributors so as to suppress a re-circulation flow, which would make the mercury temperature rise excessively under high heat generation density caused by the spallation reaction between the mercury and the proton beam. The target container, presently under design, will be made of SUS316L, with dimensions, 200-500mm in width (beam window- flange), 100mm in height, and 800mm in effective length.

Figure 2 shows the time-averaged volumetric heat generation (heat density) distribution in the mercury target under 1MW of the proton beam power (3GeV and 0.33mA), assuming a margin of 30% for the original value obtained with the NMTC/JAERI code system[1]. In this analysis, the proton beam has a rectangular shape of 130mm×50mm and a uniform beam power density in the beam shape. The maximum heat density of 314W/cc occurs at around 30mm apart from the beam window.

Figure 3 shows the CFD design flow for optimizing the thermal-hydraulic performance of the CFT mercury target. The CFD design flow is divided into three groups: (1) modeling with detailed drawings, (2) grid-generation, and (3) analysis. Detailed drawings of the CFT target are made with CAD softwares such as the AUTO-CAD or the Pro-Engineering, which are also used for structural analyses of the CFT target as well as the 3D simulation of the target disassembling by using remote handling devices.

3D computational grids are made of the detailed drawings by using the ICEM CFD/CAE code, which is a time consuming job. Then, 3D computational grids are transfered into the CFD code, the STAR-CD, which is a numerical solver for analyzing the three dimensional steady-state velocity, pressure and temperature distributions coupled with the heat density distribution shown in Fig.2. By using the FIELDVIEW and TECPLOT, analytical results obtained with the STAR-CD, which are digital data, are transformed into visible figures such as vector map, contours, etc.

3. Analytical Results

Figure 4 shows the history of the CFT target concepts for the 5MW operation which is our future goal. As for the first CFT target concept with wing type flow distributors, it was very difficult to distribute mercury near the beam window, which caused excessive temperature rise in mercury due to the heat density distribution shown in Fig.2. Based on these results, the second concept using bladed flow distributors (extended wing type blow distributors) was made. Through several modifications, thermal-hydraulic performance of this concept has been almost optimized under the 5MW operation.

In the High-Intensity Proton Accelerator Project (JAERI/KEK Joint Project), since the initial proton beam power was set at 1MW, a simplified concept of the CFT target has been made on the basis of the 5MW CFT target concept. Figure 5 shows computational grids and boundary conditions of the 1MW CFT target. Computational domain is the upper half of the target considering the structural symmetry. The total number of the mesh (grid) is 400,000.

From the viewpoint of conservative estimation on temperature distributions, the outer surface of the target container was assumed to be adiabatic, so that the heat generated in the beam window was assumed to be transferred to mercury. Then, the heat flux on the beam window of 2.5mm thick was 0.5MW/m^2 using the heat density of 212W/cm^3 shown in Fig.2. Analyses were carried out under the inlet temperature of 323K and the inlet mercury velocity of 1m/s (mercury flow rate of 40m^3/h) , which would be a reference condition to optimize the inlet condition.

Table 1 shows the analytical conditions. Analyses were carried out under steady-state incompressible turbulent flow conditions. The two-equation (standard) k- ε model associated with the log-low-based wall function was used to analyze flow fields, and the zero-equation model to analyze heat transfer by using the experience value of the turbulent Prandtl number (Prt). In the heat transfer analyses, Prt of 1.5 was given on the basis of experimental results obtained by a mercury test loop of JAERI [2]. To improve the precision of the analytical results, the SIMPLE method was used as algorithm, and 2nd-order self-filtered central differencing scheme to non-liner terms.

Figure 6 shows a velocity contour and temperature distribution on the center plane of the target. Analytical results shown in Fig.6 were obtained under the structure with 10mm thick bladed flow distributors. As seen in the velocity contour, a large re-circulation flow was clearly evident in the rear of the target. The maximum local temperature (hot spot) occurs in the re-circulation flow, whose temperature rise is only 58.2K against the inlet mercury temperature of the 323K, as seen in the temperature distribution. This temperature rise is far below the mercury saturation temperature of 629K. On the other hand, mercury temperature near the beam window where the heat density distribution has its peak is suppressed below 381.2K, which shows that mercury is distributed efficiently by the bladed flow distributors. The pressure drop between the inlet and the outlet of the target is a rather low value, 21.5kPa, which would indicate the possibility of making a compact mercury pump. The maximum velocity of the 2.4m/s appears at the outlet side wall, so that countermeasures against erosion should be considered. In the present design, the side wall thickness is more than 10mm considering erosion and structural strength.

Figure 7 shows the temperature distributions obtained with 7.5mm and 5mm thick blade structures. The maximum local temperature appears in a wake region near the beam window, which is 398.5K for 7.5mm thick blades and 403.8K for 5mm thick blades. Based on these results, a rather large value of 1mm might be accepted as the tolerance of the blade thickness.

4.Concluding Remarks

Thermal-hydraulic simulations by using the CFD code, STAR-CD, have been carried out from the viewpoint of effective removal of the high heat density generated in the mercury target under the 1MW proton beam operation. As a result, a concept of the cross flow type target with bladed flow distributors was found to be feasible. Since the Proton beam profile injected into the target is not fixed yet, further optimization of the flow structure such as bladed flow distributors will be carried out after we obtain a realistic proton beam profile.

References

[1] M.Teshigawara (JAERI), private communication.

[2] H.Kinoshita et al, "Experimental Study on Mercury Target and Proton Beam Window", ICANS-XV, Tsukuba, (2000).

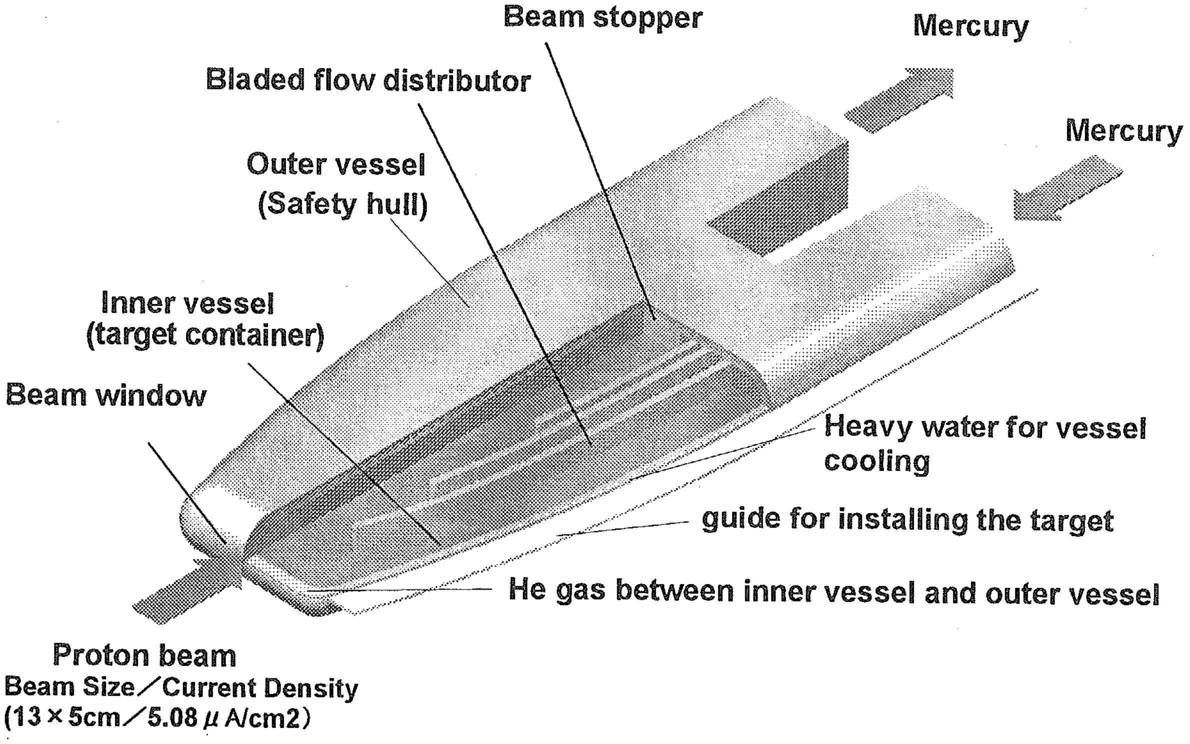


Fig.1 Cutaway view of the CFT mercury target for 1MW operation

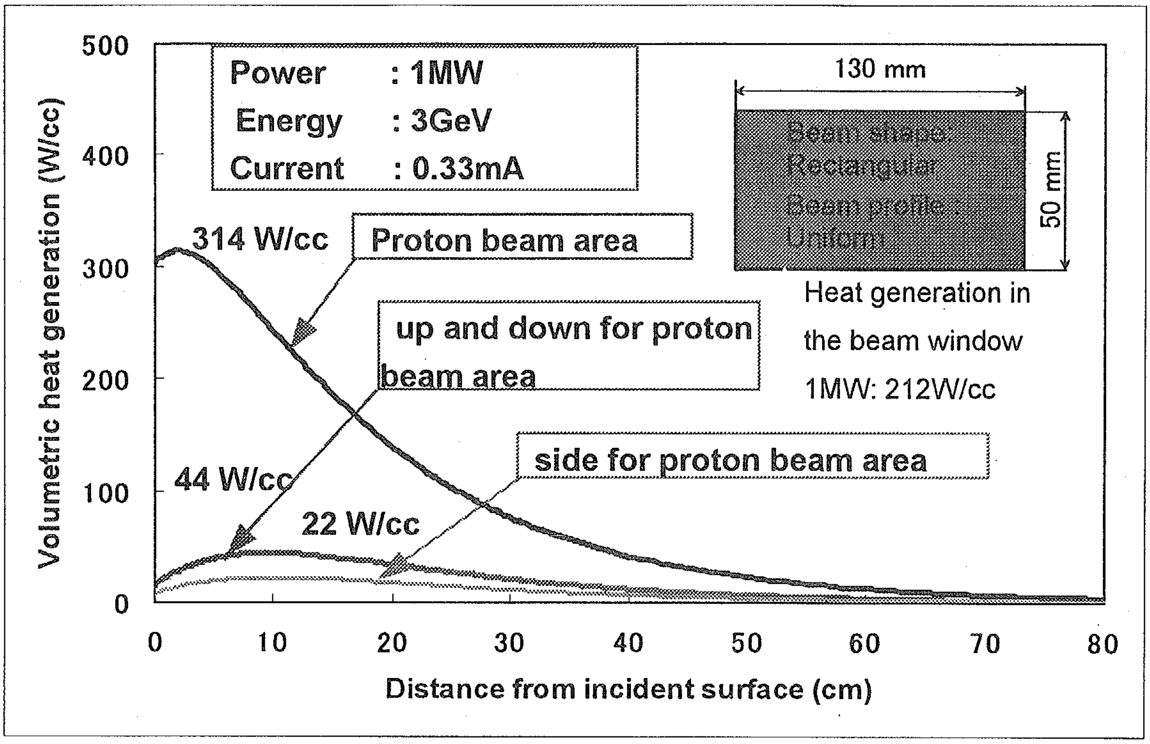


Fig.2 Axial heat density distribution of the mercury target (Time averaged)

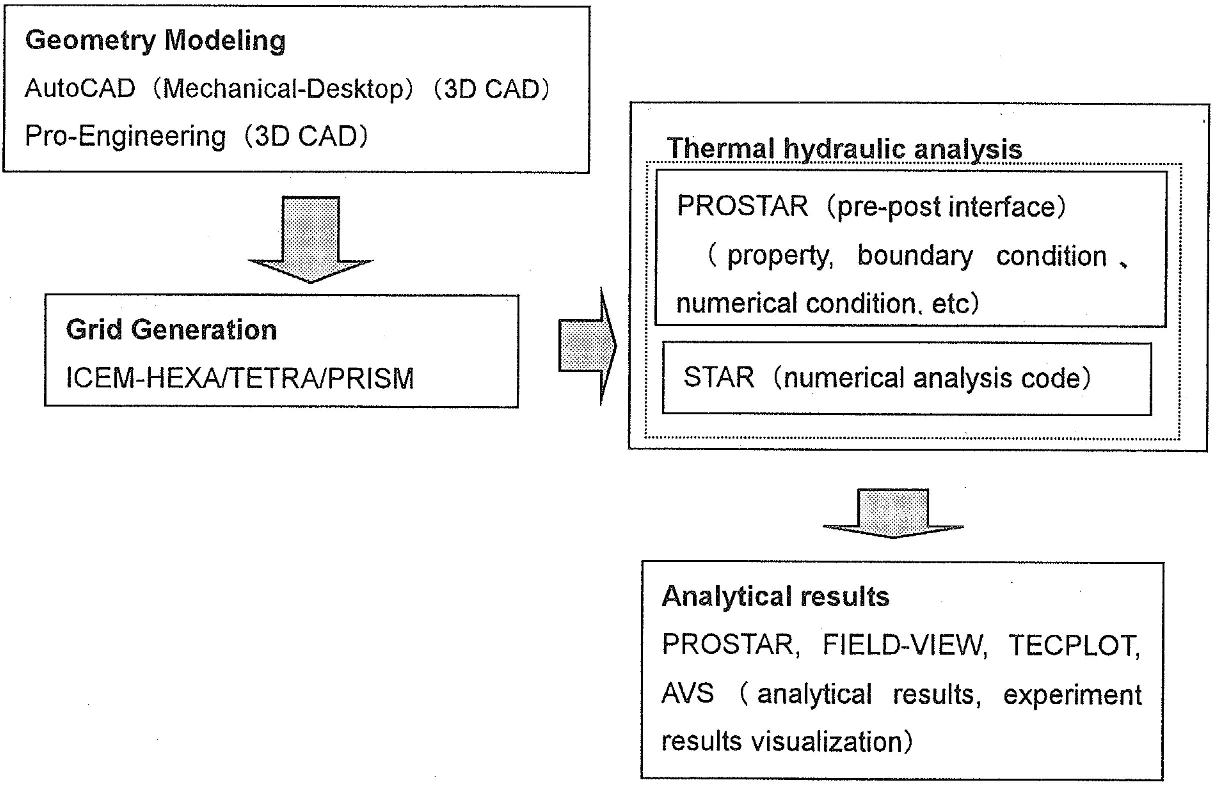
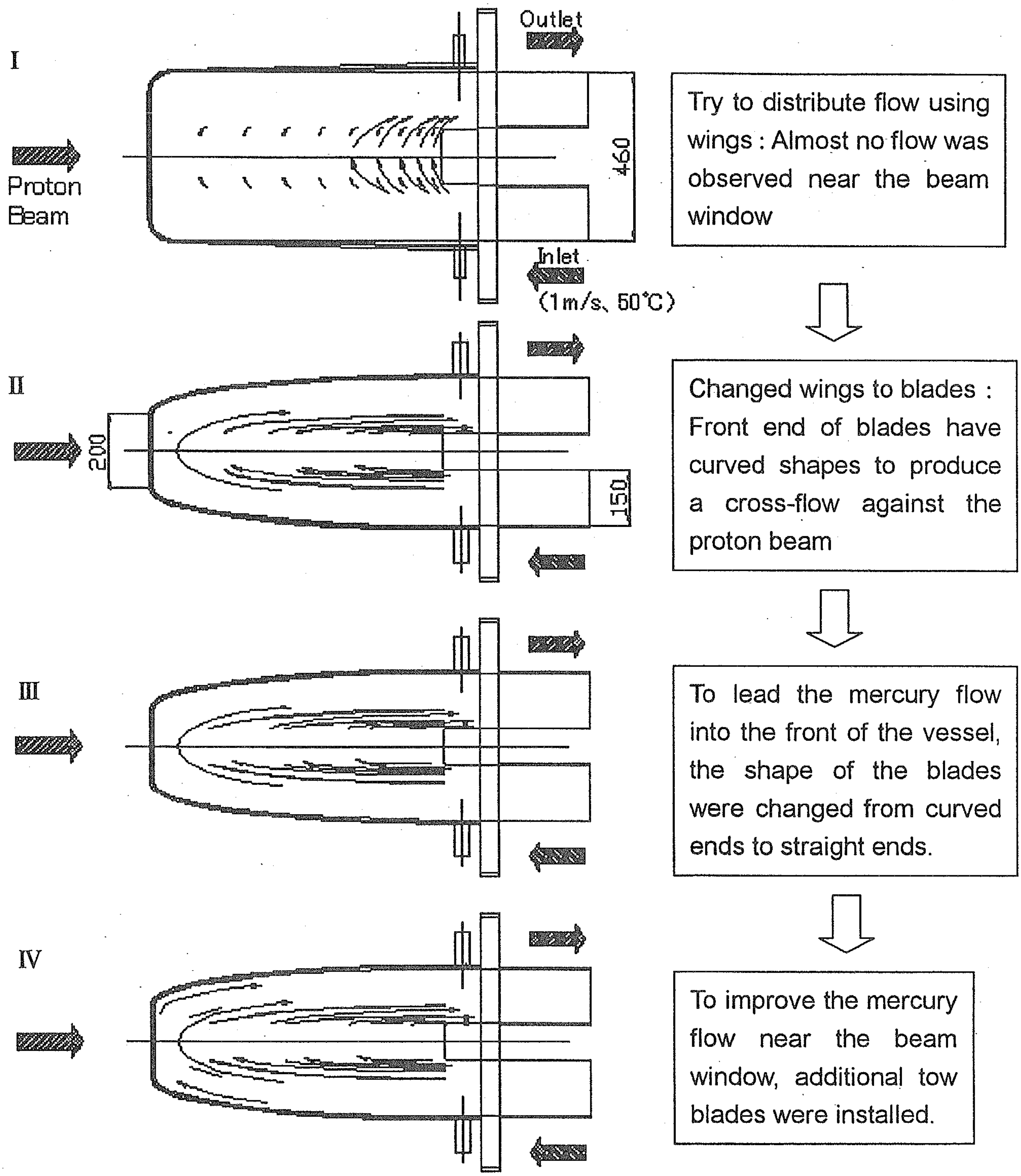


Fig.3 CFD design flow for mercury target



Try to distribute flow using wings : Almost no flow was observed near the beam window

Changed wings to blades : Front end of blades have curved shapes to produce a cross-flow against the proton beam

To lead the mercury flow into the front of the vessel, the shape of the blades were changed from curved ends to straight ends.

To improve the mercury flow near the beam window, additional tow blades were installed.

Fig.4 History of the CFT mercury target concepts for the 5MW operation

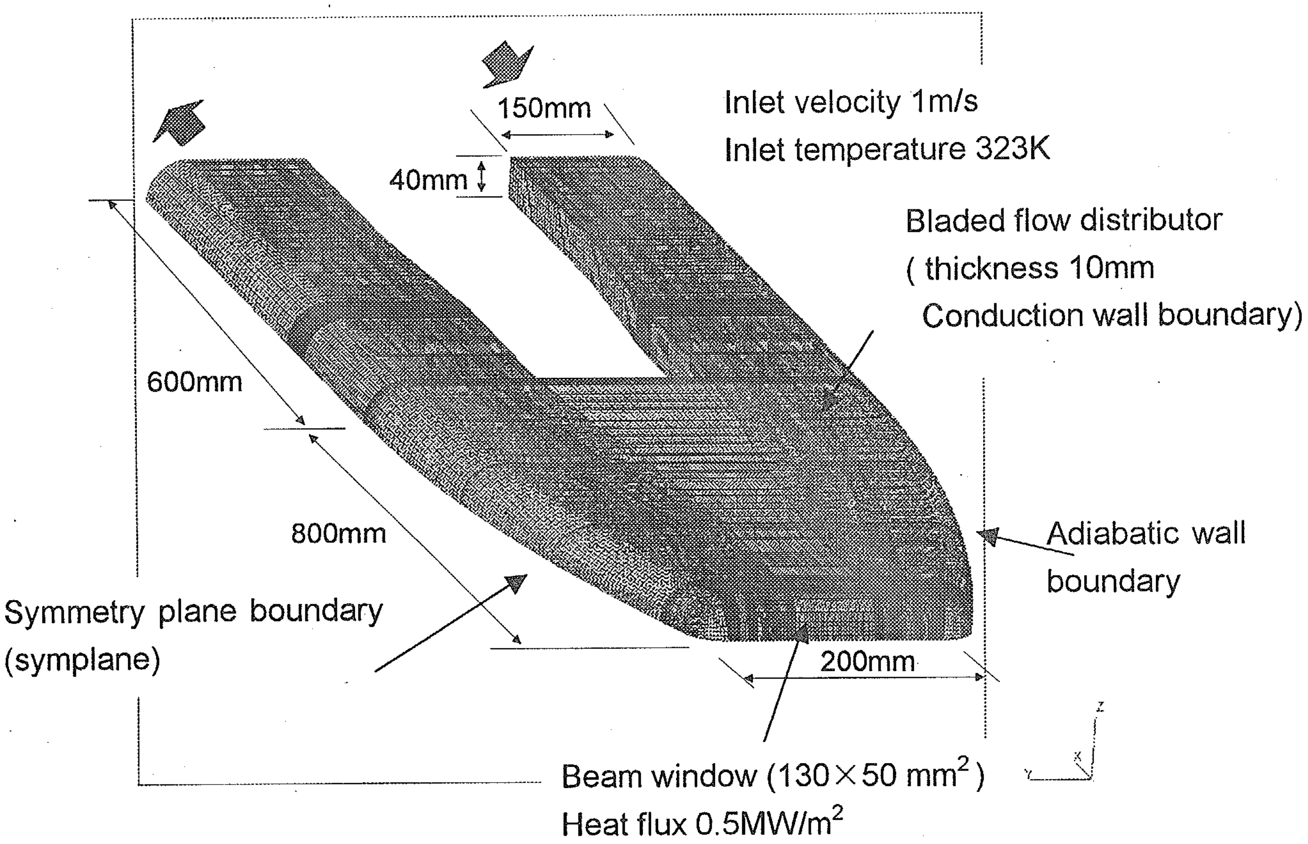


Fig.5 Computational grids

Table.1 analytical condition

Condition	Steady-state, Incompressible, Viscosity Flow
Discretisation	Finite Volume Method
Algorithm	SIMPLE
Non-liner term	self-filtered central differencing scheme (2nd order)
Turbulence model	Standard k-ε model associated with log-low-based wall function
Heat-transfer model	0-equation heat-transfer model Turbulent Prandtl number 1.5

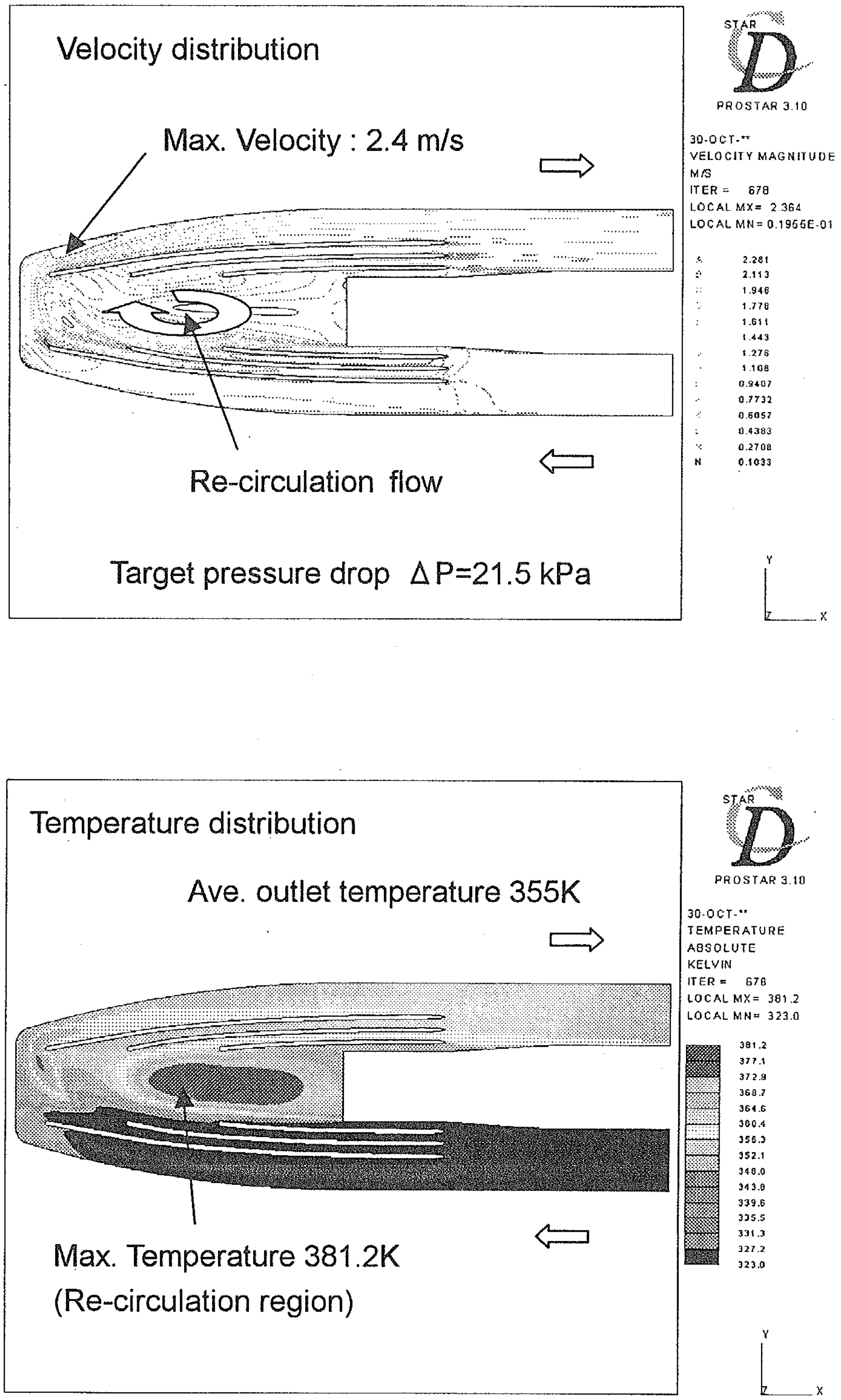


Fig.6 Velocity contours and temperature distribution on symplane of the target

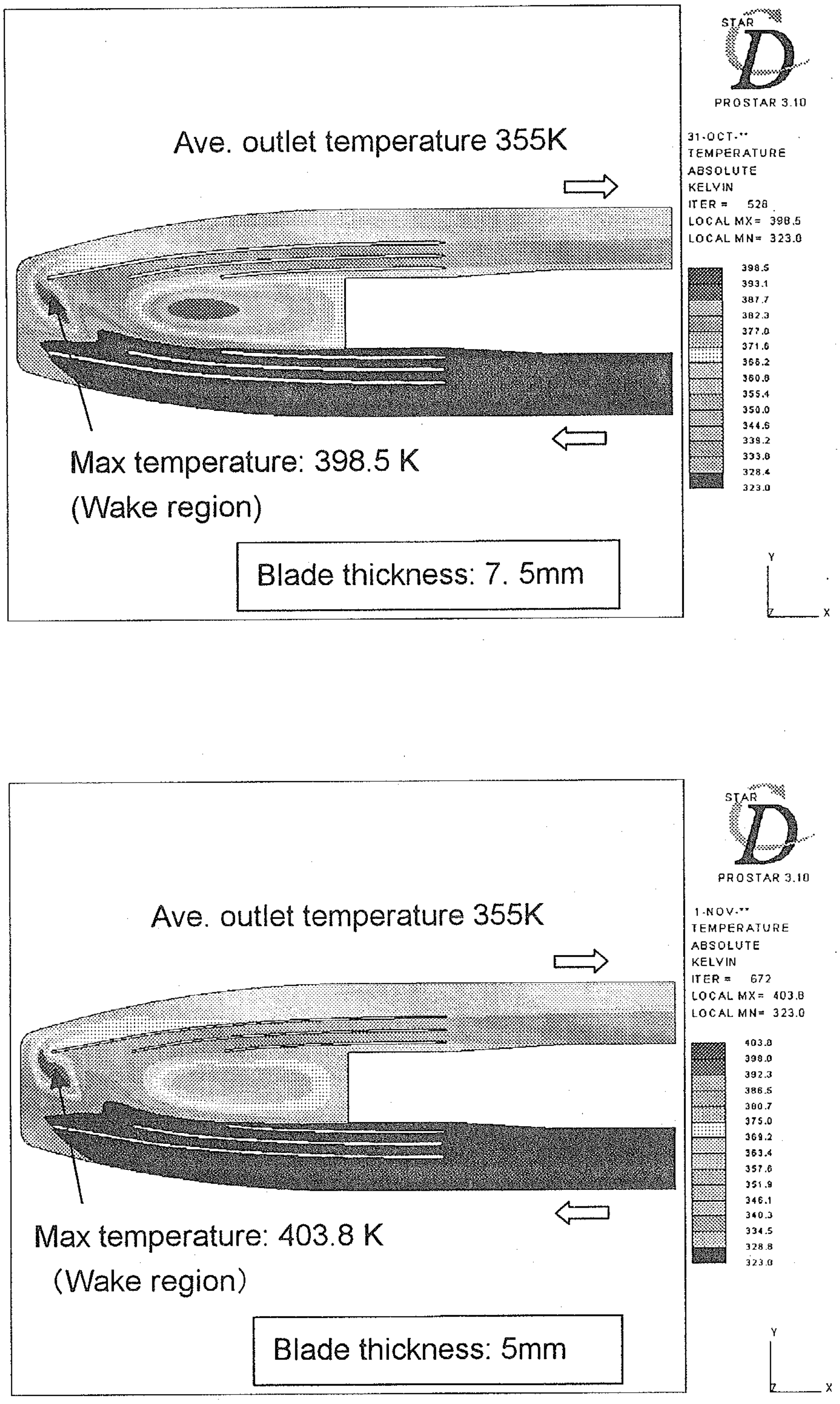


Fig.7 Temperature distributions on symplane of the target