Thermal Hydraulic Analysis of LANL/IPPE/EDO-GP 1MW LBE Target

by Xiaoyi He^{1,2}, William Gregory³, and Ning Li¹¹ LANSCE-ATW² T-13³ ESA-DE

Abstract

We have carried out numerical simulations of the thermal hydraulic behavior of the LANL/IPPE/EDO-GP neutron spallation to be tested in LANSCE beam in year 2,000. The LANL¹/IPPE²/EDO-GP³ target design, along with energy deposition provided by the Los Alamos Meson Physics Facility (LAMPF) accelerator beam, has been analyzed with two commercial computational fluid dynamic (CFD) computer codes at Los Alamos. The LAMPF proton beam deposits energy in the target window diaphragm as well as the liquid metal coolant target. The computer codes were used to determine the maximum temperature in the spallation target diaphragm. The computational results from the two CFD codes are in general agreement and are consistent with the preliminary IPPE analysis. Limited studies were also performed to investigate ways to enhance cooling of the target by directing more flow through the center of the target. The goal of this latter part of the study is to provide guidance for the future design of a spallation target for the ATW project.

1. INTRODUCTION

The Accelerator-driven Transmutation of Waste (ATW) concept has been proposed by the United States and other countries to essentially eliminate plutonium, higher actinides, and environmentally hazardous fission products. One of the key components in the ATW concept is a target which, via spallations, produces neutrons to drive the burning of nuclear wastes. Since significant heat is generated during fissioning of the waste actinides, an efficient heat removal system is necessary. Liquid lead-bismuth eutectic (LBE) is an efficient coolant as well as a good spallation target for production of neutrons. The LBE coolant technology has been successfully used in Russian submarine's nuclear reactors.

The International Science and Technology Center (ISTC) has funded the Institute of Physics and Power Engineering (IPPE) of Russian to develop a preliminary design study of a pilot target that incorporates Russian LBE technology into the ATW concept. The final design target is to be fabricated by Experiment and Design Organization-"Gidropress" (EDO-GP) of Russia and tested with a beam-on condition at the Los Alamos National Laboratory (LANL) in three years. The pilot study by IPPE proposed four preconceptual designs of the spallation target [1]. These four designs are shown in Fig. 1. Based on their thermal hydraulic analysis, IPPE recommended target number 3 for further study because it appeared to give the best cooling to the target diaphragm window.

^a Los Alamos National Lab, USA^b Institute of Physics and Power Engineering, Russian^c Experiment and Design Organization-"Gidropress", Russian

MASTER

DISCLAIMER

This report was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government nor any agency thereof, nor any of their employees, makes any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or any agency thereof. The views and opinions of authors expressed herein do not necessarily state or reflect those of the United States Government or any agency thereof.

DISCLAIMER

**Portions of this document may be illegible
electronic image products. Images are
produced from the best available original
document.**

The present study has two objectives. The first objective is to confirm the thermal hydraulic analysis by IPPE. This is because the DUPT code used in the IPPE analysis does not account for the convective heat transfer and empirical formulas were used to compensate the important effect of the convective heat transfer. This leaves some uncertainties in the temperature predictions in the IPPE report. The accuracy of this estimation needs be confirmed. The second objective of this study is to extend the IPPE analysis to include a wider parametric range. This will aid future thermal hydraulic designs of the ATW target.

The rest of this report is organized as following. Section 2 describes the parameters of the studied target, such as, geometry, flow condition, and energy deposition. Section 3 briefly describes the Computational Fluid Dynamic (CFD) codes used in this study. Section 4 presents the confirmatory studies by the commercial CFD codes of the thermal hydraulic analysis results obtained by IPPE. Section 5 presents parametric studies which aim to help future design of the spallation target. Section 6 discusses the results and concludes the report.

2. TARGET PARAMETERS

2.1 Geometry

The geometries of the preconceptual designs of the LANL/IPPE spallation targets are shown in Fig. 1. The main component of all the targets is a cylindrical tube with flowing liquid LBE. The target window is made of a cone-spherical steel diaphragm with a thickness of 1.5 mm. Cooling of the target window is one of the major concerns in the target design.

This study will mainly focus on target version 3, which was shown to offer the best cooling of the window out of four conceptual designs in the preliminary study by IPPE. This target has a diffuser plate placed immediately downstream (2.5 mm) from the hot window. The plate converges the flow towards the center of the target window to reduce window temperature. However, this configuration also allows for possible flow separation and consequent recirculation behind the plate. Recirculation of coolant in regions of intense energy deposition could lead to a hot region. To reduce possible recirculation, a total of 48 holes are punctuated in the plate as shown in Fig. 2.

2.2 Flow Conditions

As shown in Fig. 1, liquid LBE flows in from the outer annulus and flows out from the inner channel (target volume). A steady flow rate through the target was chosen to be $14 \text{ m}^3/\text{h}$, which yields approximately a 120°C temperature rise for a 1 MW LAMPF beam condition. Based on the target dimensions and the viscosity of the liquid LBE in the operating temperature range, this flow rate yields a Reynolds number of approximately 10^6 . At this high Reynolds number, it is reasonable to assume that the flow is fully turbulent. In both FIDAP and CFX simulations, we choose to use the $k-\epsilon$ model for the turbulent flow simulation.

2.3 Energy Deposition

The first stage of the ATW project is to demonstrate the feasibility of the LBE target and that no fissioning of nuclear waste is involved. The only energy deposition is due to the interaction of charged particles when the proton beam hits the target. The present study only focuses on this scenario.

The energy deposition for the LAMPF beam condition was studied in the IPPE pilot study [1] and was confirmed by an independent simulation study using the MCNP-LAHET code at LANL. The energy deposition in the coolant satisfies:

$$q(r,z) = 1600 \exp(-z/17 - r^2/(2.68+0.033z)) \text{ W/cm}^3, \quad (1)$$

where r is the radial position and z is the longitudinal distance to the target window. In the steel diaphragm, the energy deposition satisfies:

$$q(r,z) = 2100 \exp(-r^2/3.125) \text{ W/cm}^3. \quad (2)$$

3. THERMAL HYDRAULIC CODES

3.1 FIDAP

One of the CFD packages used in this study for thermal hydraulic analysis of the LANL/IPPE/EDO-GP spallation target is the FIDAP computer code (version 7.62, marketed by Fluent Inc.) [2]. FIDAP is a general purpose computer program that uses the finite element method (FEM) to simulate many classes of fluid flows and heat transfer problems. This study only uses its functions of thermal and hydraulic analyses.

In FEM, a computational domain is divided into a series of small areas, called finite elements. The partial difference conservation equations governing the fluid flow and heat transfer are subsequently discretized into a series of algebraic equations in each element. The velocities, pressure, and temperature are calculated by solving these algebraic equations using sophisticated numerical schemes. In this study, the computational domain is divided into 8052 finite elements with a total of 7470 nodal points (Fig. 3). Dense meshes are placed near the target window to capture the detailed flow and temperature information there. Simulations on a finer mesh gave almost the same results (data not shown), which implies this mesh set-up is fine enough for our thermal hydraulic analysis.

FIDAP provides several strategies to solve the nonlinear system of equations. In this study, we choose to use the "segregated" algorithm with pressure projection. The segregated solver solves sequentially and separately for each active degree of freedom in the problem. As a result, the disk storage requirement is greatly reduced. The simulation is carried out in two steps. The flow field is first solved independently, and then the temperature field is solved using the velocity results from the flow simulation.

Two user's subroutines (FORTRAN) were used in the simulation, one to input the energy deposition and the other to input the porosity of the diffuser plate.

3.2 CFX

Another CFD package used in this study is the CFX code, version 4.0, marketed by AEA Technology [3]. CFX is a CFD computer code that solves the simultaneous numerical solution of the continuity, momentum, and energy equations for a flow field geometry with specified boundary conditions. The general mathematical approach relies on a finite volume method based on the SIMPLEC mathematical algorithm to solve these equations. Although the CFX code is capable of handling fully three-dimensional geometries in Cartesian coordinates, the analyses made use of the two-dimensional axi-symmetric capability of the code with cylindrical coordinates. In addition to the axi-symmetric modeling, a body fitted grid was also used. Turbulent flow with the standard k- ϵ model was used with temperature dependent LBE density and thermal conductivity. Because of the non-constant density, the code was run in compressible mode.

The user FORTRAN routines in the code were used to implement distributed heat source to the diaphragm and coolant. In addition, FORTRAN routines were used to implement the temperature-dependent density and thermal conductivity. The heat transfer coefficient FORTRAN subroutine was made flexible so that the IPPE reported formulations for the Nusselt number could be used if desired. A capability to use a multiplier to modify the CFX calculated heat transfer coefficient was also added. However, the results reported here used the CFX calculated values for the heat transfer coefficients.

All of the CFX calculations were carried out on an SGI Indigo workstation with a 256 Mb RAM, 250 MHz IP22 processor, MIPS R4010 Rev 0 floating point chip, and a MIPS R4400 Rev 6.0 processor chip.

The computational domain was divided into 63 blocks. The blocks were further divided into 10548 computational cells. The computational mesh used in the CFX analysis is shown in Fig. 4.

4. RESULTS

4.1 FIDAP Results

4.1.1 Modeling Assumptions

Several constitutive laws for porous media flow are available in FIDAP. We choose to use the Forchheimer law:

$$\nabla p = \rho b |\mathbf{u}| \mathbf{u} / k_i, \quad (3)$$

where ∇p is the pressure gradient, ρ is the fluid density, $\mathbf{u}$ is the superficial velocity, k_i is the permeability, and b is a coefficient. Since the perforations in the diffuser plate are oriented horizontally, the permeability in the radial direction is set to be zero. The use of the Forchheimer law for this study can be justified by classic hydrodynamic experiments which show that, at $Re=10^6$, the pressure loss of flows through a sharp-edged orifice satisfies [4]:

$$\Delta p = 1/2 \rho u^2 (1-\alpha^2) / (C\alpha^2) \quad (4)$$

where α is the porosity of the diffuser and C is the coefficient of discharge which is approximately 0.62 for the flow condition in this study.

In the FIDAP simulations, we assume the porosity satisfies:

$$\alpha = \alpha_0 r_0 / r. \quad (5)$$

where α_0 is the mean porosity of the diffuser plate and $r_0 = 3.25$ is the radius at middle of the diffuser plate. In the origin design by IPPE, the mean porosity of the diffuser plate is 0.15.

4.1.2 Flow Field

The typical flow field in the LANL/IPPE target as calculated by FIDAP is shown in Fig. 5 using a streamline plot. As shown, the flow field is relatively uniform except near the target window and behind the diffuser. The diffuser successfully converts a significant amount of coolant towards the center of the target window. The flow rate through the center hole is 38.1% of the total flow rate (notice that the area ratio of the center hole to the inner channel is only 9.0%).

It is also interesting to notice that, at this porosity, a small recirculation zone just starts to appear behind the diffuser plate. This implies that this porosity falls right in the range where the flow undergoes a transition to a recirculation state. Although not obtained for the present conditions, the recirculation zone can generate an undesirable hot region downstream from the diffuser plate.

The FIDAP calculated pressure distribution in the target is shown in Fig. 6. As expected, the biggest pressure loss (6.2×10^5 dyn/cm² or 9.0 psi) occurs around the diffuser plate. A total of 8.19×10^5 dyn/cm² pressure difference (11.88 psi) is required to drive the flow through the target with a flow rate of 14 m³/h. This pressure difference is well within the power range of the current-used pump (110 psi).

The turbulence kinetic energy is shown in Fig. 7. As shown, the diffuser plate generates quite a high turbulence kinetic energy behind the diffuser. This high kinetic energy helps to mix the hot and cool fluids and enhance the heat transfer in the flow.

4.1.3 Temperature Field

The overall temperature field and the detailed temperature field near the target window, as calculated by FIDAP, are shown in Fig. 8a and Fig. 8b, respectively. As expected, the coolant is heated up as it flows through the target. The temperature increases from 220 °C at the inlet to about 340 °C at the outlet. The biggest temperature variation occurs near the target window and diffuser, while the temperature distributions in the outer channel and downstream inner channel are relatively uniform. Although a slight recirculation zone has been calculated, the temperature effect is not significant.

To illustrate in detail the temperature variation near the target window, we plotted the temperature distribution along the axis (Fig. 9). The solid diaphragm has a much higher

temperature than the coolant. The maximum temperature (491 °C) occurs at the vacuum side (exterior wall) of the diaphragm and the temperature at the coolant side (inner wall) of the diaphragm is 390 °C. The temperature in the diaphragm has an approximately parabolic shape. That is because the energy deposition along the axis in the diaphragm is essentially constant.

4.1.4 Convective Heat Transfer Coefficient

One of the uncertainties in the thermal hydraulic analysis by IPPE is the convective heat transfer coefficient, h , from solid to liquid. Using two empirical formulas, they estimated this coefficient within a range of 1.569 ~ 2.536 W/cm² °C. To check the accuracy of the convective heat transfer coefficient used in IPPE's analysis, we calculate it from our simulation results:

$$h = k/(T - T_0) dT/dn, \quad (6)$$

where k is the heat conduction coefficient and T_0 is the temperature in main stream. Notice that the choice of T_0 is very difficult when energy is deposited in the coolant. To avoid this difficulty, we reran the temperature simulation without the heat deposition in the coolant. In this case, the reference temperature can be taken as the inlet temperature.

Figure 10 shows the variation of the convective heat transfer coefficient along the cone-sphere wall. As shown, h is about 2.4 W/cm² °C in most part of the cone-sphere wall. The high h at the beginning of the channel is due to the direct impingement of the coolant around the septum. The increase of h downstream is due to the narrowing of the channel. The heat transfer coefficient sharply decreases to 2.1 at the window tip, which is due to the decrease of coolant velocity at the window tip.

4.2 CFX Results

4.2.1 Modeling Assumptions

The inlet and outlet flow pipes were neglected in order to focus on the diaphragm target region. In addition, certain assumptions were made to simulate the presence of the porous plate. In the CFX calculation, the porous plate was modeled as a porous medium with the body forces in the momentum equation modified to account for the resistance to flow through the holes in the plate. The formulation used to calculate the resistance to flow through the plate was:

$$K = (P_1 - P_2)/(1/2\rho U^2) \quad (7)$$

where

K = loss coefficient,
 P_1 = upstream pressure (Pa),
 P_2 = downstream pressure (Pa),
 ρ = fluid density (kg/m³), and
 U = face velocity (m/s).

Two references were used to calculate the coefficient K . One reference was in Idelchik's "Handbook on Hydraulic Resistance" for a porous plate [5]. In this reference tables can be used to calculate the values for K . The second reference is a book entitled "Applied Fluid

Dynamics Handbook" by Blevins [6]. In this reference the coefficient K is specified as a function of the porosity, α as given below:

$$K = [1/2(1 - \alpha) + (1 - \alpha^2)]/(\alpha)^2, \quad \text{for } D < l < 50D \quad (8)$$

or

$$K = [1/2(1 - \alpha) + (1 - \alpha^2) + f l/D]/(\alpha^2) \quad \text{for } l > 50D. \quad (9)$$

where

α = porosity,
 D = characteristic diameter (m),
 l = plate thickness (m), and
 f = friction factor.

Considering that the plate thickness is 50 mm and diameter of the hole is 53 mm, Eq. (8) was used. Table I lists the values obtained for the loss coefficient K from the two references. Note that the value of K varies in the radial direction. Although the number of holes in the plate is the same at selected radii, the plate area changes and therefore the plate porosity changes.

Table I

Porosity (α)	K	
	Idelchik	Blevins
0.211	26.5	30.3
0.182	43.3	41.5
0.142	77.4	69.9
0.103	148.8	135.5

The results for the loss coefficient, K, are essentially the same for both references. This gives confidence in the values to be used in the CFX calculation for accounting for the porous medium pressure loss through the plate.

The friction loss through the plate is calculated by the CFX code using a resistance speed factor. Noting that CFX uses velocities within the porous medium and not the face velocity, U, the resistance speed factor relationship is written as

$$RSF = \alpha^2 K p / 2 l \quad (10)$$

where

RSF = resistance speed factor.

Using the values of K given by Idelchik, the resistance speed factor going from the inside of the plate to the outer edge gives

RSF1 = 1.23 E06,
 RSF2 = 1.49 E06,
 RSF3 = 1.52 E06, and
 RSF4 = 1.64 E06.

These coefficients were modified based on a separate modeling of the porous plate with the holes and a simulation of the plate with porous media. It was necessary to decrease the resistance speed factors by 20% to reduce the center hole velocity to be close to the actual plate with holes simulation.

4.2.1 Flow Field

The flow field that results from the CFX calculation is shown in Figs. 11, 12, and 13. Figure 11 shows velocity vectors in the coolant for the overall target. Notice that the separator between the inlet and the channel near the diaphragm is clearly shown. As shown on the color scale, the maximum velocity in the porous media varies between 1.5 and 2.7 m/s. The center hole velocity downstream of the plate has a value of 3.6 m/s. Figure 12 shows an enlarged view of the flow field near the target window. This figure shows the flow through the porous plate as well as the flow through the central hole. A distribution of velocity magnitude is shown in Fig. 12 with the higher velocity near the outer edge of the porous plate and smaller velocities near the center hole where the porosity is higher. Approximately 32 % of the flow goes through the central hole. Figure 13 shows a contour plot of velocity. Notice the appearance of a recirculation zone downstream from the diffuser plate and near the wall of the annulus.

4.2.2 Temperature Field

Figs. 14 and 15 show the temperature field calculated by CFX. Figure 14 shows the overall temperature field in the target. The coolant temperature distributions enters with an inlet temperature of 220 °C and increases to 399 °C at the diaphragm tip. The temperature then decreases to 347.8 °C at the outlet. Therefore the temperature difference across the target is 127.8 °C. Figure 15 shows an enlarged view of the temperature field in the coolant and diaphragm with the computational mesh superimposed on the temperature field. Notice that there are four computational cells in the diaphragm wall thickness of 1.5 mm. The temperature distribution in the diaphragm wall at the tip ranges from 534 °C to approximately 432 °C.

4.2.3 Pressure Field

Figure 16 shows the pressure field in the target region as calculated by CFX. As shown in the scale in Fig. 16, the pressure varies from 5.0433×10^4 to -1.7448×10^4 Pa. The differences in these two pressures with a conversion of units gives a pressure drop across the diffuser plate of 9.84 psi. The pressure drop is slightly higher than that calculated by FIDAP due to the CFX approximation of the diffuser which has more flow through the plate.

4.3 FIDAP, CFX, and IPPE comparison

A comparison of the exterior and inner diaphragm wall temperatures and the maximum diaphragm temperatures from the computer codes used by IPPE and the results obtained by FIDAP and CFX are shown in Table II.

Table II
Diaphragm temperatures in °C for IPPE, FIDAP, and CFX Calculations

DUPT code (IPPE)		FIDAP	CFX
without convection	with convection		

Exterior Wall	610	465-533	491	534
Inner Wall	556	376-445	390	432

The maximum diaphragm window temperatures are shown in Table II to be 465-533 °C, 491 °C, and 534 °C for IPPE, FIDAP, and CFX respectively. The temperature changes in the diaphragm wall are 88 °C for the IPPE 2-D calculations and the CFX and FIDAP calculations are 102 °C and 101 °C respectively.

While the results calculated by FIDAP and CFX agree in general, there are differences that should be pointed out. For the maximum temperature in the diaphragm, CFX gives a more conservative higher temperature (534 °C compared to 491 °C). In addition, the pressure difference across the diffuser is higher for the CFX calculation than that for FIDAP. This difference could well be due to the different values used for the resistance of the porous media approximation in the two codes. The two codes compare well for the temperature difference across the wall of the diaphragm. Another area of discrepancy is in the flow field results. While both codes show the possibility of a recirculation zone downstream of the porous plate, the FIDAP results show a larger zone as compared to CFX. Again, this may be due to the different values used for resistance of the porous media. As shown in section 5.2, this discrepancy between FIDAP and CFX can be greatly reduced if both codes use the same value for the resistance of the porous media.

4.4 Version 1 Target Analysis

A model of version 1 was also constructed to check IPPE's conclusion that version 3 with the porous plate yielded lower target window temperatures. This concept uses a constriction near the spherical part of the target window to force the coolant flow toward the tip hot spot. The flow and temperature fields as calculated by the CFX code are shown in Figs. 17-20.

4.4.1 Flow Field

Figures 17 and 18 show the velocity field in the target as calculated by CFX. Figure 17 shows the overall velocity distribution in the coolant as it flows around the target. The maximum velocity calculated is 1.8 m/s. Notice that the maximum velocity occurs at the tip of the constriction wall. Figure 18 shows an enlarged view of the velocity vectors and it is clear that there is a recirculation region around the tip of the diaphragm window and therefore causes the tip temperature (as shown in the section 4.4.2) to be higher than that obtained for target version 3.

4.4.2 Temperature Field

The temperature field from the CFX simulation of version 1 are shown in Figs. 19 and 20. Figure 19 shows the overall temperature distribution in the target. Notice the constriction that expands near the tip of the diaphragm. As expected, Fig. 20 shows an increase in temperature in the diaphragm wall. The maximum temperature is 596.7 °C or an increase of approximately 68 °C larger than the tip temperature in version 3. As noted in the velocity field, this increase in temperature is caused by the recirculation region near the tip.

5. POROSITY STUDY AND RESULTS

5.1 Analysis objective

As discussed in earlier sections, the diffuser plate plays an important role in controlling the temperature distribution in the target. If the diffuser plate is too permeable or nonexistent as in target version 1, it can not converge enough coolant towards the center of the target and the target window may have a high temperature. On the other hand, if the diffuser plate is too compact, a flow separation may be generated behind the diffuser and a high temperature may occur in the recirculation zone. To quantify the effect of the plate porosity on flow and temperature fields, we used FIDAP to perform simulations with the mean porosity varying from 0.05 to 0.25.

5.2 Results

Shown in Fig. 21 are streamlines in the target for different plate porosities. No flow separation can be observed when the plate mean porosity is larger than 0.15. With a further decrease in the porosity, a recirculation zone begins to appear behind the baffle. The recirculation zone has a very large size when the plate mean porosity reduces to 0.05.

The temperature distributions for different plate porosities are shown in Fig. 22. When there is no flow separation ($\alpha > 0.15$), the temperature behind the diffuser gradually increases moving downstream. When the flow separation occurs, a temperature peak appears in the recirculation zone, as indicated by the close contours of temperatures. For the LAMPF beam condition, the energy deposition is concentrated along the axis and this temperature peak in the recirculation zone is not significantly high. However, for other beam conditions with a spread energy deposition (which is likely the case for future accelerator used in ATW), the temperature in the recirculation zone may become very high. As an example, we calculated the temperature distribution under the PSI-II beam condition with a plate mean porosity of 0.05. The energy deposition of the PSI-II beam is:

$$q(r,z) = 470 \exp(-0.0025 z - 0.048 r^2) \text{ W/cm}^3, \quad (11)$$

in coolant, and

$$q(r,z) = 430 \exp(-r^2/19) \text{ W/cm}^3, \quad (12)$$

in the steel diaphragm.

As shown in Fig 23, with the PSI-II beam condition, the peak temperature in the recirculation zone is much higher than the maximum temperature in diaphragm (490 °C vs 341 °C). This implies that additional attention is necessary to reduce the recirculation zone for a spread beam condition.

Fig. 24a shows the variations of the maximum temperature in the target window (LAMPF beam) and flow rate through the center hole with the plate porosity. With the plate porosity increases from 0.05 to 0.25, the flow through the center hole decreases by about 50% and, in the mean time, the maximum temperature in the target window increases from 430 °C to 530 °C. Since our simulations cover a wide range of the porosity, this target design appears to be satisfactory from the thermal hydraulic point of view, at least for the operation conditions listed above.

Fig. 24b shows the same results of the flow rate and maximum temperature, but the x-axis is replaced with the loss coefficient K (see Eq.(7)). With this x-axis, we can include the

result from the CFX simulation for direct comparison. As shown, both the maximum temperature and the flow rate through the center hole from the CFX simulation fall right into the estimated range by FIDAP simulations. This fact implies that, once the resistance of the porous media is correctly modeled, simulations by FIDAP and CFX give consistent thermal hydraulic results.

6. DISCUSSION AND CONCLUSIONS

We have carried out numerical simulations of the thermal hydraulics for the LANL/IPPE/EDO-GP spallation target, version 3, with LAMPF beam conditions. Simulation results have been obtained using two commercial CFD codes, FIDAP and CFX. The results are in general agreement although the CFX code predicts more conservative results. The results from the two codes are generally consistent with the preliminary analysis obtained by the IPPE. With a flow rate of 14 m³/h, the temperature in the target is within the desired operation range (< 600 °C). A simulation of version 1 using CFX confirms the IPPE results that version 3 has better cooling for the target window.

The porosity of the diffuser plate appears to have a strong effect on the flow field and temperature distribution. With the LAMPF beam condition, the flow separation behind the diffuser plate does not result in a high temperature downstream from the diffuser plate. Decreasing the plate porosity is desirable in order to direct more flow around the hot tip of the diaphragm window. For other beam conditions with a spread energy deposition, the flow separation behind the diffuser plate may lead to a high temperature region if there is a recirculation zone. In a design optimization study, the possibility of a recirculation zone should be decreased.

Reference

1. Institute of Physics and Power Engineering, Russian, "Preconceptual Design of 1-MW Flow Lead-Bismuth Target", Final Report, Reg. No. 35-06/64, May 29, 1996.
2. Fluent Inc., "FIDAP Manual", 1993
3. AEA Technology, "CFX 4.1 Flow Solver User Guide", 1995
4. Perry and Chilton, "Chemical Engineering Handbook", McGraw-Hill, 1977
5. I. E. Idelchik, "Handbook of Hydraulic Resistance", 3rd Edition, CRC Press Inc, Boca Raton Florida, 1994
6. R. D. Blevins, "Applied Fluid Dynamics Handbook", Van Nostrand Reinhold Co., New York, New York, 1984

Figure Captions

- Fig. 1. Four preconceptual designs of spallation target by IPPE.
- Fig. 2. Cross section of diffuser plate behind diaphragm in target version 3.
- Fig. 3. Computational mesh for FIDAP model of target version 3.
- Fig. 4. Computational mesh for CFX model of target version 3.
- Fig. 5. Flow field by streamlines in the overall target coolant as calculated by FIDAP.
- Fig. 6. Pressure field in the overall target coolant as calculated by FIDAP.
- Fig. 7. Turbulence energy field in the overall target coolant as calculated by FIDAP.
- Fig. 8a. Overall temperature field in the target coolant as calculated by FIDAP.
- Fig. 8b. Enlarged view of temperature field near the target as calculated by FIDAP.
- Fig. 9. Temperature distribution along the axis near the target window.
- Fig. 10. Convective heat transfer coefficient along the cone-sphere diaphragm.
- Fig. 11. Velocity vectors in the overall target coolant as calculated by CFX.
- Fig. 12. Enlarged view of velocity vectors near the target window as calculated by CFX.
- Fig. 13. Contour plots of velocity illustrating the recirculation region as calculated by CFX.
- Fig. 14. Overall temperature field in the target coolant as calculated by CFX.
- Fig. 15. Enlarged view of temperature field near the target window as calculated by CFX.
- Fig. 16. Pressure field in the target coolant near the diffuser plate as calculated by CFX.
- Fig. 17. Velocity vectors in the overall target version 1.
- Fig. 18. Enlarged view of the velocity vectors near the target window for target version 1.
- Fig. 19. Temperature distribution in the overall target version 1.
- Fig. 20. Enlarged view of temperature distribution near the target window for target version 1.
- Fig. 21a. Flow fields in target coolant for a diffuser plate (LAMPF beam)
 $\alpha_o = 0.05$.
- Fig. 21b. Flow fields in target coolant for diffuser plate (LAMPF beam)
 $\alpha_o = 0.10$.

- Fig. 21c. Flow fields in target coolant for a diffuser plate (LAMPF beam)
 $\alpha_o = 0.15$.
- Fig. 21d. Flow fields in target coolant for a diffuser plate (LAMPF beam)
 $\alpha_o = 0.20$.
- Fig. 21e. Flow fields in target coolant for a diffuser plate (LAMPF beam)
 $\alpha_o = 0.25$.
- Fig. 22a. Temperature fields in target coolant for a diffuser plate (LAMPF beam)
 $\alpha_o = 0.05$.
- Fig. 22b. Temperature fields in target coolant for a diffuser plate (LAMPF beam)
 $\alpha_o = 0.10$.
- Fig. 22c. Temperature fields in target coolant for a diffuser plate (LAMPF beam)
 $\alpha_o = 0.15$.
- Fig. 22d. Temperature fields in target coolant for a diffuser plate (LAMPF beam)
 $\alpha_o = 0.20$.
- Fig. 22e. Temperature fields in target coolant for a diffuser plate (LAMPF beam)
 $\alpha_o = 0.25$.
- Fig. 23. Temperature fields in target coolant with PSI-II beam
($\alpha_o = 0.05$).
- Fig. 24a. Variations of the maximum temperature and flow rate through the center hole with the mean porosity α_o .
- Fig. 24b. Variations of the maximum temperature and flow rate through the center hole with the loss coefficient K.

Fig. 1. Four preconceptual designs of spallation target by IPPE.

(scale 1:1)

Fig. 2. Cross section of diffuser plate behind diaphragm in target version 3.

2-D IPPE target Case 3

ELEMENT
MESH PLOT

R
↑
Z
→

Fig. 3. Computational mesh for FIDAP model of target version 3.

SCREEN LIMITS

ZMIN $-.167\text{E}+01$
ZMAX $0.752\text{E}+02$
RMIN $-.294\text{E}+02$
RMAX $0.387\text{E}+02$

FIDAP 7.62

29 Jan 98

08:48:03

Fig. 4. Computational mesh for CFX model of target version 3.

2-D IPPE target Case 3

STREAMLINE
CONTOUR PLOT

LEGEND

--- 0.9987E+02
--- 0.2996E+03
--- 0.4996E+03
--- 0.6996E+03
--- 0.8996E+03
--- 1.0996E+03
--- 1.2996E+03
--- 1.4996E+03
--- 1.6996E+03
--- 1.8996E+03
--- 2.0996E+03
--- 2.2996E+03

MINIMUM
-0.36629E-02
MAXIMUM
0.39949E+04

SCREEN LIMITS
ZMIN -.167E+01
ZMAX 0.752E+02
RMIN -.294E+02
RMAX 0.387E+02

Fig. 5. Flow field by streamlines in the overall target coolant as calculated by FIDAP.

FIDAP 7.62
3 Feb 98
15:58:30

2-D IPPE target Case 3

KINETIC ENERGY CONTOUR PLOT

LEGEND

-- 0.9284E+02
 -- 0.2785E+03
 -- 0.4642E+03
 -- 0.6499E+03
 -- 0.8356E+03
 -- 0.1021E+04
 -- 0.1193E+04
 -- 0.1378E+04
 -- 0.1764E+04
 -- 0.1950E+04
 -- 0.2135E+04

MINIMUM
 0.00000E+00
 MAXIMUM
 0.37137E+04

SCREEN LIMITS

ZMIN -.167E+01
 ZMAX 0.752E+02
 RMIN -.294E+02
 RMAX 0.387E+02

FIDAP 7.62
 17 Feb 98
 15:59:08

Fig. 7. Turbulence energy field in the overall target coolant as calculated by FIDAP.

2-D IPPE target Case 3

TEMPERATURE CONTOUR PLOT

LEGEND

MINIMUM
0.22000E+03
MAXIMUM
0.49133E+03

SCREEN LIMITS
ZMIN -.167E+01
ZMAX 0.752E+02
RMIN -.294E+02
RMAX 0.387E+02

FIDAP 7.62
9 Feb 98
16:13:14

Fig. 8a. Overall temperature field in the target coolant as calculated by FIDAP.

2-D IPPE target Case 3

TEMPERATURE CONTOUR PLOT	
<u>LEGEND</u>	
	- 0.461E+03
	- 0.431E+03
	- 0.401E+03
	- 0.371E+03
	- 0.341E+03
	- 0.310E+03
	- 0.280E+03
	- 0.250E+03
	MINIMUM 0.22000E+03
	MAXIMUM 0.49133E+03
<u>SCREEN LIMITS</u>	
	ZMIN 0.126E+02
	ZMAX 0.333E+02
	RMIN -.451E+01
	RMAX 0.139E+02
	FIDAP 7.62
	9 Feb 98
	16:13:59

Fig. 8b. Enlarged view of temperature field near the target window as calculated by FIDAP.

Fig. 9. Temperature distribution along the axis near the target window.

Heat Transfer Coefficient along the Diaphragm (coolant side, $\varepsilon=0.15$)

Fig. 10. Convective heat transfer coefficient along the cone-sphere diaphragm.

Velocity (m/s)

Fig. 11. Velocity vectors in the overall target coolant as calculated by CFX.

Fig. 12. Enlarged view of velocity vectors near the target window as calculated by CFX.

Fig. 13. Contour plots of velocity illustrating the recirculation region as calculated by CFX.

Temperature (C)

Fig. 14. Overall temperature field in the target coolant as calculated by CFX.

Temperature (C)

Fig. 15. Enlarged view of temperature field near the target window as calculated by CFX.

Fig. 16. Pressure field in the target coolant near the diffuser plate as calculated by CFX.

Fig. 17. Velocity vectors in the overall target version 1.

Fig. 18. Enlarged view of the velocity vectors near the target window for target version 1.

Fig. 19. Temperature distribution in the coolant for the overall target version 1.

Fig. 20. Enlarged view of temperature distribution near the target window for target version 1.

2-D IPPE target Case 3

STREAMLINE CONTOUR PLOT

LEGEND

-- 0.1268E+03
 -- 0.3805E+03
 -- 0.6341E+03
 -- 0.8877E+03
 -- 1.1413E+03
 -- 1.3949E+03
 -- 0.1903E+04
 -- 0.2156E+04
 -- 0.2410E+04
 -- 0.2663E+04
 -- 0.2917E+04

-- 0.4693E+04
 -- 0.4946E+04

MINIMUM
 -0.88431E-04
 MAXIMUM
 0.50730E+04

SCREEN LIMITS
 ZMIN -.167E+01
 ZMAX 0.752E+02
 RMIN -.294E+02
 RMAX 0.387E+02

FIDAP 7.62
 3 Feb 98
 15:55:44

Fig 21a. Flow fields in target coolant for a diffuser plate
 (LAMPF beam) $\alpha_0 = 0.05$.

2-D IPPE target Case 3

STREAMLINE CONTOUR PLOT

LEGEND

-- 0.1061E+03
 -- 0.3184E+03
 -- 0.5306E+03
 -- 0.7428E+03
 -- 0.9550E+03
 -- 0.1167E+04
 -- 0.1384E+04
 -- 0.1592E+04
 -- 0.1804E+04
 -- 0.2016E+04
 -- 0.2229E+04
 -- 0.2441E+04

-- 0.3901E+04
 -- 0.4139E+04

MINIMUM
 -0.15811E-05
 MAXIMUM
 0.42448E+04

SCREEN LIMITS

ZMIN -.167E+01
 ZMAX 0.752E+02
 RMIN -.294E+02
 RMAX 0.387E+02

FIDAP 7.62
 3 Feb 98
 15:51:48

Fig. 21b. Flow fields in target coolant for a diffuser plate (LAMPF beam) $\alpha_o = 0.10$.

2-D IPPE target Case 3

STREAMLINE CONTOUR PLOT

LEGEND

-- 0.9987E+02
 -- 0.2996E+03
 -- 0.4994E+03
 -- 0.6991E+03
 -- 0.8988E+03
 -- 1.0985E+03
 -- 1.2982E+03
 -- 1.4979E+03
 -- 1.6976E+03
 -- 1.8973E+03
 -- 2.0970E+03
 -- 2.2967E+03

-- 0.3694E+04
 -- 0.3895E+04

MINIMUM
 -0.36629E-02
 MAXIMUM
 0.39949E+04

SCREEN LIMITS

ZMIN -.167E+01
 ZMAX 0.752E+02
 RMIN -.294E+02
 RMAX 0.387E+02

FIDAP 7.62
 3 Feb 98
 15:58:30

Fig. 2lc. Flow fields in target coolant for a diffuser plate (LAMPF beam) $\alpha_o = 0.15$.

2-D IPPE target Case 3

STREAMLINE CONTOUR PLOT

LEGEND

-- 0.9965E+02
-- 0.2989E+03
-- 0.4982E+03

-- 0.6975E+03
-- 0.8968E+03
-- 1.0961E+03

-- 1.2954E+03
-- 1.4947E+03
-- 1.6940E+03

-- 1.8933E+03
-- 2.0926E+03
-- 2.2919E+03

-- 2.4912E+03
-- 2.6905E+03
-- 2.8898E+03

-- 3.0891E+03
-- 3.2884E+03
-- 3.4877E+03

-- 3.6870E+03
-- 3.8863E+03
-- 4.0856E+03

MINIMUM
-0.34284E-05

MAXIMUM
0.39860E+04

SCREEN LIMITS

ZMIN -.167E+01

ZMAX 0.752E+02

RMIN -.294E+02

RMAX 0.387E+02

FIDAP 7.62

3 Feb 98

15:53:41

Fig. 2ld. Flow fields in target coolant for a diffuser plate (LAMPF beam) $\alpha_0 = 0.20$.

2-D IPPE target Case 3

STREAMLINE CONTOUR PLOT

LEGEND

-- 0.1001E+03
-- 0.3002E+03
-- 0.5004E+03
-- 0.7005E+03
-- 0.9006E+03
-- 0.1100E+04
-- 0.1301E+04
-- 0.1501E+04
-- 0.1701E+04
-- 0.1901E+04
-- 0.2101E+04
-- 0.2302E+04

MINIMUM
-0.17605E-04
MAXIMUM
0.40028E+04

SCREEN LIMITS
ZMIN -.167E+01
ZMAX 0.752E+02
RMIN -.294E+02
RMAX 0.387E+02

FIDAP 7.62
3 Feb 98
15:57:05

Fig. 2le. Flow fields in target coolant for a diffuser plate (LAMPF beam) $\alpha_0 = 0.25$

2-D IPPE target Case 3

TEMPERATURE CONTOUR PLOT

LEGEND

-- 0.2279E+03
 -- 0.2437E+03
 -- 0.2595E+03
 -- 0.2753E+03
 -- 0.2911E+03
 -- 0.3070E+03
 -- 0.3228E+03
 -- 0.3386E+03
 -- 0.3544E+03
 -- 0.3702E+03
 -- 0.3860E+03
 -- 0.4018E+03
 -- 0.4176E+03

-- 0.5125E+03
 -- 0.5283E+03

MINIMUM
 0.22000E+03

MAXIMUM
 0.53620E+03

SCREEN LIMITS

ZMIN -.167E+01
 ZMAX 0.752E+02
 RMIN -.294E+02
 RMAX 0.387E+02

FIDAP 7.62
 3 Feb 98
 15:56:59

Fig. 22a. Temperature fields in target coolant for a diffuser plate
 (LAMPF beam) $\alpha_0 = 0.05$

2-D IPPE target Case 3

TEMPERATURE CONTOUR PLOT

LEGEND

-- 0.2274E+03
 -- 0.2422E+03
 -- 0.2570E+03
 -- 0.2718E+03
 -- 0.2866E+03
 -- 0.3014E+03
 -- 0.3162E+03
 -- 0.3310E+03
 -- 0.3460E+03
 -- 0.3608E+03
 -- 0.3756E+03
 -- 0.3904E+03

-- 0.4942E+03
 -- 0.5090E+03

MINIMUM
 0.22000E+03
 MAXIMUM
 0.51639E+03

SCREEN LIMITS

ZMIN -.167E+01
 ZMAX 0.752E+02
 RMIN -.294E+02
 RMAX 0.387E+02

FIDAP 7.62
 3 Feb 98
 15:53:35

Fig. 22b. Temperature fields in target coolant for a diffuser plate (LAMPF beam) $\alpha_0 = 0.10$.

2-D IPPE target Case 3

TEMPERATURE CONTOUR PLOT.

LEGEND

-- 0.2268E+03
 -- 0.2403E+03
 -- 0.2539E+03
 -- 0.2675E+03
 -- 0.2810E+03
 -- 0.2945E+03
 -- 0.3081E+03
 -- 0.3217E+03
 -- 0.3353E+03
 -- 0.3489E+03
 -- 0.3624E+03
 -- 0.3760E+03

-- 0.4710E+03
 -- 0.4845E+03

MINIMUM
 0.22000E+03
 MAXIMUM
 0.49133E+03

SCREEN LIMITS

ZMIN -.167E+01
 ZMAX 0.752E+02
 RMIN -.294E+02
 RMAX 0.387E+02

FIDAP 7.62
 3 Feb 98
 15:58:25

Fig. 22c. Temperature fields in target coolant for a diffuser plate
 (LAMPF beam) $\alpha_0 = 0.15$.

2-D IPPE target Case 3

2-D IPPE target Case 3

Fig. 22e. Temperature fields in target coolant for a diffuser plate
(LAMPF beam) $\alpha_0 = 0.25$.

2-D IPPE target Case 3

TEMPERATURE CONTOUR PLOT

LEGEND

-- 0.2276E+03
 -- 0.2429E+03
 -- 0.2582E+03
 -- 0.2735E+03
 -- 0.2888E+03
 -- 0.3041E+03
 -- 0.3194E+03
 -- 0.3346E+03
 -- 0.3499E+03
 -- 0.3652E+03
 -- 0.3805E+03
 -- 0.3957E+03

-- 0.5027E+03
 -- 0.5180E+03

MINIMUM
 0.22000E+03

MAXIMUM
 0.52562E+03

SCREEN LIMITS

ZMIN -.159E+01
 ZMAX 0.333E+02
 RMIN -.105E+02
 RMAX 0.204E+02

FIDAP 7.62
 3 Feb 98
 16:03:25

Fig. 23. Temperature fields in target coolant with PSI-II beam ($\alpha_0 = 0.05$).

Fig. 24a. Variation of the maximum temperature and flow rate through the center hole with the mean porosity α_0 .

Fig. 24b. Variation of the maximum temperature and flow rate through the center hole with the loss coefficient K.