LA-UR- 00-2549

Approved for public release; distribution is unlimited.

Title: INFLUENCE OF CFD ANALYSIS ON THE DESIGN OF COOLING CHANNELS FOR THE CCL CAVITY FOR THE SPALLATION NEUTRON SOURCE

Author(s): Snezana Konecni and Nathan K. Bultman
Spallation Neutron Source Division
Los Alamos National Laboratory
Los Alamos, NM 87545

Submitted to: International Mechanical Engineering
Congress & Exposition
November 5-10, 2000
Walt Disney World Dolphin
Orlando, Florida

Los Alamos
NATIONAL LABORATORY

DISCLAIMER

DISCLAIMER

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

Proceedings of IMECE 2000:
International Mechanical Engineering
Congress & Exposition
November 5-10, 2000
Walt Disney World Dolphin
Orlando, Florida

INFLUENCE OF CFD ANALYSIS ON THE DESIGN OF COOLING CHANNELS FOR THE CCL CAVITY FOR THE SPALLATION NEUTRON SOURCE

Snezana Konecni and Nathan K. Bultman
Spallation Neutron Source Division
Los Alamos National Laboratory
Los Alamos, NM 87545

ABSTRACT

Water flow in cooling channels was simulated using the computational fluid dynamics (CFD) code CFX4. Pressure drop in the cooling channels of the coupled-cavity linac (CCL) cavity was calculated. The effects of the manifold on the pressure drop were studied also. Reducing the pressure drop was a primary goal of this exercise that led to changing the cooling channel entrance regions. Results of this analysis were used in sizing pumps required for the cooling system.

For the validation of the simplified numerical model, an experiment was performed to measure the pressure drop in the cooling channels for variable flow rate, using a flow loop. Deionized water was circulated through the test section with a pump and its flow rate was monitored with a turbine flow meter. Pressure was monitored with pressure transducers at five locations including a differential pressure transducer across the test section, and water temperature was taken at the exit of the pump.

Pressure drop across the inlet and outlet of the test section was measured and recorded for different flow rates. Flow rate was also monitored and stored simultaneously. From the recorded data, an empirical correlation was derived to describe the pressure drop, *dp*, as a function of flow rate through the four cooling channels.

INTRODUCTION

The Spallation Neutron Source (SNS) is an accelerator-based facility that produces pulsed beams of neutrons by bombarding a mercury target with intense beams of 1-GeV protons. Los Alamos National Laboratory is responsible for the design, fabrication, installation, and testing of the normal-conducting section of the Linear Accelerator (linac) for the SNS project. The normal-conducting linac accepts a beam from the front end system and accelerates it from 2.5 MeV to 187 MeV. The linac consists of a drift-tube linac (DTL) up to 87 MeV and a coupled-cavity linac (CCL) up to 188 MeV. The DTL operates at an RF resonant frequency of 402.5 MHz, while the CCL operates at 805 MHz. Power dissipated in the linac due to RF resonant frequency is removed by water circulating through the cooling channels of the cavities and the drift tubes. CCL cavities are figures of revolution about the beam axis. Several cooling passage schemes were studied. The original scheme includes four main parallel annular paths around the cavity nose in the septum, fed and collected by internal plenums, and a circular path on the opposite face. The plenums are supplied by water from a manifold that feeds a group of eight cavities called an accelerator segment. In order to design the cooling system for the linac, a pressure drop of the water flowing through the cooling channels was needed.

This paper discusses the development of and results from the CFX models including model geometry, boundary conditions, and modeling approximations. Details are also provided about discretization (optimization), convergence criteria, and empirical benchmarking. Results are presented in form of contour plots of the water flow speed. In addition, graphical comparison of pressure predictions with empirical data is provided.

NOMENCLATURE

dp pressure drop (psi)

y+ local thickness Reynolds number.

CAVITY COOLING PASSAGE DESIGN

CCL cavities are figures of revolution about the beam axis An automated tuning program sets up the geometry for a symmetric accelerating cavity and runs SUPERFISH repetitively, varying the geometry to tune each cavity to the desired frequency for the electromagnetic fields. SUPERFISH is an in-house code that solves Maxwell's equations in 2-D. A large portion of the RF power (70-80%) applied to accelerate protons is a waste heat deposited on the inside of the copper cavity. This waste heat is removed most efficiently with water circulating through cooling passages. The waste heat needs to be removed in order to minimize thermal deformations and with it control the resonance of the cavities.

Several different cooling schemes were studied. In one case, cooling channels were brazed on the outside walls of the cavities. The study of this scheme showed that the temperatures and stresses at the nose area at the center of the cavities were very high, resulting in high frequency shifts. Consequently, it was decided to machine cooling channels within the copper structure on the backside of the cavity. This cooling passage design removes heat efficiently from the nose area and results in more uniform temperature and stress distributions. The current design has four concentric cooling channels with a 1 mm by 7 5 mm cross section, as shown in Figure 1.

CFD ANALYSIS AND COOLING PASSAGE DESIGN

A CFD analysis was performed to minimize the pressure drop and improve the cooling flow distribution within the cooling channels. CFD refers to the simultaneous numerical solution of continuity, momentum, and energy equations for flow field geometry with specified boundary conditions. CFX4, the commercial CFD package used in this study, uses a finite volume (finite difference) scheme based on the SIMPLEC mathematical algorithm discussed by Van Dormaal and Raithby (1984) A predecessor of this algorithm, SIMLE is given by Patankar (1980).

CFD codes sometimes offer advantages over experimental techniques in investigating fluid flow. These advantages include savings in time and money associated with constructing and testing large scale experiments, enhanced visualization of the flow, and the ability to quickly and conveniently perform parametric and optimization studies on a fluid system. However, computational models can inadvertently yield inaccurate predictions. It is good engineering practice to benchmark the CFD results with an experiment.

NUMERICAL METHOD

The geometrical model of the CCL cavity is shown in Figure 1. An IGES model was extracted from the Unigraphics model and imported in the CFX4 preprocessing tool called Build. The focus of this CFD study was isothermal flow, so from the IGES model, only the cooling channels were used to build the block structure. It is assumed that the cooling channels have a total depth of 2 mm. This will be true when two half-cells are brazed together There are four channels on the back side of the septum and one channel to cool the short coupling cell, which at this time has not been modeled. The four channels are fed from a common manifold, as displayed in Figure 1. CFD was used to design the shape of the cooling channels.

A multi-block scheme with a body-fitted grid structure was used to discretize the three-dimensional numerical model into 146,220 elements. The grid corresponding to the first design displayed in Figure 2, was optimized to capture the boundary layer flow and provide a grid independent solution. Grid-independence was determined by running a particular problem to steady-state convergence with finer mesh until the solution failed to deviate between consecutive runs. The criterion for capturing the boundary layer was the value of *y+*, the "law of the wall". *y+* is a parameter representing a local thickness Reynolds number (Kays and Crawford, 1993). In this case the values for *y+* are below 200.

The convergence of a particular model to a solution is determined with residuals. Residuals are a measure of the error of each dependent variable determined at each cell, and summed over all cells in the model for a single iteration Convergence is determined when the sum of the residuals for each dependent variable dropped by several orders of magnitude and ceased to vary between consecutive iterations. Mass balance in the model systems are used as self-consistency checks. For the steady-state problems considered in this study, adequate convergence was assumed when the mass balance agreed to within 5%. The most important verification of the numerical methods was performed by experimental validation, in which empirical data were compared with numerical predictions.

There are several CFD models developed in this study. With these CFD models the following goals were accomplished.

1. To determine the optimum shape of the cooling channels to minimize pressure drop and evenly distribute the water flow.
2. To determine the pressure drop through the cooling channels and the cooling assembly. This pressure drop later is required as a flow resistance in a flow network modeling effort of the linac cooling system.

MODELING AND RESULTS

The modeling effort began with a cooling channel design developed in the conceptual design phase of the CCL The channel's cross-section was 2 mm by 10 mm. With an average channel velocity of 3 m/s the flow through the channels is turbulent, which results in high pressure drops, revealing an non-optimized design. Turbulent flow through the channels is important in the effective removal of the heat from the cavity. Consequently turbulent flow and low pressure drop for each cavity is an optimum situation. Figures 3-4 show the development of the geometry of the cooling channels and respective flow fields and pressure drops. In Figure 3 velocity contours are shown for a 0.239 kg/s (3.81 gpm) flow rate and the first design of the cooling channels. The pressure drop and flow rate per channel for this cavity is given in Table 1. Pressure drop for this flow between inlet and outlet of the computation domain is 70729 Pa (10.26 psi). Channel 1 is the channel closer to the outer edge of the cavity. Figure 4 shows the velocity contours in a cooling channel design in which the corners of the flow channel have been rounded. Pressure drop for this design is about 2 psi lower than the previous design for the same flow rate (Table 2).

Figure 5 shows the structure and velocity contours of the final optimized cooling channel design in which the channels have been configured to minimize pressure drop and evenly distribute the water flow. In order to reduce the total flow needed for cooling the cavity and maintain turbulent flow, the area of the channels was reduced to 2 mm by 7.5 mm. With a flow rate of 1.42 gpm, average velocity in a channel is 2 m/s and Reynolds number is 7650. The results are presented in Table 3 and velocity contours in Figure 5.

Table 1. Pressure drop and mass flow rate per 10 mm by 2 mm channel for the CCL cavity, for a flow of 0.234 kg/s (3.7 gpm); this is for sharp corners.

	Flow (kg/s)	Flow (gpm)	Pressure drop Pa	Pressure drop (psi)
Channel 1	0.0588	0.932	28973	4.12
Channel 2	0.0522	0.827	25116	3.64
Channel 3	0.0573	0.908	18035	2.62
Channel 4	0.0655	1.038	17505	2.5
Total flow	0.2341	3.7	70729	10.26

Table 2. Pressure drop and mass flow rate per 10 mm by 2 mm channel for the CCL cavity, for a flow of 0.239 kg/s (3.8 gpm), this is for rounded corners.

	Flow (kg/s)	Flow (gpm)	Pressure drop Pa	Pressure drop (psi)
Channel 1	0.0627	0.995	29381	4.12
Channel 2	0.0534	0.845	19506	2.83
Channel 3	0.0556	0.882	15484	2.24
Channel 4	0.0672	1.066	13366	1.94
Total flow	0.239	3.8	58094	8.42

Table 3. Flows and pressure drop for four 7.5 mm by 2 mm channels, for a mass flow of 0.119 kg/s (1.885 gpm).

	Flow (kg/s)	Flow (gpm)	Pressure drop Pa	Pressure drop (psi)
Channel 1	0.0288	0.456	12006.4	1.74
Channel 2	0.0267	0.423	8757.8	1.27
Channel 3	0.0288	0.456	7791.7	1.11
Channel 4	0.0347	0.550	8173.7	1.18
Total flow	0.119	1.885	20896	3.03

EXPERIMENTAL PRESSURE DROP VERIFICATION

For the validation of the simplified numerical model, an experiment was performed to measure the pressure drop in the cooling channels for variable flow rate, using a flow loop. Deionized water is circulated through the test section with a pump and its flow rate is monitored with a turbine flow meter. Pressure is monitored with pressure transducers at five locations including a differential pressure transducer across the test section, and water temperature is taken at the exit of the pump. The test section can be supplied with variable flow rate from 1.89 l/min to 75.7 l/min. For this experiment, the water flow rate was limited to a maximum of 15.14 l/min at room temperature of 20°C.

To determine the accuracy of the numerical model, a portion of a CCL cavity was machined out of aluminum (see Figure 6). Figure 6 is a photograph of the CCL cavity septum cooling channels. Cooling channels are 2 mm deep and 7.5 mm wide and match the optimized CFD model. The top of the channels was covered with a Lexan plate and sealed with vacuum grease. Lexan was used because it is transparent and allows for flow visualization. Food coloring was injected at the entrance of the test section and it was noticed that besides few recirculation zones in the channels entrance regions, the flow is uniform throughout the cooling channels. Turbulence of the flow was noticed to mix the dye with the water almost immediately as it entered the test section.

Pressure drop across the inlet and outlet of the test section was measured and recorded for different flow rates. Flow rate was also monitored and stored simultaneously. From the recorded data, the following empirical correlation was derived to describe the pressure drop (psi), *dp*, as a function of *flow rate* (gpm) through the four cooling channels:

$$dp = 0.8648(flow\ rate)^2 + 0.5271 \quad (1)$$

In order to compare the experimental versus numerical model pressure, the pressure drop through the various hose fittings was subtracted since it was not accounted for in the CFD model. Equation 1 represents pressure drop only through the cooling channels with the losses of the fittings subtracted. Pressure drop measurements were performed for the CCL cooling passages in the range of 1.89 l/min to 15.14 l/min (0.5 gpm to 4 gpm). Comparison between measured and predicted pressure drop is shown in Figure 7.

Experimental results compare well with the numerical simulation. Numerical prediction of the pressure drop underestimates the measured one possibly because the numerical method treats the walls of the channels as smooth. Adding surface roughness to the model might increase the calculated pressure drop. Inlet boundary condition for the calculation is a constant mass flow rate which results in fully developed flow at the entrance. In the experimental setup the flow does not have time to develop fully before entering the test section. This difference might contribute to higher pressure drop during testing. Taking in account the uncertainty of fitting loss correlations and the uncertainty in the measurements of the flow rate and pressure, predicted results are within 15% of the measured for a flow rate of 4 gpm and lower for smaller flow rates.

UNCERTAINTY ANALYSIS

Uncertainty in the flow rate and pressure measurements is calculated using Kline and McClintok (1953) formulas and they are:

1. Flow rate is measured with Omega FTB-603 flowmeter and it has 3.9% uncertainty
2. Differential pressure is measured ±0.0608 psi up to 5 psi, and 0.48 psi above 5 psi. Differential pressure transducer Validyne DP-15 was used to measure test section pressure drop up to 5 psi. Differental pressures larger than 5 psi were calculated as a difference from the gage pressure transducers Omega PX 615-060 placed at the inlet and outlet of the test section.
3. The error in reading the differential pressure above 5 psi is almost 10 times larger than the measurements bellow 5 psi.
4. Pressure drop through the fittings was calculated using well known correlations for losses through pipe fittings.

SUMMARY

In this study a CFD analysis was used to design the shape of the CCL cooling channels and determine their pressure drop. Test section was designed to represent the cooling channels of the cavity and water was circulated through them. Experimentally obtained pressure drop and flow rate measurements were compared to the numerical prediction. The comparison is presented in Figure 6. The numerical predictions are within 15% of the measured pressure drop for the given flow rate range.

REFERENCES

Kays, W.M. and Crawford, M.E., 1993, Convective Heat and Mass Transfer, McGraw-Hill Inc., pg. 210-212

Kline, S.J. and McClintock, F.A., 1953, "Describing Uncertainties in Single-Sample Experiments," *Mechanical Engineering*, Vol. 75, pp. 3-8.

Patankar, S.V., 1980, Numerical heat transfer and fluid flow, McGraw-Hill: New York, New York.

Van Dormaal, J.P., Raithby, G.D., 1984, "Enhancements of the SIMPLE method for predicting incompressible fluid flows," *Num. Heat Trans.*, Vol. 7, pp. 147-163

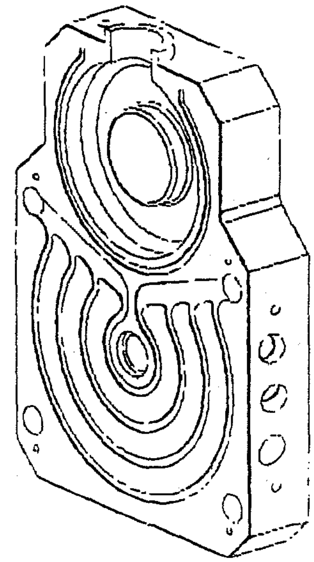

Figure 1. CCL Cavity geometry

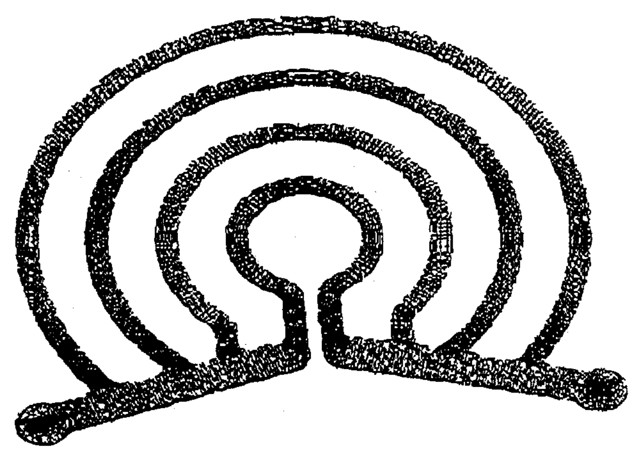

Figure 2. Grid distribution for the first design of the cooling channels

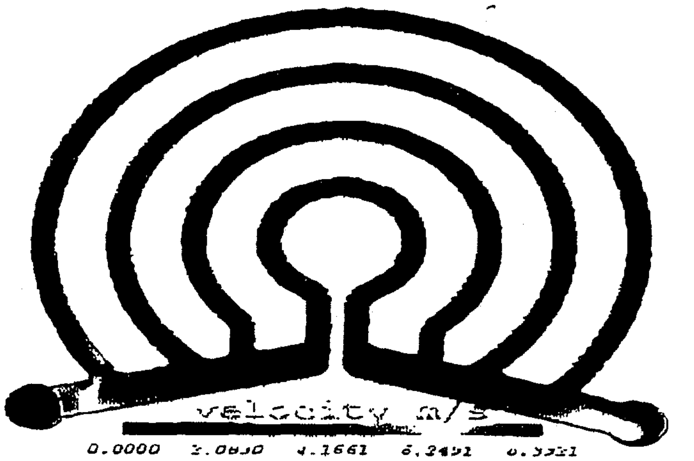

Figure 3. Velocity contours for the first design for 0.239 kg/s (3.8 gpm) flow rate and 70729 Pa (10.25 psi) pressure drop.

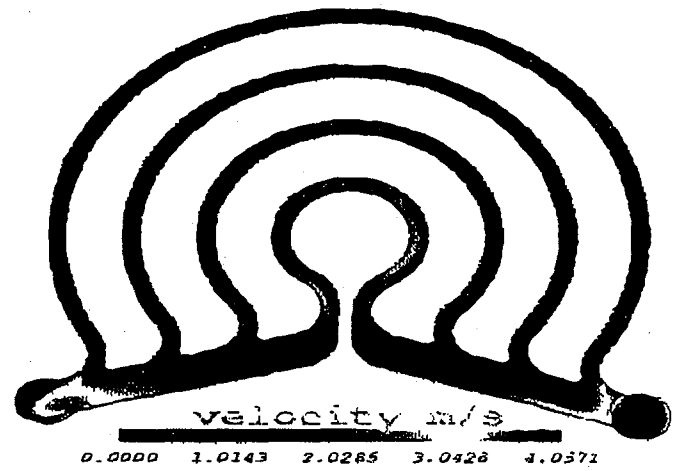

Figure 5. Velocity contours for the "0.75 cm" design 0.119 kg/s (1.88 gpm) flow rate and 20896 Pa (3.03 psi) pressure drop.

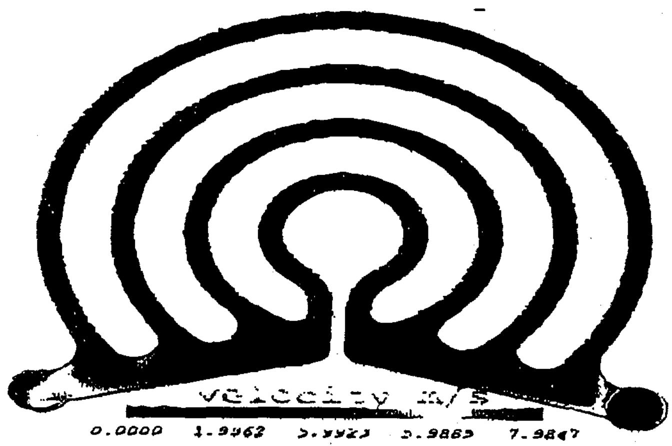

Figure 4. Velocity contours for the "rounded corners" design 0.239 kg/s (3.8 gpm) flow rate and 57917 Pa (8.4 psi) pressure drop.

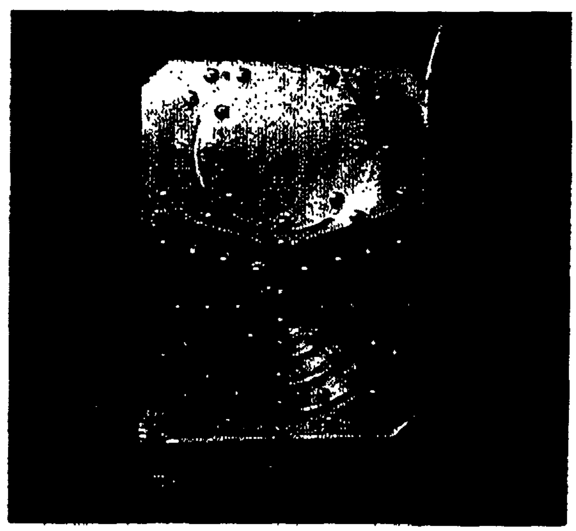

Figure 6. Photographs of the experimental test section showing the cooling channels

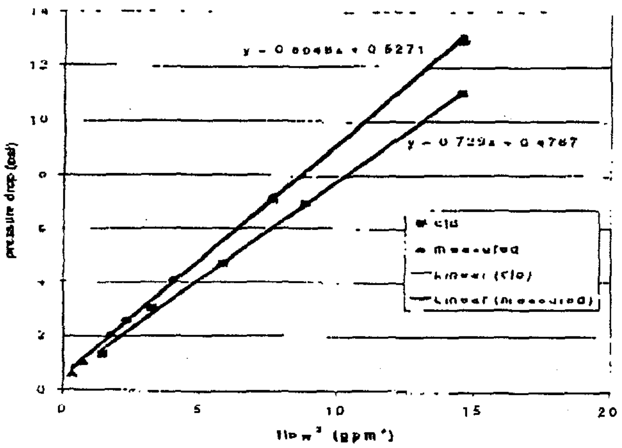

Figure 7. Comparison between measured and computed pressure drop as a function of flow^2.