PRELIMINARY ASSESMENT OF

A THICK LIQUID BLANKET \ FIRST WALL CONCEPT

Karani Gulec

APEX Study Meeting

July 26-28, 1998

UCLA

SIMULATIONS (I)

* A simulation that contains two compartments and a thin jet flowing at the front section is performed, in order to assess the physical limitations of the minimum blanket/liquid first wall concept with the fusion power constraints.

* A transient and steady state simulation of the fluid motion in the compartment with modified size and inner wall contours is performed in order to observe that shear induced rotation may continue in the compartment without any contribution by the primary jet.

* A volumetric heat generation in the blanket region is defined in a discrete spatial distribution (from plasma boundary to blanket container, 0-10 cm : 50 MW/m^3, 10-30 cm : 20 MW/m^3, 30-50 cm : 5 MW/m^3)

SIMULATIONS (II)
Operating & Boundary Conditions

* Flibe is used as an operating fluid with thermo-physical properties at 550 Celsius.

* Zero velocity at the solid boundary has been used, free surface tracking is activated in the code and it is assumed that the fluid completely wets the structure.

* Continuos boundary condition is used for the open surfaces (where there is a discharge.)

* Symmetry boundary condition is used in the torodial direction.

* A constant inlet velocity is used as an inlet boundary condition to the compartment.

ONGOING STUDY (I)

* The shape, size, and operating condition of the exit section of the compartments has an important role in the performance of the liquid blanket/first wall. The exit should be located in the compartment where its effect on the streamlines in the compartment are minimized or zero.

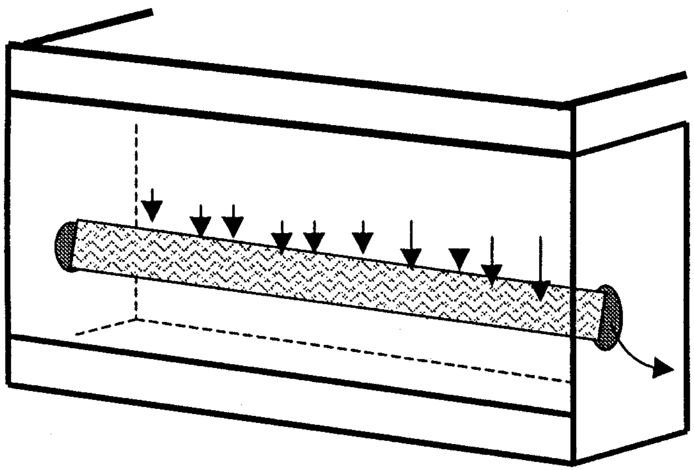
* There are two main options for the exit designs currently under investigation:

* The first option,

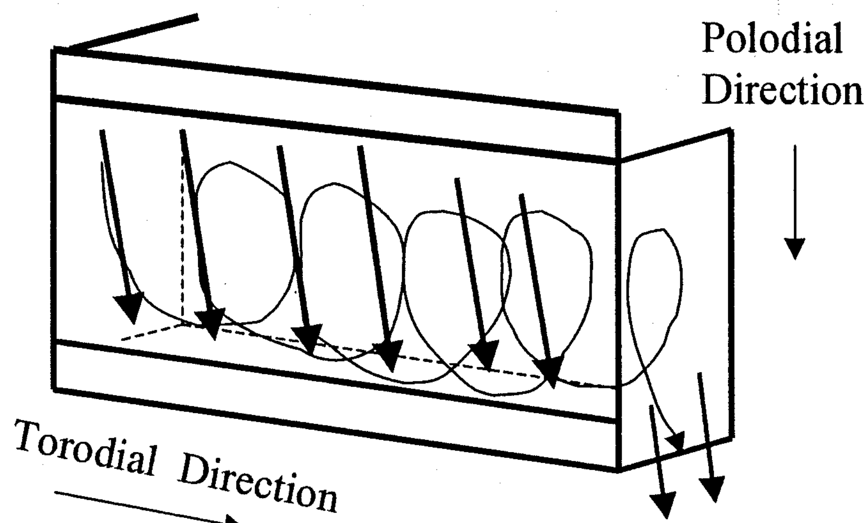
  > The exits for each compartment may be located at the two side planes of the compartments in the torodial direction directly across the stagnant point of the rotating fluid.

  > Then a pipe with a certain diameter may be inserted into the compartment at the stagnant velocity level to remove the fluid coming to this section as a result of gravitational acceleration.

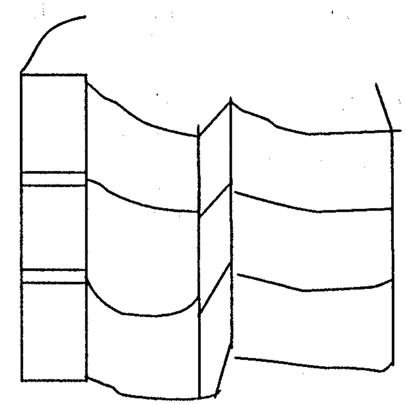
ONGOING STUDY (II)



First Option



Second Option



Outside of Vacuum Vessel

ONGOING STUDY (III)

* The second option,

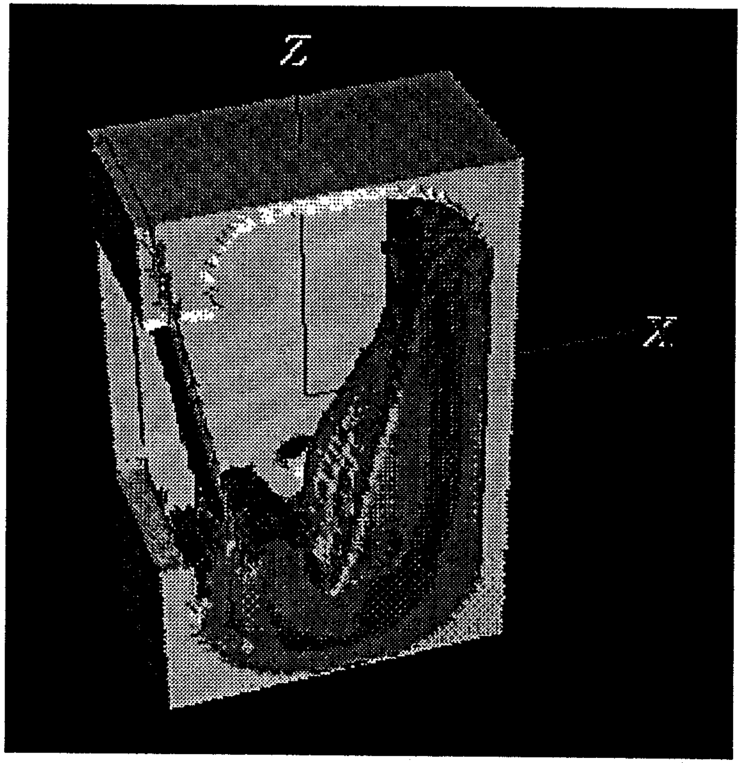
  > The jets in the fluid compartments may be injected with a velocity component in the torodial direction in addition to their main velocity component in the polodial direction.

  > Then the rotating fluid may be directed to the outlet based on the value of velocity components of the jets in the torodial direction.

  > This technique also maximizes the blanket/first wall fluid residency time in the compartment, and therefore the temperature of the blanket/first wall fluid.

ONGOING STUDY (IV)

* Three-dimensional CFD analysis is required in order to design optimum exit locations and operating conditions for the compartment.



GOALS IN IMPROVING THE CONCEPT & FUTURE WORK

* Increase resident time of the blanket/first wall fluid in the compartment in to enhance thermal efficiency.

  --> Compartment design will be optimized.

  --> Outlet design will be chosen for outlet as discussed.

* Isolate the compartments from the high speed jets so that instead of having one stable system we will have a blanket/first wall system with redundant safety.

* Minimize the primary jet thickness as much as possible.

  > The contour of backplate for the high velocity thin jet will be optimized in order to eliminate possibility of splashing and the energy losses while the high velocity jet changes its direction as it moves over .

  > The drag on the back plates may be minimized using riblets instead of clean mirror like surfaces or by using a porous material through which some fluid in the compartment can be leaked to the concave plate surface.

INTRODUCTION

The minimum structure thick liquid wall (liquid blanket) concept consists of two main sections.

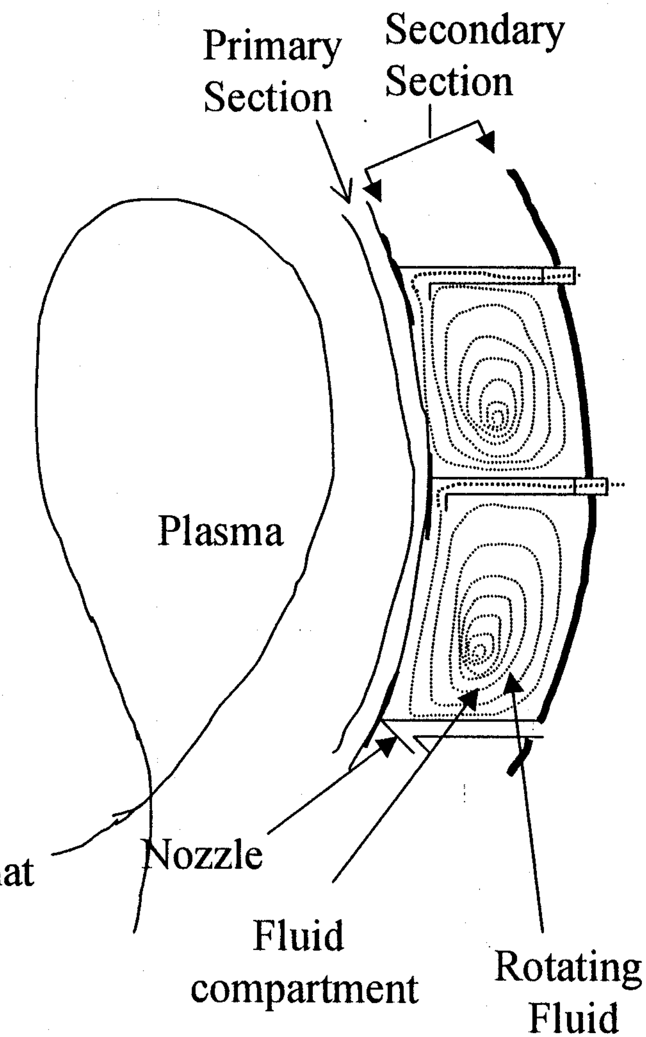
Primary Section:

High velocity (>10 m/sec), thin (<0.02 m) liquid jet flowing in the polodial direction (facing to the plasma).

Secondary Section:

Series of fluid compartments along the polodial and torodial direction filled by Flibe.

The operating fluid in the Primary Section is recycled in the Secondary Section in order to increase fluid exposure time for nuclear heating so that higher thermal efficiency can be obtained.



DESIGN APPROACH

* A base case has been modeled that has two compartments in the polodial direction and a thin jet flowing at the front section of the compartments (between plasma and the compartment section). The trajectory of the high velocity thin jet is controlled by the concave walls located at the edges of the compartment's upper and lower walls.

Performing base case simulation,

--> The streamlines and the range of required operating conditions for the flow in the compartments and the high velocity jet have been determined.

--> The concave jet flowing at the front section of the compartments pushes back any possible deflections of the rotating fluid in the compartments by centrifugal acceleration.

DESIGN APPROACH (II)

* Attention has been given in improving the compartment efficiency with the main purpose that each fluid compartment should maintain its thickness and stability by itself.

  --> Fluid compartment dimensions, size, inlet jet velocity and its thickness are varied to optimize the performance of the pocket (minimize the solid mass ratio).

  --> The contour of the inner wall of the compartments is varied so that losses in the kinetic energy of the fluid are minimized while the direction of the fluid changes as it rotates in the compartment.

* A spatial volumetric heat deposition has been used in the fluid compartment in order to estimate the thermal performance of a cross section of the rotating blanket fluid in the compartment.

* Using 2-D simulation results, two design options for the exit locations of the compartments have been determined.

ABOUT FLOW-3D

* Flow-3D*, is a three dimensional free surface flow code that uses VOF (Volume of Fluid Method) and includes a variety of features which enable one to model acoustic waves, cavitation, the solidification and shrinkage of metals, flow through porous media, surface tension, and wall adhesion.

* The Flow-3D code has been developed for aerospace, petroleum, casting industries for problems such as in jet design, control of propellant (i.e. slosh) for the fuel delivery systems for aircraft and spacecraft and design of oil rigs on the ocean against ocean waves.

* It has been widely used and continuously improved for the various industries at several universities and research centers for 18 years. Have validated results and over 100 papers written.

* Flow 3D- www.flow3d.com

ABOUT FLOW-3D
Continuity Equations

Simplest Case, $\nabla . \vec{v} = 0$

Incompressible Flow

$$\frac{\partial}{\partial x}(uA_x) + \frac{\partial}{\partial y}(vA_y) + \frac{\partial}{\partial z}(wA_z) = \frac{RSOR}{\rho}$$

Variable Density or Compressible Flow

$$V_F \frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho u A_x) + R\frac{\partial}{\partial y}(\rho v A_y) + \frac{\partial}{\partial z}(\rho w A_z) + \zeta \frac{\rho u A_x}{x} = RDIF + RSOR$$

$$RDIF = \frac{\partial}{\partial x}\left(\upsilon_p A_x \frac{\partial \rho}{\partial x}\right) + R\frac{\partial}{\partial y}\left(\upsilon_p A_y \frac{\partial \rho}{\partial y}\right) + \frac{\partial}{\partial z}\left(\upsilon_p A_z \frac{\partial \rho}{\partial x}\right) + \zeta \frac{\rho \upsilon_p A_x}{x}$$

RDIF = Density Diffusion

RSOR = Mass Source/Sink

ABOUT FLOW-3D
Momentum & (Volume-of-Fluid) Advection Equations

Momentum Equations:

$$\frac{\partial u}{\partial t}+\frac{1}{V_F}\left\{uA_x\frac{\partial u}{\partial x}+vA_y\frac{\partial u}{\partial y}+wA_z\frac{\partial u}{\partial z}\right\}=-\frac{1}{\rho}\frac{\partial p}{\partial x}+F_x-\frac{RSOR}{\rho V_F}u$$

$$\frac{\partial v}{\partial t}+\frac{1}{V_F}\left\{uA_x\frac{\partial v}{\partial x}+vA_y\frac{\partial v}{\partial y}+wA_z\frac{\partial v}{\partial z}\right\}=-\frac{1}{\rho}(R\frac{\partial p}{\partial y})+F_y-\frac{RSOR}{\rho V_F}v$$

$$\frac{\partial w}{\partial t}+\frac{1}{V_F}\left\{uA_x\frac{\partial w}{\partial x}+vA_y\frac{\partial w}{\partial y}+wA_z\frac{\partial w}{\partial z}\right\}=-\frac{1}{\rho}\frac{\partial p}{\partial z}+F_z-\frac{RSOR}{\rho V_F}w$$

F= Forces -- Gravity and non-inertial body acceleration
Viscous stresses
Drag (porous baffles, porous obstacles)

Volume-of-Fluid, Advection Equation :

$$\frac{\partial F}{\partial t}+\frac{1}{V_f}\left\{\frac{\partial}{\partial x}(FuA_x)+\frac{\partial}{\partial y}(FvA_y)+\frac{\partial}{\partial z}(FvA_y)\right\}=FDIF+FSOR$$

FDIF= Diffusion of Fluid Fraction , FSOR= Fluid Source / Sink

ABOUT FLOW-3D
Sola Solution Outline

Step 1: Explicit Velocity Estimation

$$\frac{\partial \vec{u}}{\partial t} + \frac{\overrightarrow{Au}.\overrightarrow{\nabla u}}{V_f} = -\frac{1}{\rho}\overrightarrow{\nabla P} + \vec{g} + \frac{\vec{\nabla}.\{vA\overrightarrow{\nabla u}\}}{V_f} + \vec{s}$$

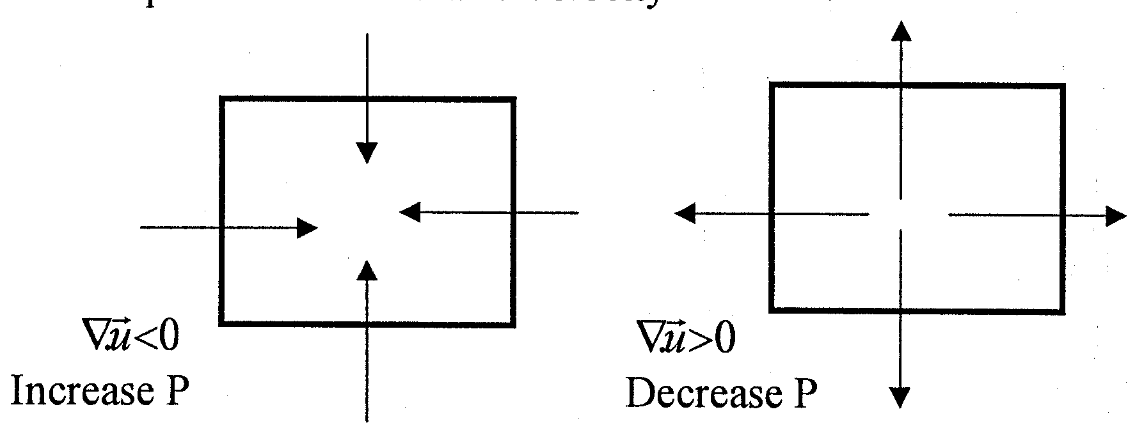
First or Second Order Forward Time Differencing
Explicit-- Momentum Flux/forces
Non-inertial Accelerations included in g

Step 2: Pressure Iteration to Satisfy Incompressibility

Continuity equation-Momentum equation Coupling gives rise to Poisson Equation for Pressure

Convergence criterion may be set by the user or chosen by the solver.

Results of Step 2. -- > Updated Pressures and Velocity



ABOUT FLOW-3D
Sola Solution Outline

Pressure Iteration Options

Cell-by -Cell (Successive Overrelaxation) or **Line by Line** (Line Implicit)

Step 3: Update Remaining Variables Using New Velocities

> VOF Advection of F

> Density evaluation for compressible or stratified Flow

> Energy/Turbulence Equations (heat transfer/Conduction)

> Derived Quantities

Viscosity

temperature dependent density.