PROGRESS ON ACCOMODATION OF PENETRATIONS IN LIQUID WALL SYSTEMS

Presented by : Karani Gulec

Contributors: M. Abdou, K. Gulec, N. Morley, S. Smolenstev, A. Ying

APEX Project Meeting
PPPL
May 12-14, 1998

INTRODUCTION

PURPOSE

To determine the design and operating conditions required for accommodation of penetrations in various liquid wall concepts.

STEPS

Accommodating penetrations in the high velocity liquid wall flow may be accomplished,.

1. Determining the critical issues in the penetration design and possible challenges that may arise when the penetration is located in the high velocity liquid wall.

2. Determining design goals.

3. Determining optimum penetration shapes required for minimum disruption to the liquid wall flow.

4. Determining required modifications to the surrounding area of penetrations for minimum disturbed flow.

INITIAL REFERENCE PENETRATION CASE FOR 3-D TIME DEPENDENT FLUID FLOW CALCULATIONS (I)

* The dimensions of the penetrations and problem domain are determined so that the problem can be computationally simulated within the capability of computational resources and reasonable computational time,

  1- Minimizing the loss of information as a result of using larger mesh sizes

  2- Maximizing the problem domain to more closely simulate a real life case.

* 3-D, time-dependent Navier-Stokes solver that utilizes VOF (Volume of Fluid) technique for free surface incompressible fluid flows have been used.

  (Detailed information www.fusion.ucla.edu/apex/ meeting # 4, Gulec et. al.)

* As a first step, a parametric penetration design study has been performed for a base penetration shape and dimension that is located on the flat plate.

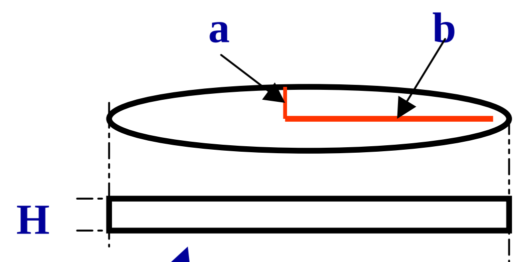
* The centripetal acceleration term is modeled as a force field with an acceleration term perpendicular to the flow direction and equal to the U^2_{axial} / R. (U_{axial} = 10 m/s, R=4.0 m)

INITIAL REFERENCE PENETRATION CASE FOR 3-D TIME DEPENDENT FLUID FLOW CALCULATIONS (II)

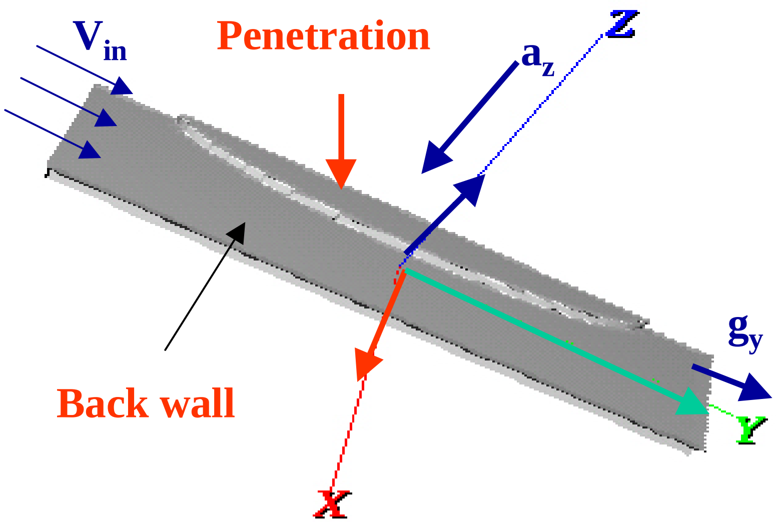
REFERENCE CASE PARAMTERS

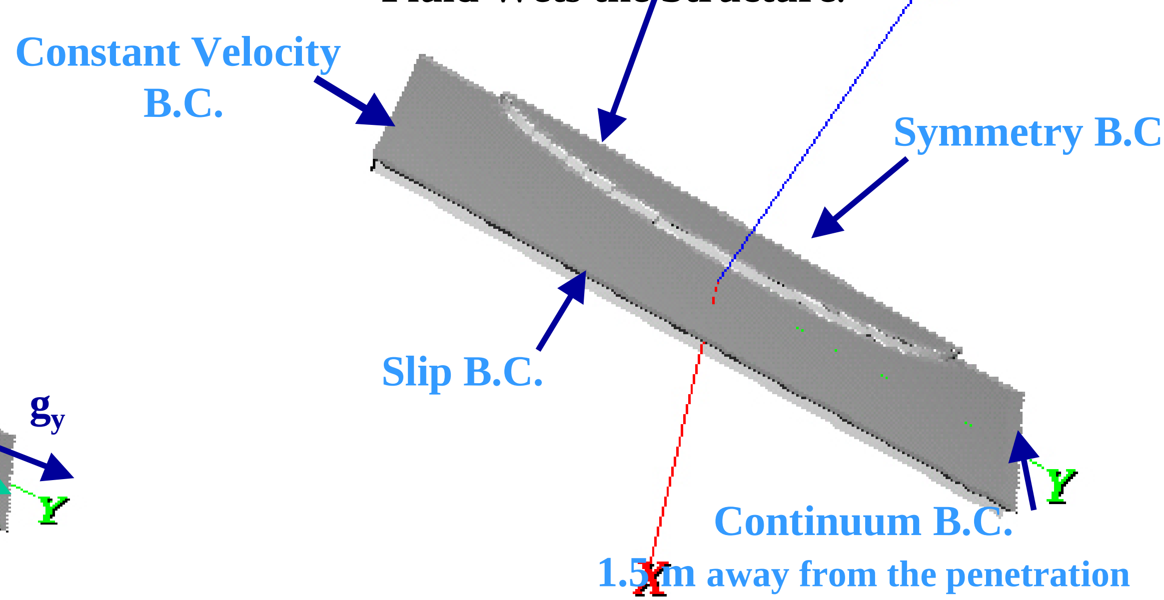
V_{in} (m/s)	10.0		
a_z (m^2/s)	25.0		
g_y (m^2/s)	9.8		
Wall Roughness (m)	10^{-5}		
Fluid-Wall Contact Angle	0.0		
Penetration Dimensions (m)	a	b	H
	.1	.45	0.02

Flibe at 550 ° C is used as a working fluid.



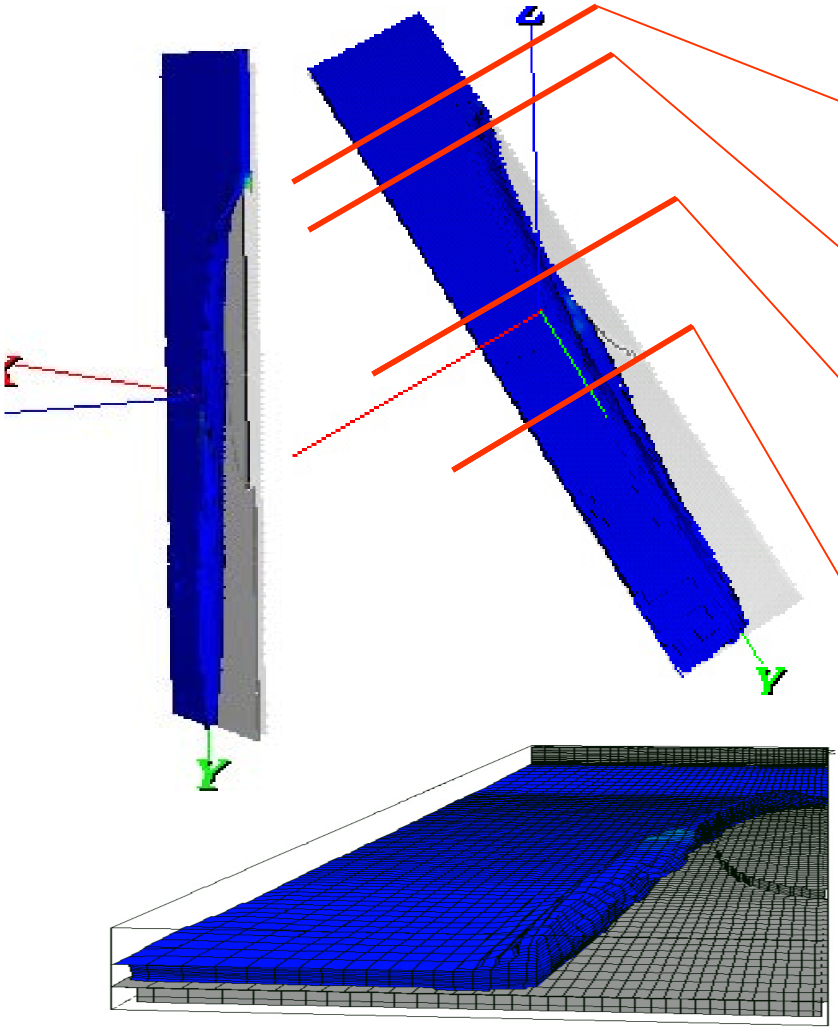
Fluid Wets the Structure.





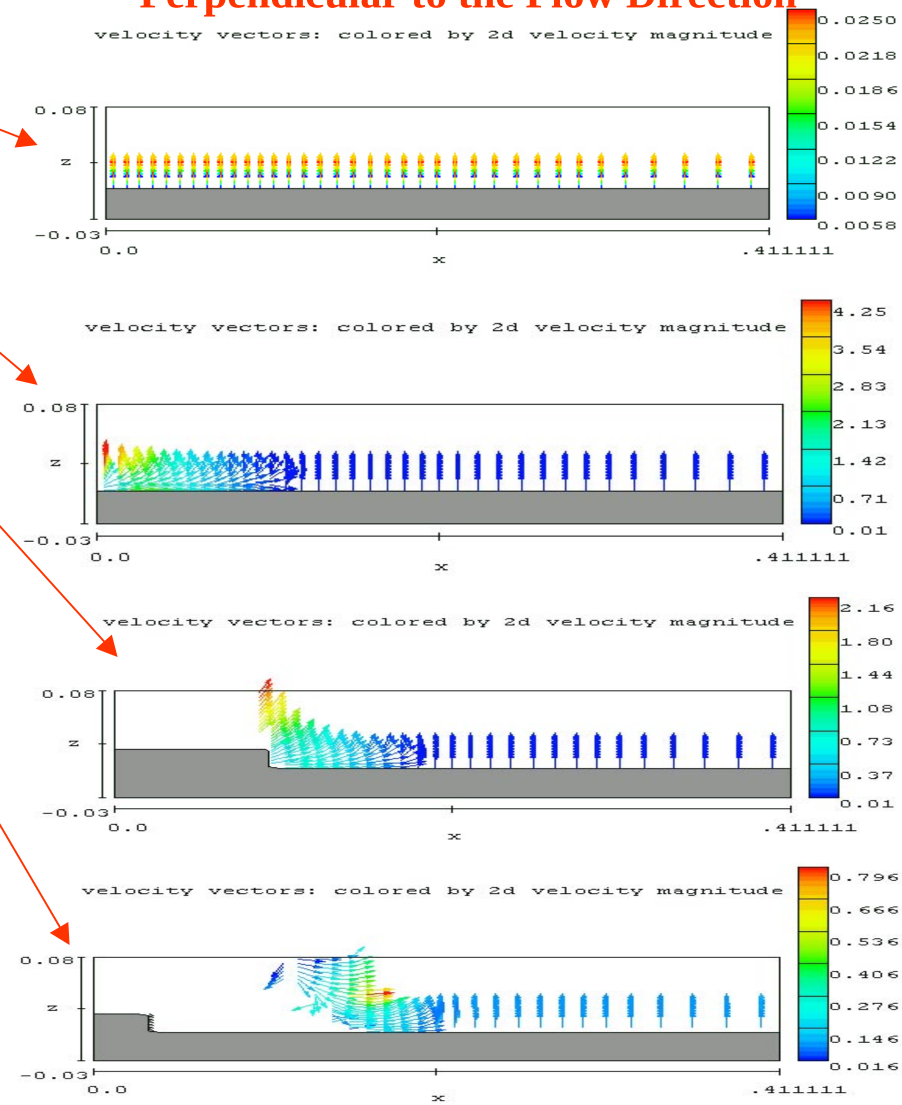
RESULTS OF 3-D TIME DEPENDENT CALCULATIONS FOR FLUID FLOW AROUND PENETRATIONS (For Initial Case)

3-D CFD Simulation Results



3-D View of the Wake Following the Penetration.

2-D Velocity Magnitude in Planes Perpendicular to the Flow Direction



POTENTIAL CHALANGES
IN LIQUID WALL BEHAVIOR AROUND PENETRATIONS

STAGNATION

- Minimizes the cooling of the front section of the penetration.
- Discharges fluid towards the plasma.

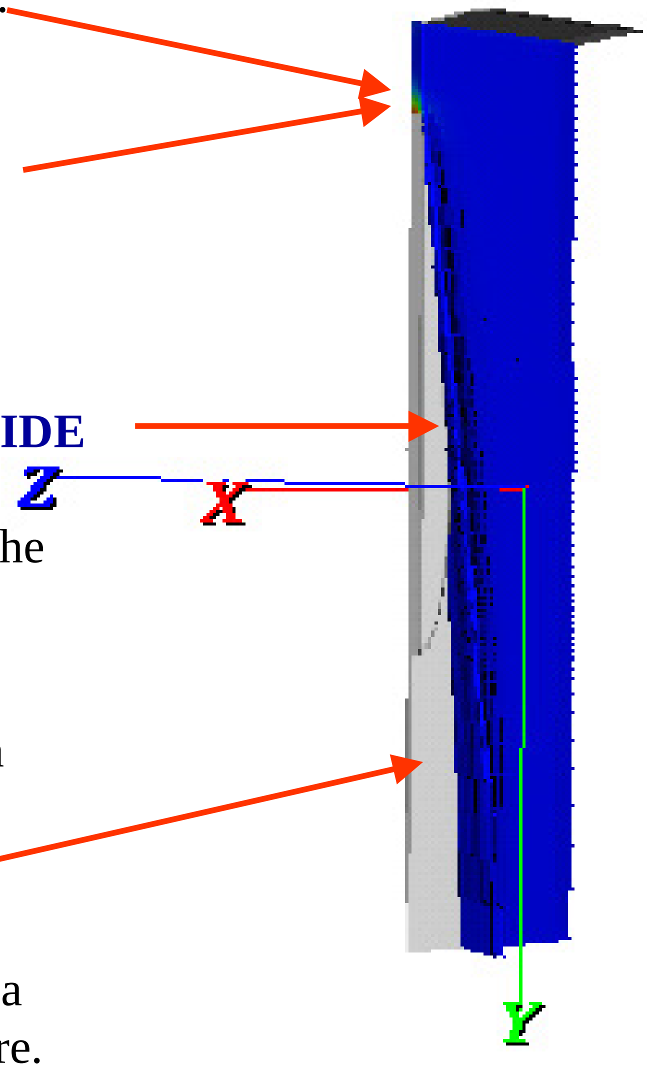
SPLASH OF THE FLUID AND DROPLET EJECTIONS

- Droplets may be generated and ejected into the plasma as the high velocity liquid layer hits the front section of the penetration.

FLUID LEVEL RISE SURROUNDINDG THE FRONT SIDE OF THE PORT

- A stream of rising fluid is diverted to the sides surrounding the penetration due to the obstruction of flow path.

  (144 m^3 of fluid per hour is displaced for a 20 cm wide (in the flow direction) penetration for the CLIFF concept with a base velocity of 10 m/s.)



WAKE FORMATION

- The wake formation at the end section of the penetration, as a result of deflection of streamlines by the penetration structure.

DESIGN GOALS

- Minimization the flow level rise in the vicinity of the side walls due to flow displacement.

- Elimination of stagnant flow section at the front of the penetration.

- Usage of active/passive methods to eliminate unwetted surface of the back wall.

- Covering the penetration side walls with moving fluid so that adequate cooling can be accomplished.

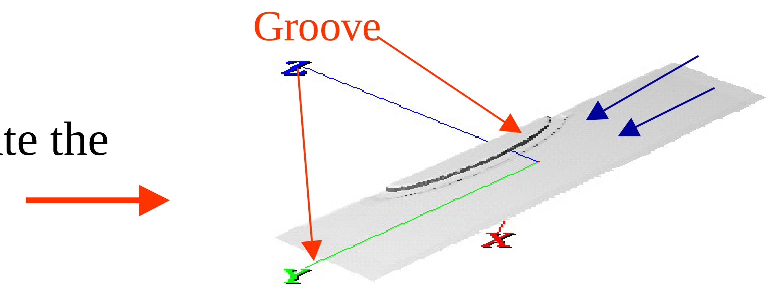
- Covering the front side of the penetration side walls (sides faced to the plasma) with fluid so that the penetration side walls may be used for longer operation time, due to elimination of thermal stress.

- Elimination of droplet formation and ejection of the fluid.

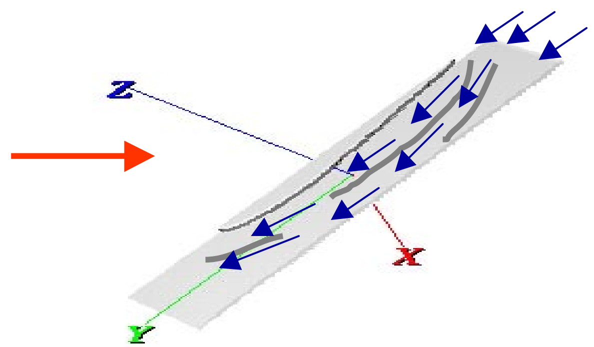
VARIUOS PROPOSED DESIGN MODIFICATIONS FOR PENETRATION ACCOMADITIONS IN LIQUID WALLS (I)

Passive Techniques

- Modification of the back wall contour to accommodate the additional fluid that is diverted on the sides by the penetrations.

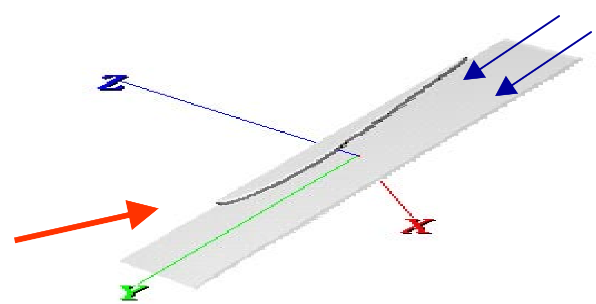


- Placing fins at various heights and angles parallel to the penetrations to divert the flow back to the end section of the penetration so that the unwetted region will be eliminated and the flow may have a more uniform height, as it diverted by penetration.



- Using a fluid divertor section in the upstream section of the penetration so that sudden fluid level rises can be minimized.

- Using a sharp edge at the front section of the penetration may minimize the stagnation.



VARIUOS PROPOSED DESIGN MODIFICATIONS FOR PENETRATION ACCOMADITIONS IN LIQUID WALLS (II)

Active Techniques

- Using a jet at the end section penetration that has the same velocity as the liquid wall flow, in order to cover the unwetted back wall behind the penetrations.

- Using various suction and flow removal mechanisms at the upstream and guiding them back to the downstream by external flow mechanisms and pumping

Design Approach

- minimize penetration's structural complicity and geometrical shape requirement.

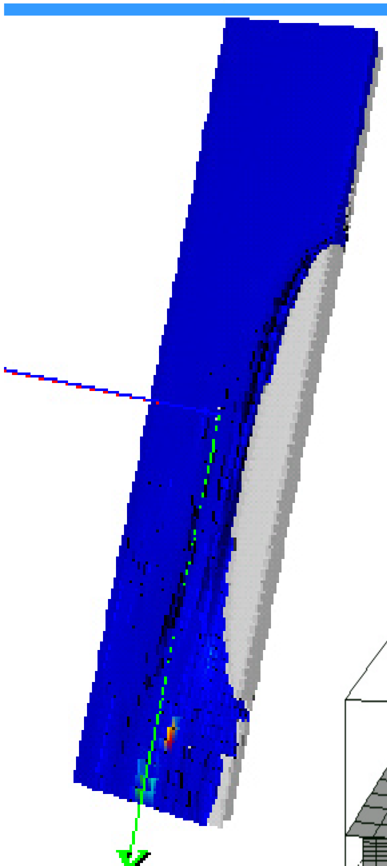
- obtain robust, reliable, continuos operation.

- minimize the perturbation to the hydrodynamic characteristics of the flow.

- eliminate the use of additional active mechanical systems.

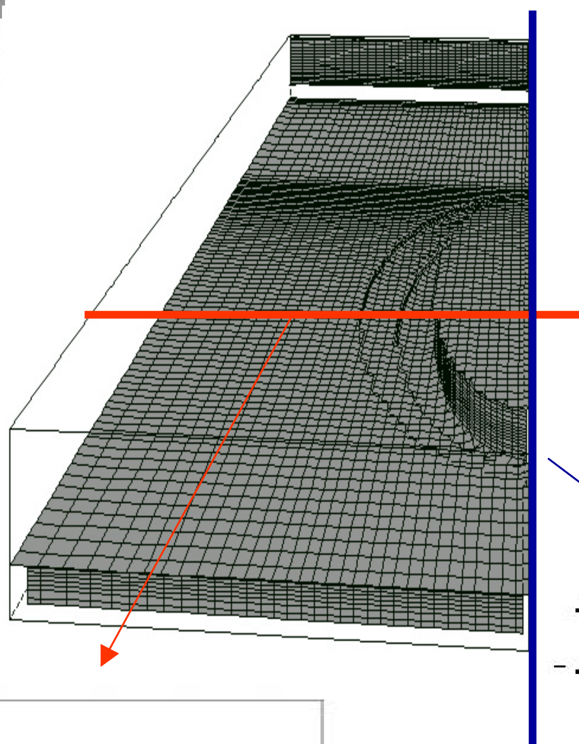
We are aware of active techniques but are not pursing them at the present time!

A PROPOSED DESIGN MODIFICATION
TWO LAYER GROOVE (I)



Two groove layers are used surrounding the penetration.

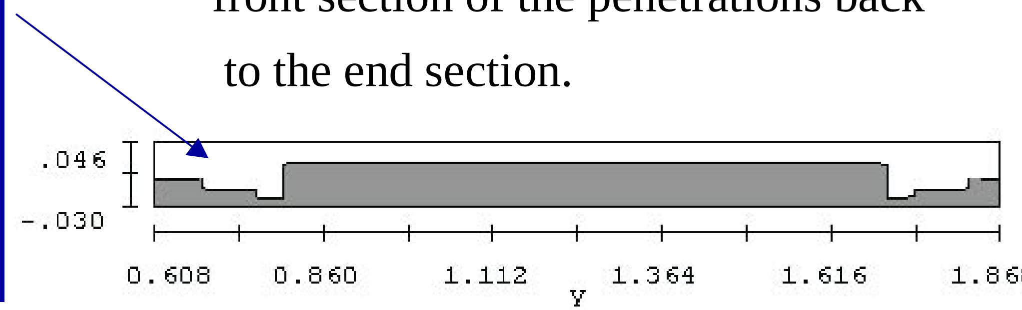
The first layer (closer to the penetration) is 5 cm wide and 2 cm deep, while the second layer is 5 cm wide and 1 cm deep.



Purpose:

a. To minimize a sudden fluid level rise by increasing the flow area.

b. To redirect the diverted flow at the front section of the penetrations back to the end section.

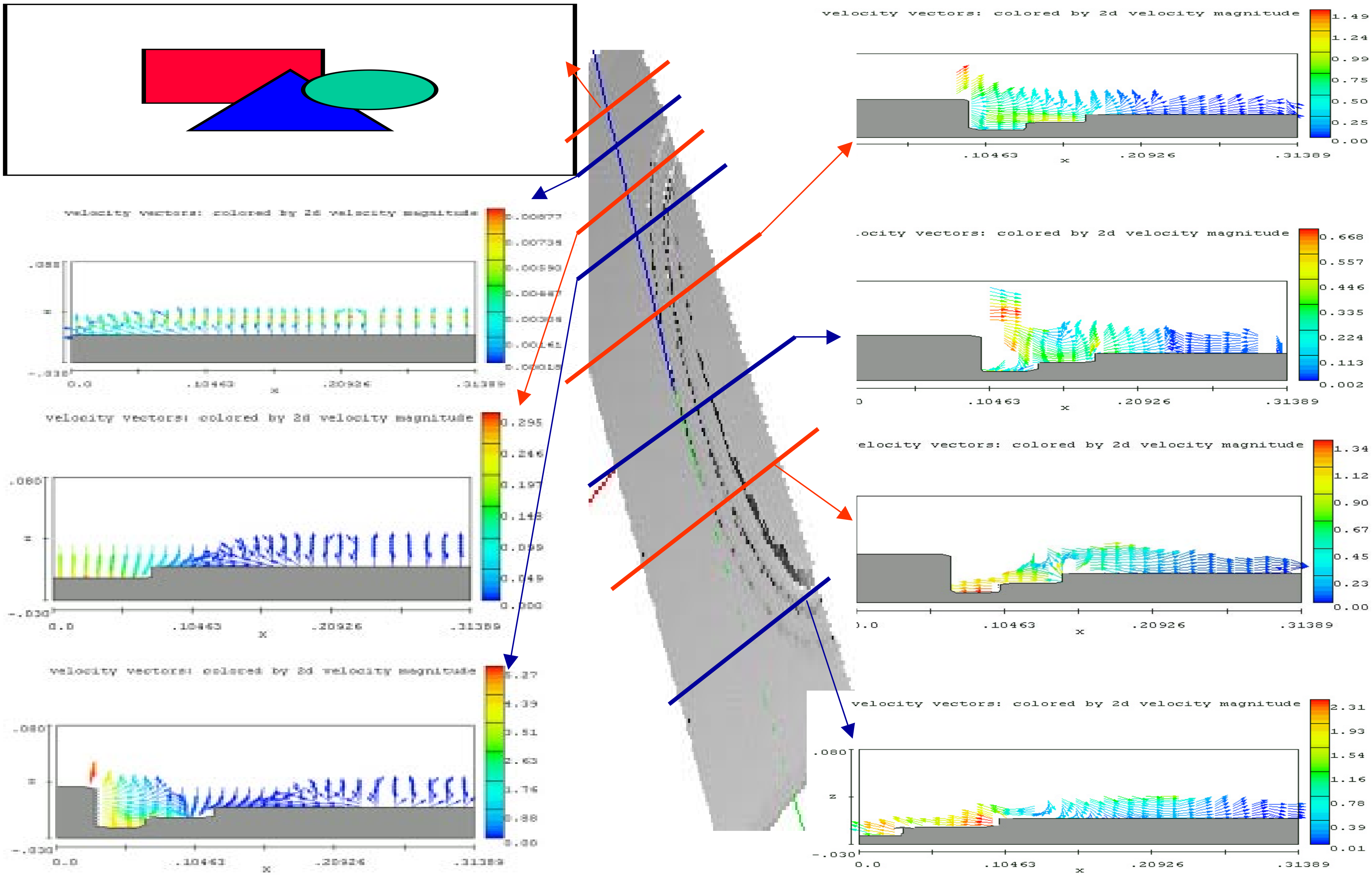


Area by the Penetration Parallel to the Flow Direction.

Area by the Penetration Perpendicular to the Flow Direction.

A PROPOSED DESIGN MODIFICATION

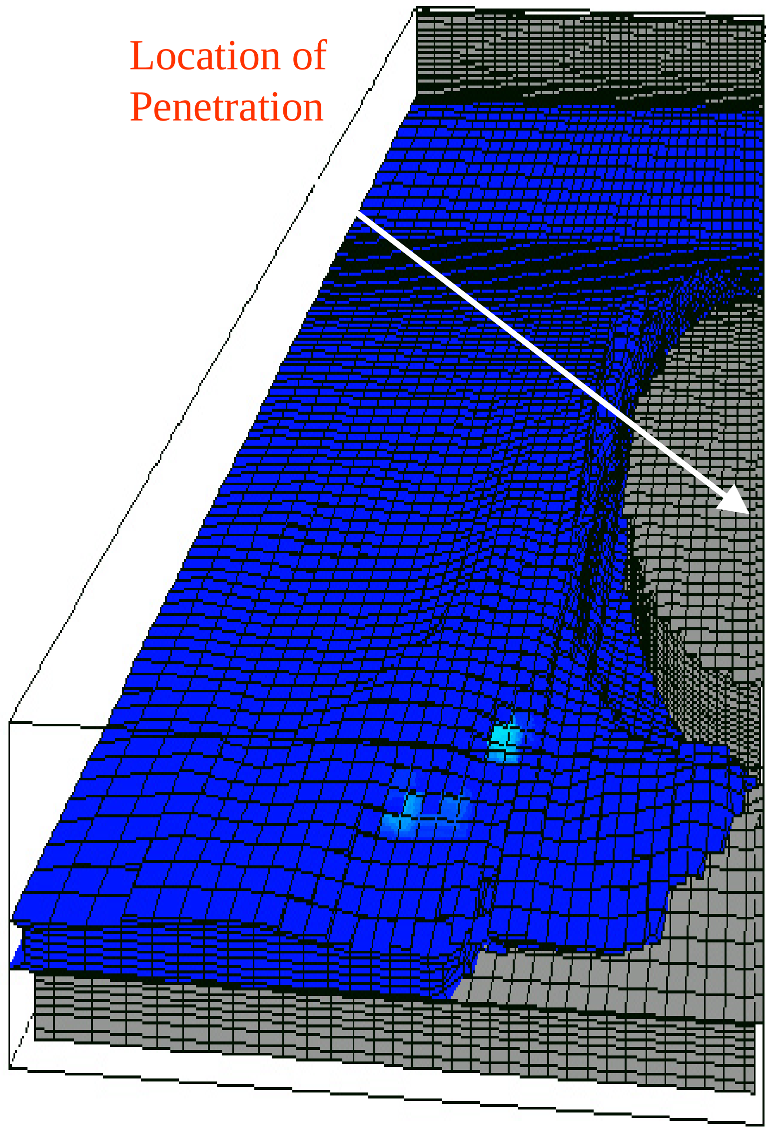
TWO LAYER GROOVE (II)



A PROPOSED DESIGN MODIFICATION
TWO LAYER GROOVE (III)

CONCLUSIONS

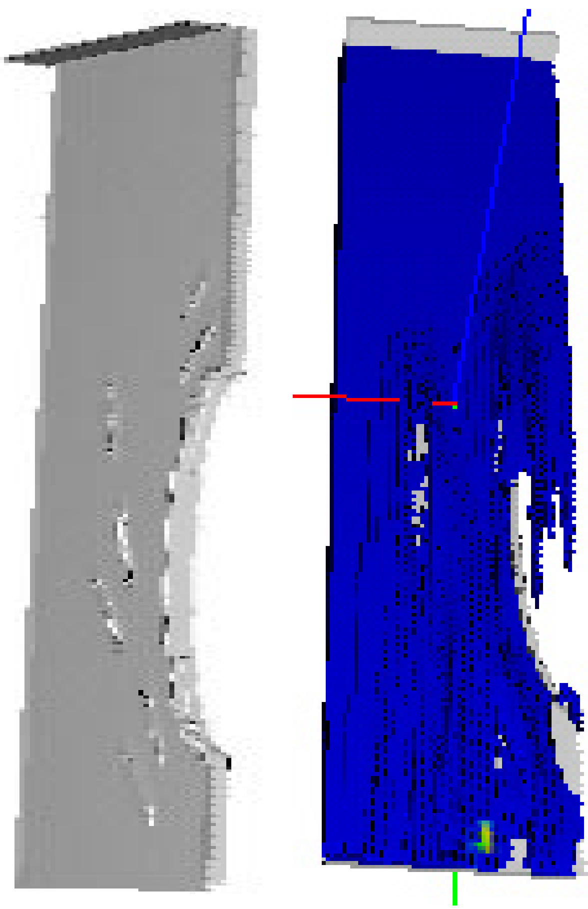
- This technique redirects the diverted flow towards the penetration's end section.
- Fluid level rise decrease as compared to the reference case (6 cm as compared to more than 8 cm).
- Cavities may from at locations where flow area changes suddenly as the flow proceeds towards to the steps of the grooves.
- A sudden rise of the back wall level, due to the step at the end of the groove section, may result in perturbation to the liquid wall flow.
- Corrosion of the groove side walls may decrease the life time and minimize the possibility of continuos operation.



A PROPOSED DESIGN MODIFICATION
FLOW GUIDER FINS (I)

* A series of flow guiders have been placed in the upstream, mid and downstream locations at very close distances to the penetrations.

Purpose:



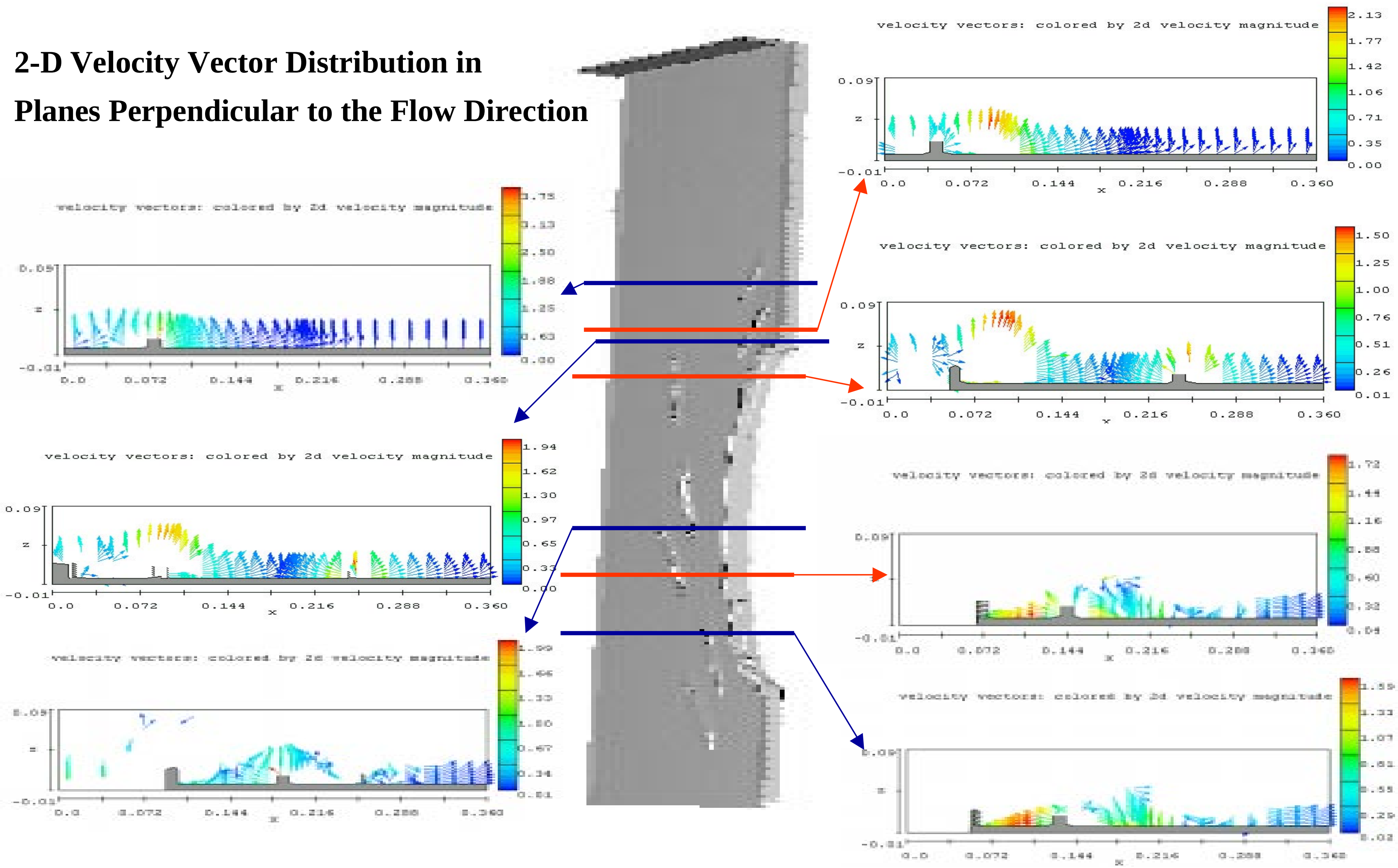
* The flow guiders located upstream are designed to gradually divert the fluid flow away from the penetrations in order to minimize sudden displacement of fluid as it hits the penetrations.

* The flow guiders located at the mid plane are designed to redirect the diverted flow towards the penetration and minimize the potential non-uniformity in the flow away from the penetration.

* The flow guiders located at the downstream of the penetrations are designed to divert the redirected flow towards the unwetted end section of the penetration.

A PROPOSED DESIGN MODIFICATION
FLOW GUIDER FINS (II)

2-D Velocity Vector Distribution in Planes Perpendicular to the Flow Direction



A PROPOSED DESIGN MODIFICATION
FLOW GUIDER FINS (III)

CONCLUSIONS

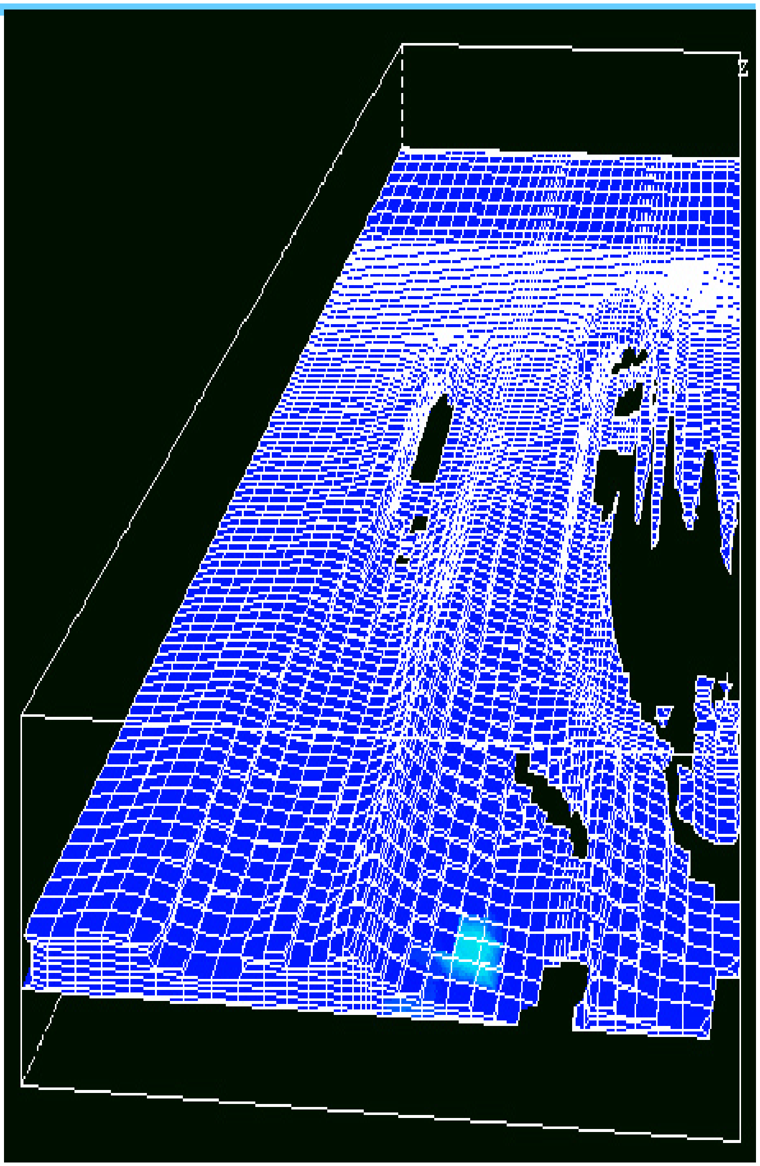
- The fins placed closer to the penetration cause sudden reflection of fluid and therefore splash.

- The height of the fins may cause unwetted regions and separation as well as uniformity of the wall thickness due to the wake formation behind the fin.

- The fins closer to the end section of the penetration do not cause fluid splash.

- The relative angle of the fins to the flow direction is very important in eliminating wake formations behind the fins.

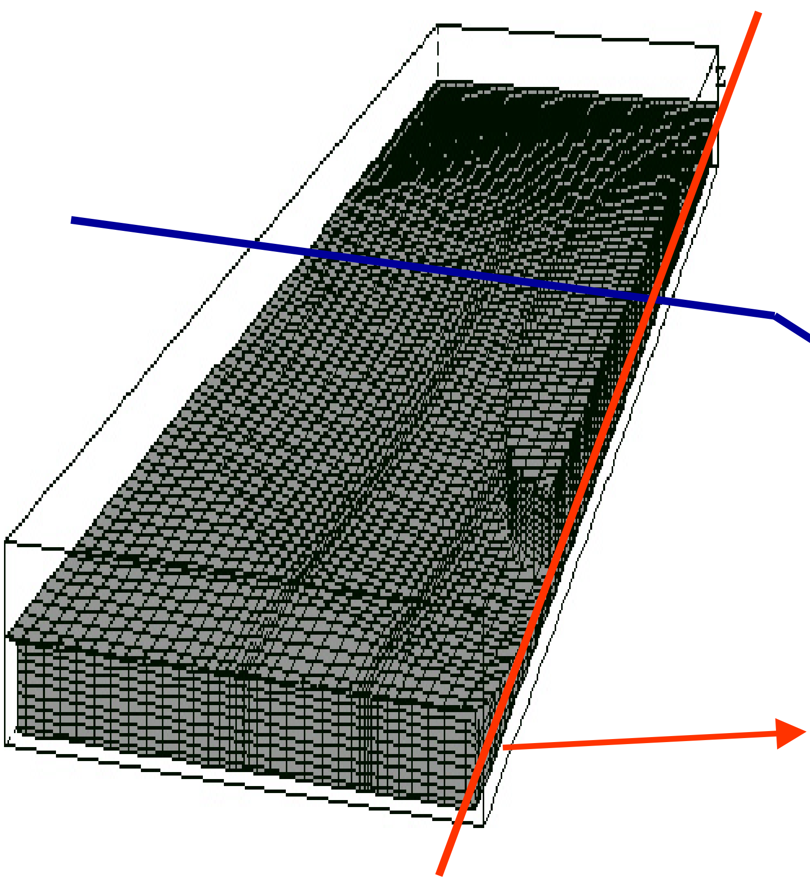
- High speed flow may cause corrosion in the fins and maintenance and replacement times may not be feasible.

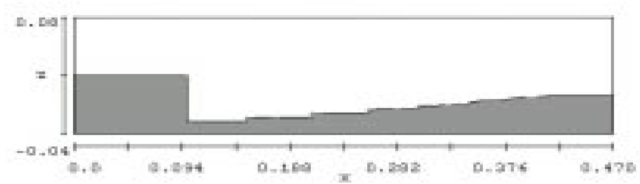


A PROPOSED DESIGN MODIFICATION
MODIFICATION OF BACK WALL TOPOLOGY(I)

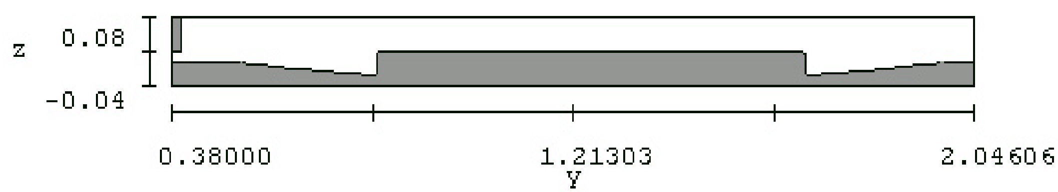
* The flat back wall has been modified to form a concave contour whose depth slightly varies away from the penetration, becoming deepest (2.2 cm deep) near the penetration.

* Gradually increasing depth of the concave contour towards the penetration minimizes the disruption of the flow height in planes perpendicular to the flow direction, while accommodating maximum additional flow area to the diverted fluid near the penetration.



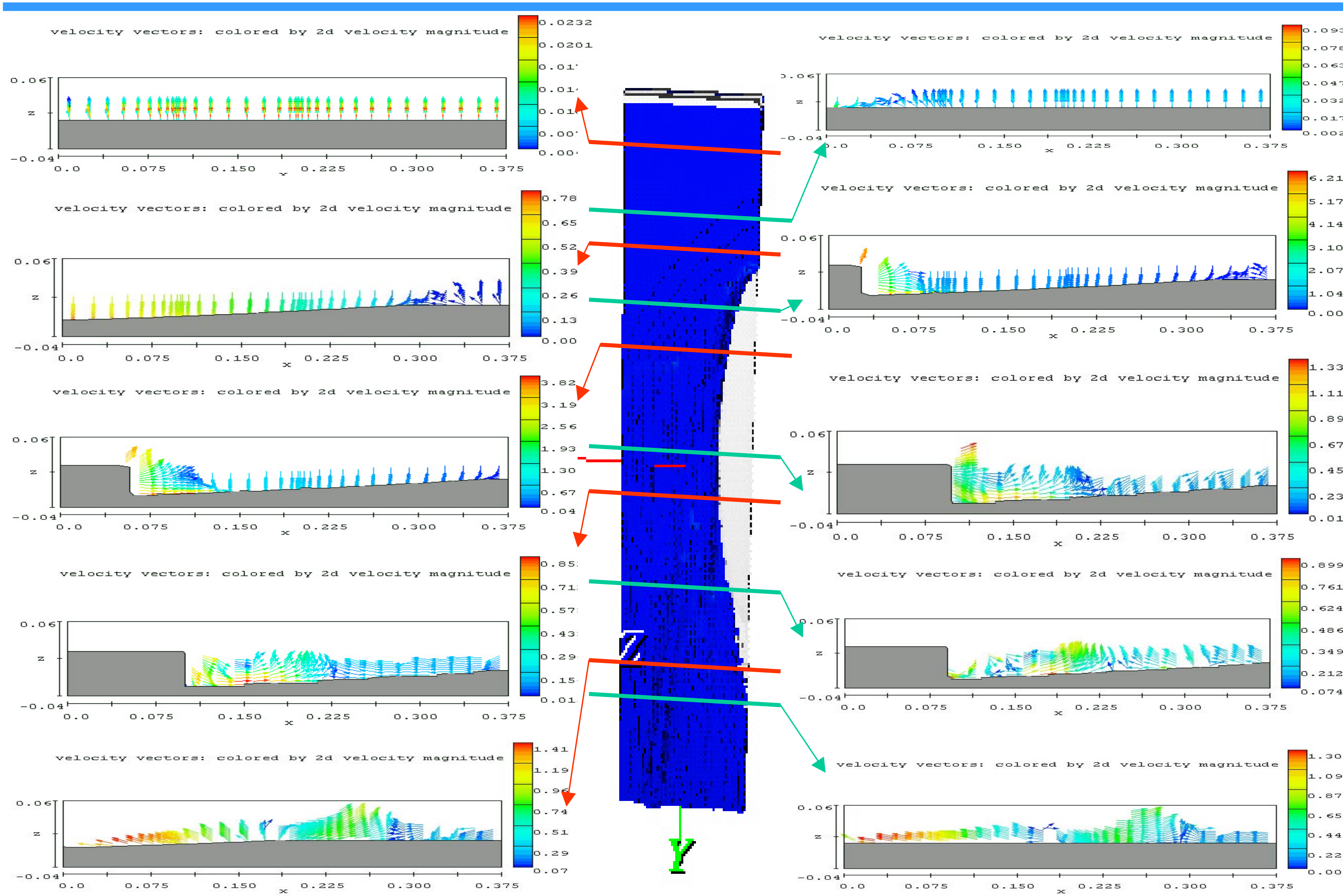


Area by the Penetration Perpendicular to the Flow Direction.



Area by the Penetration Parallel to the Flow Direction.

DESIGN SOLUTIONS, SUCH AS MODIFICATIONS TO BACK WALL TOPOLOGY RESULT IN MORE ATTRACTIVE FLUID FLOW CHARACTERISTICS AROUND PENETRATIONS



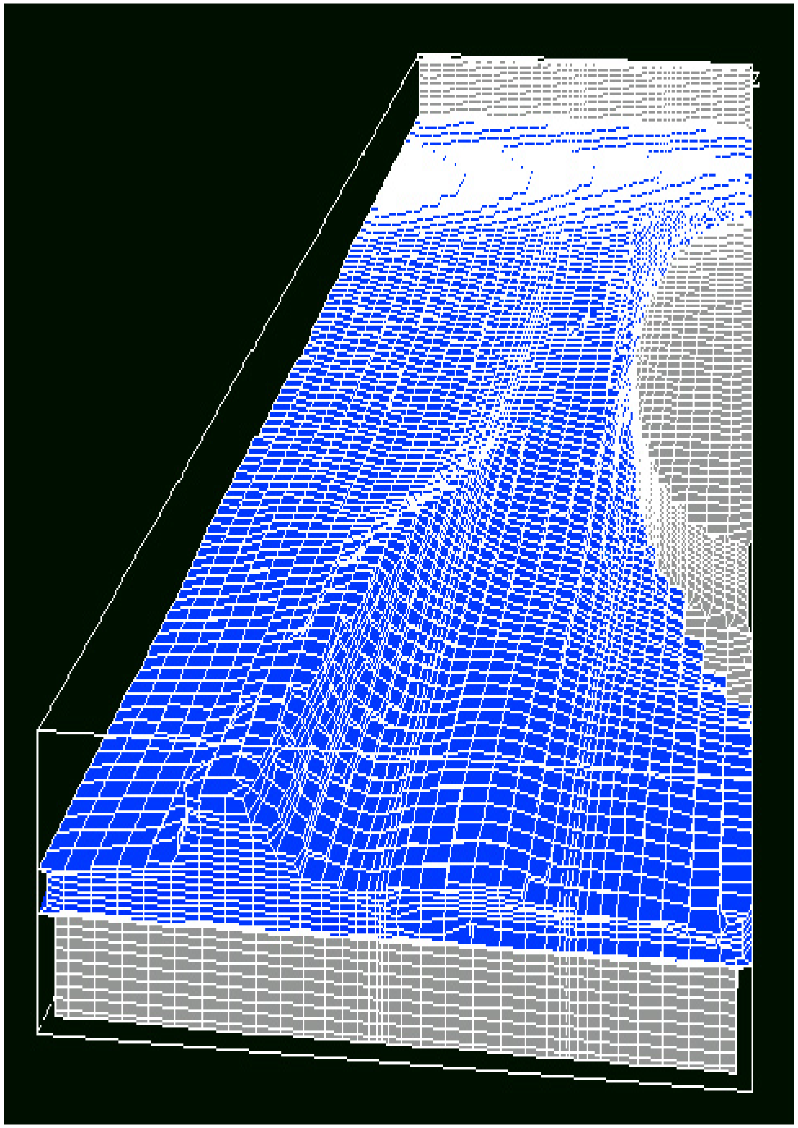
DESIGN MODIFICATIONS
MODIFICATION OF BACK WALL TOPOLOGY(III)

CONCLUSIONS

Using a gradually increasing depth towards to the penetration:

- eliminates fluid splash and droplet ejection.
- directs the fluid towards downstream of the penetration.
- accommodates excessive fluid due to sudden obstruction of the flow.

Further parametric analysis is required to obtain a uniform liquid wall thickness by modifying the concave contour depth and its distribution surrounding the penetration



CONCLUSIONS (I)

* Accommodating gradually increasing depth of the concave contour towards the penetration minimizes the disruption of the flow area and covers the unwetted section at the downstream of penetrations.

* Increasing the radius of curvature increases the centripetal acceleration that is perpendicular to the flow direction. This condition minimizes the fluid level change in the vertical direction and redirects it in the lateral direction.

* Problems arising at the front section of the penetration (stagnant flow section and droplet ejection) are independent of type of hydrodynamic liquid wall configurations and can be eliminated by sharp edge or gradually increase in the height of the front section.

CONCLUSIONS (II)

- **Suggested Optimum Port Location for FRC:**
  - The bottom section perpendicular to the gravitational acceleration minimizes the ejection of droplets towards the plasma.
  - Modification of the contour of the circular vacuum boundary wall slightly beyond where the flow passes the port (by forming a slight flow contraction in the radial direction) may generate a backward pressure which can eliminate wake formation.

- **Optimum Port Location for Tokamaks and ST Configurations:**
  - Use of vertical back wall is not suggested.
  - Increasing the wall curvature at locations where the ports are placed is recommended.
  - The lower half and near bottom section of the tokamak geometry may be an optimum location due to decrease in radial flow length and increasing the acceleration which perpendicular to the flow direction.

FUTURE WORK

Back wall surface topology will be optimized to accommodate penetrations while minimizing the disturbance to the flow and non uniformity of the flow height.

Implementing the optimized wall topology along with the optimized penetration shape to the concave back wall.

Temperature distribution on the penetration walls (with a specified thickness, radiation and neutron wall loading) will be analyzed using 3-D heat transfer analysis by taking into account of convective heat transfer between fluid solid and conduction in the penetration structure.