SCM ENGINEERING SERVICES

Technical Report on

Fluid-Structure Interaction Analysis in a Channel Flow causing Membrane Deformation

Executed By:

S Sunil Krishna
sunilkrishna.s@scmengg.com

S Sampath Kumar
sampath.s@scmengg.com

Technical Director

Dr. Subhash Deo
subhash@scmengg.com

SCM Engineering Services
Plot #156, First Floor,
Motilal Nagar,
Begumpet,
Hyderabad 500019,
India.
Ph: 040 – 65449094
Email: projects@scmengg.com
URL: http://www.scmengg.com

INTRODUCTION

Micro mechanical systems have gained large importance in the medical device technology today. The problems are often multi stream in nature and require two or more different types of analysis coupled together. The present study is concerning the structural deformation of a thin elastic membrane due to fluid forces in a micro channel. **This was accomplished in 50 hours for a leading American client.**

The study is on the effects of applied forces on a highly deformable membrane placed over a fluid domain for a valve type application. Fluid flow channel is with in the substrate. The deformation is a critical design parameter for the micro valve. **SCM Engineering Services** has performed a sequential coupled field analysis was carried out for the same.

OBJECTIVES

The study was divided into three parts.

(b) To find out forces exerted on membrane

(c) To find out membrane deformation

METHODOLOGY

The study was divided into following two parts and finally coupling them to do fluid structure coupled analysis together.

(A) Fluid flow analysis considering fluid region only for the given geometry. Inlet velocity is specified as an input to problem.

(B) Structural analysis of substrate-membrane, considering pressures obtained from fluid analysis as input and finally coupling them to do fluid structure coupled analysis together.

ASSUMPTIONS USED IN THIS ANALYSIS

The following assumptions are used while solving the fluid flow analysis problem.

a) The flow is treated as steady state and incompressible and hence constant density for water is used.

(a) To simulate fluid-membrane interaction

b) The problem is setup with plugged flow inlet boundary condition and turbulence effects are captured using the standard k-e model with the turbulence intensity of 1% at inlet and characteristic length equal to one order less than inner hydraulic diameter of inlet cross-section.

The following assumptions are used while solving fluid structure interaction coupled analysis problem.

a) Material for substrate-membrane is considered as linear, elastic, isotropic.

b) Geometric non-linearity is considered in the present analysis.

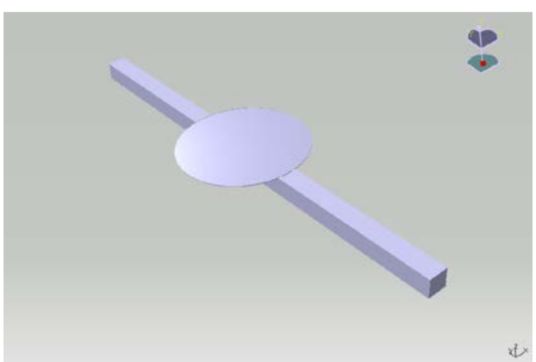

Fig.1 CAD model of the fluid domain

HARDWARE AND SOFTWARE USED

SOFTWARE

The analysis activity consists of Mesh generation, Pre–processing, Fluid-Structural analysis and Post-processing. Mesh generation was done using ANSYS. The pre & post processing - for setting problem and analyzing results, Considering present geometry topology quality requirements, HYPERMESH is used for mesh generation. Accordingly, all surfaces of fluid walls and corresponding curves are identified and picked from geometry for meshing the Fluid flow region. Hex mesh is chosen for present analysis. Same pattern of mesh is extended to remaining structural region with hex shape solid element. All necessary care has been taken to ensure that fluid and structural elements are of good quality with the aspect ratio between 0.5 to 1.0. Elements close to wall region, in fluid analysis, are carefully adjusted to capture the boundary layer effect. The qualities of the elements maintained were by strict standards used in **SCM Engineering Services**

RESULTS AND DISCUSSIONS:

Material properties were then assigned to the three entities.

The analysis was first carried out with only the fluid flow. The flow pressures were evaluated on nodes and were sequentially mapped onto the substrate and membrane. The flow characteristics were noted and also the membrane deformation was found as required.

The results were then compared to simplified analytical results for accuracy.

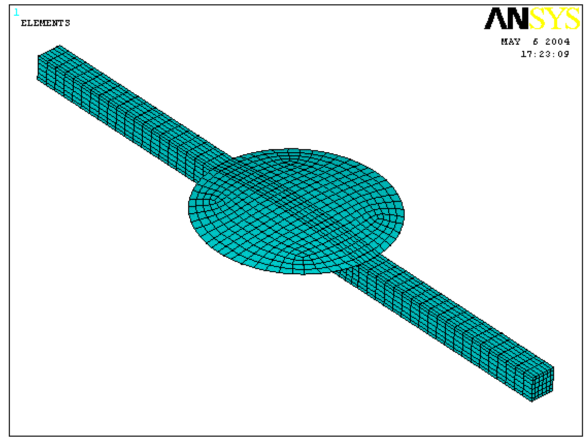

Fig.2 Fluid domain mesh

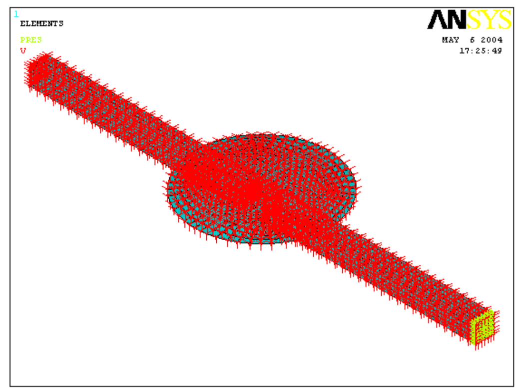

Fig.3 Flotran boundary conditions

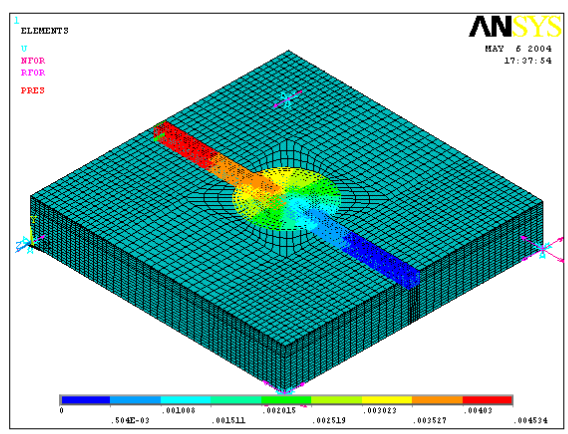

Fig.4 Coupled loading on membrane

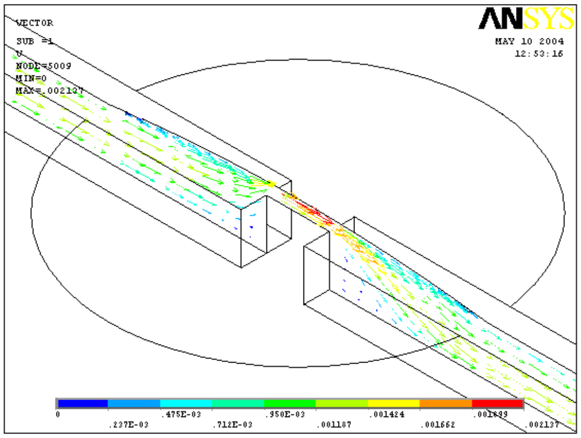

Fig.6 Flow analysis

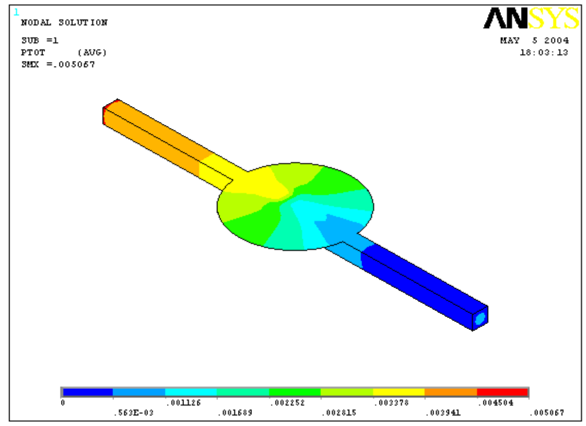

Fig.5 Pressure contours for one of the cases

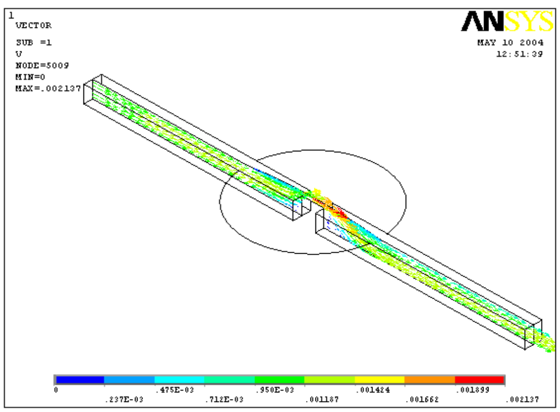

Fig.8 Velocity vectors

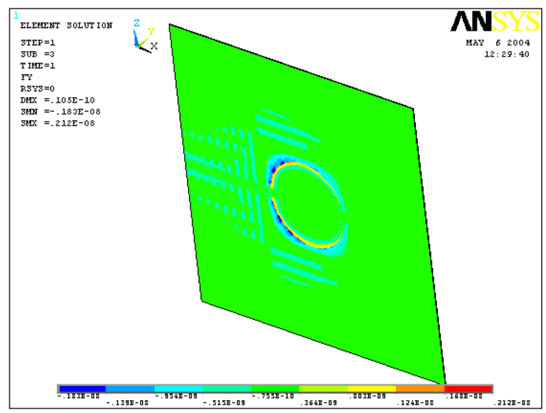

Fig.7 Deformation of membrane

CONCLUSION

The results from the analysis can be used to improve on the design at an early stage. Necessary changes can be made in properties or geometry before manufacturing.

.

The required flow characteristics can be understood better and the design can be improved

REFERENCES

ANSYS INC theory and reference

HYPERMESH tutorials