A 3D-CFD CODE FOR ACCURATE PREDICTION OF FLUID FLOWS AND FLUID FORCES IN SEALS

M.M. Athavale and A.J. Przekwas
CFD Research Corporation
Huntsville, Alabama

and

R.C. Hendricks
NASA Lewis Research Center
Cleveland, Ohio

ABSTRACT

Current and future turbomachinery requires advanced seal configurations to control leakage, inhibit mixing of incompatible fluids and to control the rotodynamic response. In recognition of a deficiency in the existing predictive methodology for seals, a seven year effort was established in 1990 by NASA's Office of Aeronautics Exploration and Technology, under the Earth-to-Orbit Propulsion program, to develop validated Computational Fluid Dynamics (CFD) concepts, codes and analyses for seals. The effort will provide NASA and the U.S. Aerospace Industry with advanced CFD scientific codes and industrial codes for analyzing and designing turbomachinery seals.

An advanced 3D CFD cylindrical seal code has been developed, incorporating state-of-the-art computational methodology for flow analysis in straight, tapered and stepped seals. Relevant computational features of the code include: stationary/rotating coordinates, cylindrical and general Body Fitted Coordinates (BFC) systems, high order differencing schemes, colocated variable arrangement, advanced turbulence models, incompressible/compressible flows, and moving grids.

This paper presents the current status of code development, code demonstration for predicting rotordynamic coefficients, numerical parametric study of entrance loss coefficients for generic annular seals, and plans for code extensions to labyrinth, damping, and other seal configurations.

INTRODUCTION

Advanced rocket and aircraft engines are aimed to work at progressively higher efficiencies, lower weight-to-thrust ratios, to have greater reliability, and a longer life. Turbomachinery, the most complex engine component, requires advanced seal configurations to control leakage, rotodynamic response, mechanical component tolerances, and to inhibit mixing of incompatible fluids. As all of the above affect engine performance, through parasitic losses or limit cycles, it is important to understand the fundamentals of flow and rotodynamics and their interdependence. As modern turbomachinery becomes lighter and is operated at increasingly higher speeds the influence of seals on the rotodynamic stability becomes an important design concern. Prime examples of the importance of seals are in the Space Shuttle Main Engine (SSME) Fuel and Oxidizer turbopumps where seals performance becomes a critical element for the stable operation of engines. Coordinated seals research efforts of NASA LeRC, MSFC and Texas A&M University have greatly contributed to the Shuttle's reliability and performance. It is realized that accurate prediction of leakage rates and seals rotodynamic coefficients requires advanced computational models. Geometrical complexity and extreme operating conditions of uncommon fluids pushed existing empirical and bulk flow models to their limits. In 1988, Tam, Przekwas, Muszynska, Hendricks, Brown and Mullen[1] demonstrated that CFD can provide new insight into complex flow through seals and can complement the analysis of rotodynamic instability of seals.

The basic objectives of this program follow.

1.   To develop a verified three-dimensional "scientific" CFD code for analyzing seals, representing the state-of-the-art in accuracy of predicting flow fields, rotodynamics and performance of a wide variety of seal configurations. The code will provide insight into flow and dynamics, contribute to the technical data base and provide an accurate standard for less complex "industrial" codes.

2.   To compile and generate a set of verified one-, two-, and simplified three-dimensional "industrial" codes that will enable the designer or researcher to expeditiously analyze the wide variety of seals.

3.   To develop a modern seal design computational environment with Knowledge Based System (KBS) and links to CAD/CAM packages and scientific data visualization tools.

The project team members include NASA LeRC, Mechanical Technology Incorporated, developing industrial codes and KBS, and CFD Research Corporation developing a scientific CFD seals code. A parallel research effort is being conducted in which the scientific code will contribute to the technical data base and provide an accuracy standard for less complex codes. The industrial codes will provide speedy analysis of seal designs. Graphical User Interface (GUI) and KBS data base will be a unifying mechanism. The technology transfer will be accomplished via annual workshops at which the program workplan will be reviewed and revised.

The aim of the code development effort is to develop a steady/transient 3D CFD code for the analysis of fluid flow for a broad range of seal configurations and for the analysis of the dynamic forces on the seals. The geometrical seal configuration will include cylindrical, labyrinth, honeycomb, damper, brush, face, wave, grooved, tip, contact, and noncontinuous seals. The fluids will range from incompressible liquid, two-phase mixtures, to compressible gases. An existing 3D CFD code, REFLEQS[2], is being adapted for seals by incorporating physical models to test turbulence, non-isotropic wall roughness, phase change, and cavitation. The scientific code was demonstrated at the 1992 annual workshop in August 1992 at NASA LeRC.

CFD CODE ENVIRONMENT

Existing CFD methodology of turbomachinery seals design is an expensive and time consuming process. Geometry definition, grid generation, selection of boundary conditions, numerical parameters selection, post-processing, and analysis of results often require CFD expert involvement. Rapidly advancing computer system software and hardware capabilities allow the automation of the CFD-based design process. In order to develop CFD-based seal design environment integration of CAD/CAM, CFD, structural mechanics, rotodynamics, computer graphics under the Knowledge Based System (KBS) is currently conducted.

The tasks of KBS are to integrate scientific and industrial codes with a consistent database, to provide a user-friendly graphical interface with context-sensitive help and to provide the Expert System to guide the seal design and optimization process. A major component of the KBS environment will be the scientific CFD code. The outline of the structure and brief description of the components of CFD seals code are provided. Technical specifications of the CFD code capabilities are discussed in the next section.

Figure 1 presents a flow chart for the CFD code design environment. The centerpiece of this system is the CFD code and most of the current development effort is dedicated to it. The other major modules CFD GUI-Preprocessor and KBS/GUI are being developed in collaboration with Mechanical Technology, Inc. The remaining components, CAD and Visualization, will be available from commercial vendors and the convenient links to them will

138

be provided. Comprehensive geometries definition and algebraic grid generation will be provided within GUI-Preprocessor. For more complex geometries, links with a CAD package such as ICEM-CFD, PATRAN, or IDEAS will be available. REFLEQS code, the starting point for the seals code, has already established interfaces with PATRAN and ICEM-CFD codes. Grids in PLOT3D standard format will be used directly without any translation. Similar standards will be adopted for graphical interfaces to scientific visualization tools. PLOT3D format files for grid, vectors, and scalars will be generated for postprocessing with PLOT3D, FAST or other commercial packages, most of which support that standard. The CFD code system can be installed on any UNIX X-window based workstation or on mainframe or mini-supercomputers.
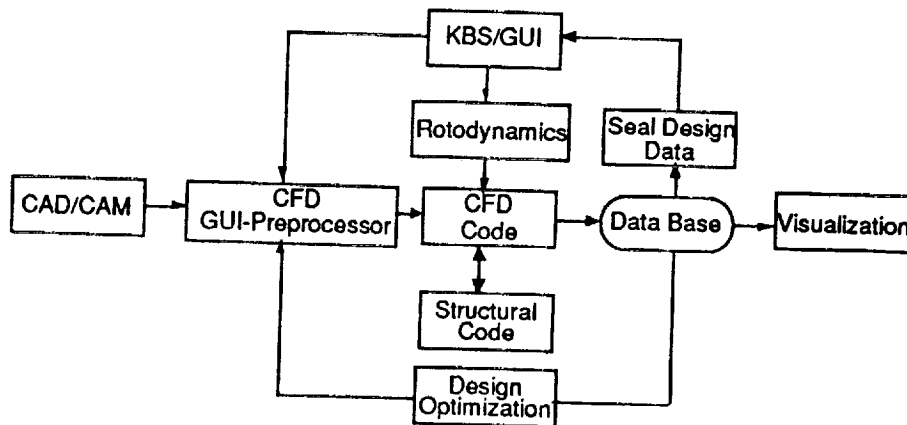

Figure 1.   Flow Chart of CFD Code Design Environment: KBS - Knowledge Base System; GUI - Graphical User Interface

CFD CODE CAPABILITIES

The CFD Scientific Code is to be used as an accuracy check for the industrial design codes, and to provide detailed flow information for a variety of seals. It will also be used to predict the details of flow and performance of new seal configurations as well as seals where inadequate testing has been performed. With this in mind, a scientific code must possess following features as a minimum:

1. 3-D Navier-Stokes analysis in generalized body-fitted coordinate grids;
2. Stationary and rotating coordinate frames;
3. Steady-state and time-accurate solution capability;
4. Incompressible and compressible flow solutions;
5. Variable physical properties (viscosity, density, specific heat etc.);
6. Cavitation effects;
7. Provision for stepped surfaces and injection ports;
9. Inclusion of viscus dissipation in energy equation;
10. Treatment of sources due to external fields;
11. Variable surface roughness treatment;
12. Provision for effects of pre-swirl and upstream effects;
13. Customized input and output features for seals; and
14. Calculation of rotordynamic coefficients;

The code will use solution procedures that are accurate, efficient and robust when used in the high-aspect ratio computational cells that are typically encountered in seal problems.

As a starting point for the scientific code, REFLEQS[2] (REactive FLow EQuation Solver) code was selected. This code was developed by CFDRC personnel under various NASA and DoD grants as well as IR&D funds. Several modifications were subsequently incorporated in this

139

code. The current version of the code is a 3-D Navier-Stokes flow solver which uses a pressure-based flow solution method. A finite volume method is used to discretize the solution domains. A colocated arrangement of variables is used, where all flow variables are stored at the cell center. Cartesian components are used as the primary velocity variables, and a strong conservation approach is used to calculate convective fluxes. The momentum and continuity equations are solved sequentially using a variation of the SIMPLEC method. Some of the features of the code are listed below.

1. High-order spatial differencing schemes including central-differencing and several third-order schemes such as Osher-Chakravarthy, Osher, Superbee, Minimod *etc.*

2. Second-order temporal differencing: Crank-Nicholson and three-point backward differencing.

3. A comprehensive set of boundary conditions: specified mass flux, specified pressures, wall and symmetry. Some seal related conditions include specified total pressure condition with inlet loss factor, rotating and whirling wall boundary, and specification of swirl velocity.

4. A variety of turbulence models including the High-Re model with wall functions, low-Re model, and the multiple scale model of Chen.

Two different methods have been incorporated to calculate the skew-symmetric set of rotordynamic coefficients associated with a centered rotor. In the first method, the rotor is assumed to undergo whirl in a circular orbit about the stator center[1]. A transformation to a frame whirling with the rotor renders the problem quasi-steady. The fluid pressures on the rotor are then integrated to provide the fluid restoring forces at several different whirl frequencies. Appropriate polynomials are then fitted to obtain the functional relation between the fluid forces and the whirl frequencies to yield rotordynamic coefficients.

The second method is based on the "external shaker" method used in experimental setups for seal dynamic characteristics[3]. In this method, the rotor center is moved along a straight line in a direction perpendicular to the rotor centerline. A sinusoidal function of time is prescribed for this motion. The resulting time-dependent pressures are integrated to provide the rotordynamic coefficients. Due to the linear rotor motion, the problem is time-dependent, and also involves a time-varying computational domain. A volume-conserving moving grid formulation available in the code is used to compute the flow (the moving grid formulation can also be used for flows where the rotor center describes arbitrary motions). The time-dependent fluid forces are then used to calculate the rotordynamic coefficients. A detailed description of the method is given in Reference 4 . Of the two methods, the first method is faster, but can be used only for the centered rotor coefficients. The moving grid method is more versatile, and can be used to calculate dynamic coefficients for seals with static eccentricity as well as rotor misalignment.

SELECTED RESULTS

Validation of the scientific code is essential for its use in the design environment. It is a time-consuming process, and should be carried out in a step-by-step manner. This process involves computing flow solutions for a variety of test problems, and comparing the numerical results with the corresponding analytical/experimental/numerical results. Four levels of validation problems were suggested by Singhal[5] to be necessary for overall validation of the code, and to ensure code reliability. These are:

1. checkout problems;
2. benchmark problems;
3. validation problems; and
4. field problems.

The CFD scientific code has been put through an extensive array of test problems to assess and ensure the accuracy of the numerical and physical models incorporated in the code. Several checkout problems on relatively simple grids were carried out to ensure mass conservation, invariance of solutions in different reference frames, uniformity of the boundary condition treatment (i.e. consistent treatment at all planes), accuracy of the boundary treatment, proper computation of grid metrics, velocities in both orthogonal and non-orthogonal BFC grids, and so on. An exhaustive list of these test cases is too long; instead a selected number of test cases are given below. These cases test the accuracy of numerical and physical models that are directly relevant to seal type problems. Most of the test cases fall into the first three levels of validation outlined in the previous paragraph.

1. Fully-developed and developing flow in pipes, channels, and long narrow annulii.
2. Laminar flow between rotating cylinders, with and without Taylor vortices[6].
3. 2-D driven cavity flow, (Re up to 10,000[7]); 3D driven cavity flow.
4. Couette flow with and without heat transfer; planar wedge flow in a slider bearing[6].
5. Laminar flow over a backstep[8].
6. Laminar flow in a square duct with a 90° bend[9].
7. Turbulent flow in a plane channel[10].
8. Turbulent flow induced by rotating disk in a cavity*[11].
9. Centripetal flow in a stator-rotor configuration*[12].
10. Turbulent flow in an annular seal[13].
11. Turbulent flow in a 7-cavity labyrinth seal[14].
12. Turbulent compressible flow and heat transfer in turbine disk cavities[15].
13. 3-D driven cavity flow with lid clearance and axial pressure gradient and control of flow through vortex imposition[16].
14. Flow in infinite and finite length bearings (without cavitation)[17,18].
15. Rotordynamic coefficient calculations for incompressible flow seals[19,20] and compressible tapered seals[21].

(* Computations performed under NASA MSFC Contract No. NAS8-38101)

Presented below are some of the results described in Item 15. These consist of rotordynamics for a short[19,3] and long[20] incompressible flow seals, and tapered gas seals.[21]

In most of the numerical simulations of seal flows the seal entrance static pressure is linked to the experimentally measured pressures through an entrance loss coefficient. This is expressed as:

$$P_a = P_{in} - 1/2 \ (1 + \xi \ ) \rho u_a^2$$

where ξ is the loss coefficient, P_{in} the experimentally measured static pressure upstream of the seal, P_a the static pressure that is imposed at the seal entrance and u_a the average axial velocity in the seal. Thus, the entrance loss coefficient is expressed as a fraction of the average dynamic head in the seal, and represents the loss in head as the fluid enters the seal from an upstream region that has a much larger radial width. A series of cases were computed in order to estimate these losses using the present code under different flow and geometry conditions. These results are also described below.

Tapered annular gas seal: This case is described in Nelson et.al.[21]. The seal parameters used were: seal radius= 32.5 mm., radial clearances at the inlet and exit = 0.172 and 0.086 mm, inlet pressure = 1.52 MPa, sump pressure = 0.65 Mpa, inlet temperature = 650 K, γ =1.4, dynamic viscosity = 1.8 X 10-5 Pa-s, and gas constant=2586 J/Kg-K. The shaft speed was 30,400 rpm, and calculations were performed at three L/D ratios of 0.1, 0.2 and 0.4. The moving grid method also was used in this case. Standard k-ε model was used for turbulence,

141

and the computational grid used had 12, 6, and 30 cells in the axial, radial and circumferential directions. The results for this case are shown in Table 1. Also shown are the results from bulk flow theory as reported in Reference 21. As seen, the two sets compare reasonably well, including the prediction of flow choking at the exit of the short seal.

Table 1. Tapered Gas Seal Rotordynamic Coefficients

L/D	K N/m	k N/m	C N-s/m	c N-s/m	Exit Mach Number
0.1	948000	19700	11.5	0.016	1.00
	1150000	15429	9.94	0.043	1.00
0.2	1700000	75700	43.9	0.073	0.96
	2125500	60886	38.62	0.09	0.97
0.4	2880000	267000	152	0.27	0.83
	3553200	233820	145.9	0.57	0.83

☐ Nelson, 1985[21]   ☐ Present Results

PA-92-24 11

Incompressible long seal: The seal geometry for this case, as described in Reference 20 is : seal radius = 39.656 mm, clearance = 0.394 mm, length = 240 mm. Properties of water at 20°C were assumed: dynamic viscosity = 1.04 x 10^{-3} Pa-s, density = 998 Kg/m^3. The grid used had 20, 15, and 30 cells in the axial, radial, and circumferential directions, respectively. The low Reynolds number k-ε turbulence model was used in the calculations. The whirling rotor method was used to calculate the rotordynamic coefficients. The calculations were performed at three shaft speeds: 1080, 1980 and 3000 rpm. Six pressure differentials for each speed were used: 20, 50, 10, 250, 500 and 900 KPa. Inlet swirl was specified using the experimental data. Results of these calculations are plotted in Figures 2 through 6. Experimental data from Reference 20 are also plotted on these figures for comparison. The comparison between the experimental and computed values is very good; the negative direct stiffness coefficients at high rpm and/or low pressure differentials are also predicted correctly.
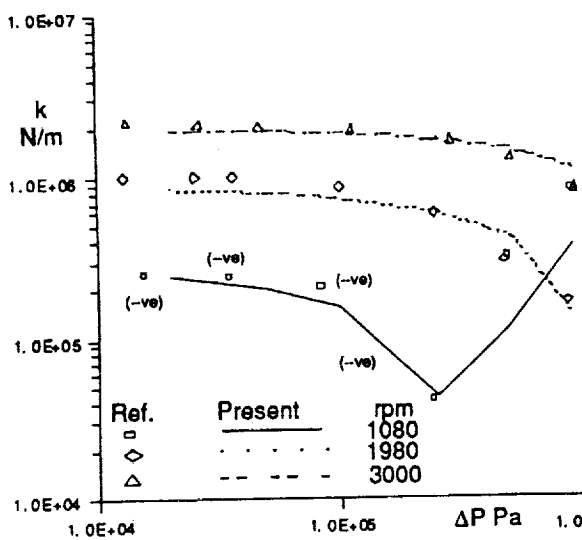

Figure 2.   Direct Stiffness Coefficient. Symbols represent experimental data from Reference 20.
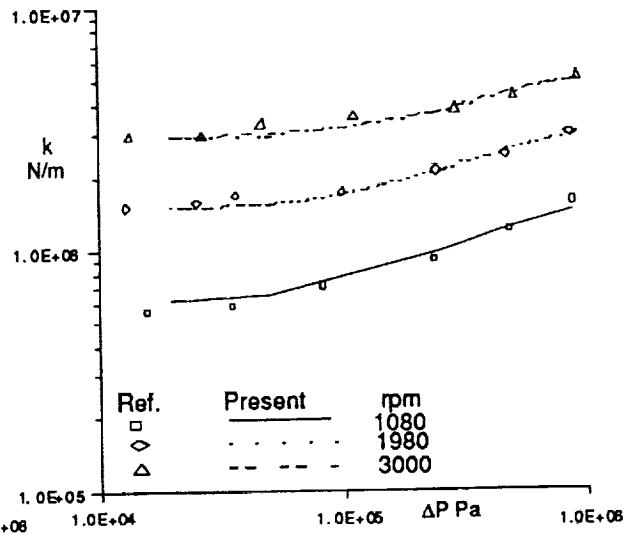
Figure 3.   Cross-Coupled Stiffness Coefficient. Symbols represent experimental data from Reference 20.
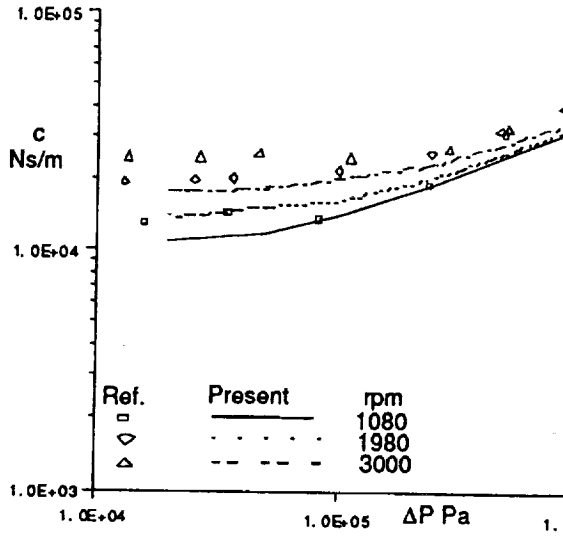
Figure 4. Direct Damping Coefficient. Symbols represent experimental data form Reference 20.
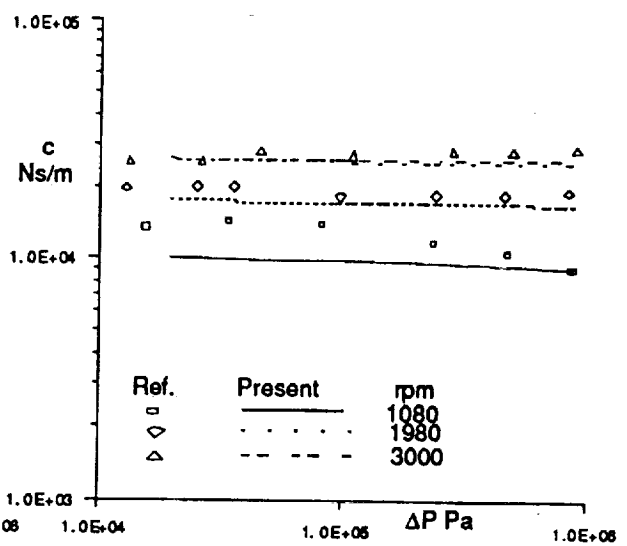
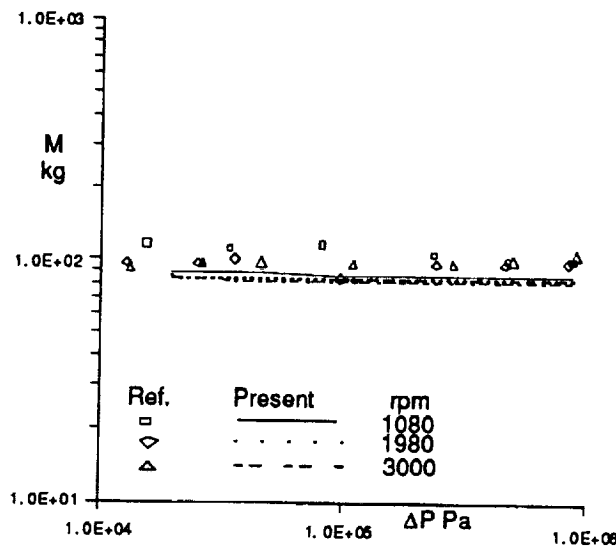Figure 5. Cross-Coupled Damping Coefficient. Symbols represent experimental data from Reference 20.

Figure 6. Direct Mass Coefficient. Symbols represent experimental data from Reference 20.

Entrance loss coefficient: As described above, the entrance loss coefficients is an empirical parameter which depends on several factors such as seal entrance geometry, flow rate *etc.* To analyze the effects of some of these parameters on the loss coefficient, a limited parametric study was performed. A generic annular seal geometry was assumed, as shown in Figure 7. The flow was assumed axisymmetric. Effects of three parameters was considered: 1) radius-to-clearance ratio, 2) entrance region radial gap to clearance ratio, and 3) flow rate. The

143

nominal values were: seal radius = 25 mm, seal length = 25 mm, and the axial length of the inlet region kept at 1.5 times the seal radius.
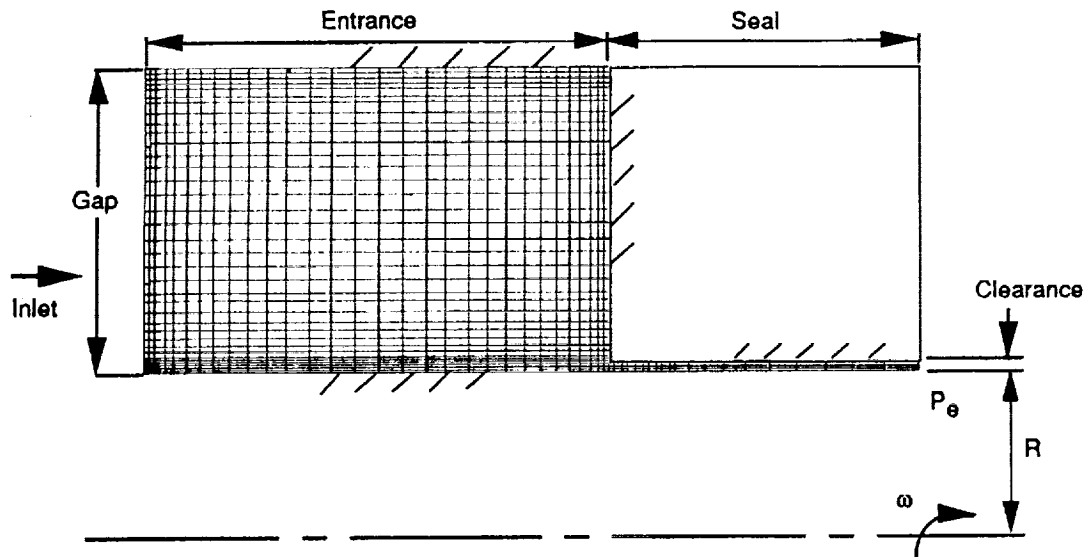

Figure 7.   Flow Domain and One of the Grids Used for the Entrance Loss Coefficient Calculations

Water at 20°C was used as the working fluid, and the rotational speed was kept at 3000 rpm, which corresponds to a rotational Reynolds number of 1.88×10^5. Three radius to clearance ratios were considered: 50, 100, and 150. Two gap-to-clearance ratios were used: 50 and 100. Each of these combinations were run at several different flow rates (axial reynolds numbers). The axial and rotational Reynolds numbers were defined as:

$$Re_{ax} = 2 \cdot (CL)u_{ax,m}/\nu$$

and

$$Re_\theta = \omega r^2/\nu$$

where ν is the kinematic viscosity, $u_{ax,m}$ is the mean axial velocity, CL is the seal clearance, r is the seal radius, and ω is the shaft speed in rad/s. The exit pressure of the seal was kept fixed at 0 Pa, and nominal flow rates were imposed at the inlet. A fully-developed turbulent axial velocity profile was imposed at the seal inlet. Computational grids used had 5 cells in the seal clearance, and 30 or 50 cells in the radial directions in the entrance region depending on the radial gap to clearance ratio. In the axial direction 30 cells were used in the entrance region and 20 cells in the seal region. Appropriate clustering was used to resolve the critical flow regions.

To calculate the loss coefficient, the seal entrance pressure is required. Moreover, this pressure should correspond to the value which will generate the specified flow rate in the seal when only the seal is considered. To estimate this, pressures in the downstream part of the seal length were used. The flow becomes fully developed in this region, and the pressure varies linearly with axial length. This slope was calculated and extrapolated to the seal inlet to provide the inlet static pressure. The loss coefficient was then calculated based on this pressure, and the computed pressure at the entrance of the flow domain.

144

Results of this initial parametric study are given in Tables 2 through 4. A comparison of the results at comparable axial Reynolds numbers reveals several general trends:

1. The radius-to-clearance ratio has the largest effect of the loss coefficient. Increasing the ratio (*i.e.* decreasing the seal clearance) increases the loss coefficient;
2. The loss coefficient also increases with higher entrance region gap to clearance to radius ratio, but the sensitivity is lower;
3. for a fixed geometry, the loss coefficient reduces with higher flow rates.

For the configurations tested, the loss coefficients were between 0.406 to 0.68.

Table 2. Entrance Loss Coefficients, Radius/Clearance = 50

Entrance Gap/Clearance = 50			Entrance Gap/Clearance = 100		
u_{ax} m/s	Re_{ax}	ξ	u_{ax} m/s	Re_{ax}	ξ
10.814	10377	0.471	10.82	10384	0.490
16.232	15484	0.431	10.24	15584	0.488
21.619	20746	0.414	21.66	20785	0.482
26.942	25854	0.406	27.06	25970	0.48

PA-92-24 12

Table 3. Entrance Loss Coefficients, Radius/Clearance = 100

Entrance Gap/Clearance = 50			Entrance Gap/Clearance = 100		
u_{ax} m/s	Re_{ax}	ξ	u_{ax} m/s	Re_{ax}	ξ
10.80	5181	0.562	10.797	5167	0.567
16.56	7945	0.54	16.176	7761	0.558
21.595	10361	0.526	21.55	10339	0.55
26.67	12796	0.51	26.934	12664	0.54
32.27	15484	0.493	32.24	15469	0.537
43.062	20667	0.478	42.533	20408	0.524

PA-92-24 13

Table 4. Entrance Loss Coefficients, Radius/Clearance = 150

Entrance Gap/Clearance = 50			Entrance Gap/Clearance = 100		
u_{ax} m/s	Re_{ax}	ξ	u_{ax} m/s	Re_{ax}	ξ
10.82	3461	0.66	10.75	3438	0.68
16.19	5178	0.65	16.09	5146	0.66
21.49	6874	0.647	21.47	6874	0.65
26.74	8553	0.637	26.81	8553	0.648
32.25	10315	0.628	32.176	10292	0.64
48.33	15461	0.606	47.87	15315	0.63
64.487	20630	0.595	64.165	20630	0.624

PA-92-24 14

145

CONCLUSION

Advanced, CFD based, turbomachinery seals design environment is being developed. It consists of several major components including a scientific CFD flow solver, and a range of industrial seal design codes. A modern graphical user interface will be integrated with a Knowledge-Based System and will provide links to flow solvers as well as to CAD/CAM and scientific visualization tools. The paper presents a novel concept of integrated CFD design code system under the KBS environment with CFD flow solver as major component. The scientific CFD flow solver incorporating state-of-the-art numerical methods has been developed, validated on a broad range of flow problems and demonstrated for several cylindrical seal flow configurations. The code has also been used to calculate the rotordynamic coefficient of number of test seals and ocmpared with experimental and other analytical/numerical results. Results of validations study show very good agreement with available analytical solutions and with well posed experimental measurements. A short numerical study was conducted to estimate the entrance loss coefficient commonly used in seal calculations. The study indicated that an increase in the seal radius-to-clearance ratio increases the loss coefficient. The ratio of entrance gap to seal clearance and the flow rate also change the loss coefficients, but to a smaller extent.

The present efforts include extending the CFD code to labyrinth and damper seals as well as incorporation of additional physical models and as cavitation. The code will be extended to cover a broad range of seal configurations for fluids ranging from incompressible to compressibel gases in two-phase flow.

ACKNOWLEDGEMENT

REFERENCES

1. Tam, L.T., Przekwas, A.J., Muszynska, A., Hendricks, R.C. Braun, M.H., and Mullen, R.L., "Numerical and Analytical Study of Fluid Dynamic Forces in Seals and Bearings," Trans. ASME, *J. or Vibration, Acoustics, Stress, and Reliability in Design,* v. 110, 1988, pp. 315-325.

2. Przekwas, A.J., Habchi, S.D., Yang, H.Q., Avva, R.K., Talpallikar, M.V., and Krishnan, A., "REFLEQS-3D: Computer Program for Turbulent Flows with and Without Chemical Reaction, Vol 1: User's Manual," CFD Research Corporation, GR-89-4-Jan. 1990.

3. Childs, D.W., Nelson, C.E., Nicks, C., Scharrer, J., Elrod, D., and Hale, K., "Theory Versus Experiment for the Rotordynamic Coefficients of Annular Gas Seals: Part 1- Test Facility and Apparatus," *Trans. of ASME, J. of Tribology,* Vol. 108, July 1986, pp.426-432.

4. Athavale, M.M., Przekwas, A.J., and Hendricks, R.C., "A Finite-Volume Numerical Method to Calculate Fluid Forces and Rotordynamic Coefficients in Seals," AIAA-92-3712, AIAA 29th Joint Propulsion Conference, July 1992.

5. Singhal, A.K., "Validation of CFD Codes and Assessment of CFD Simulations," Presented at the Fourth International Symposium on Transport Phenomena and Dynamics of Rotating Machinery (ISROMAC-4), April 5-8, 1992, Honolulu, HI.

6. Schlichting, H., *Boundary Layer Theory,* McGraw-Hill, 1979.

7. Ghia, U., Ghia, K.N., and Shin. C.T., "High-Re Solutions for Incompressible Flow Using the Navier-Stokes Equations and a Multigrid Method," *J. of Comp. Physics,* V.48, 1982, pp.387-411.

8. Armaly, B.F., Durst, F., Pareira, J.C.F., and Schonung, B., "Experimental and Theoretical Investigation of Backward-Facing Step Flow," *J. of Fluid Mech.,* V.127, 1983, pp.473-496.

9. Taylor, A.M.K.P., Whitelaw, J.H., Yianneskis, M., "Measurements of Laminar and Turbulent Flow in a Curved Duct with Thin Inlet Boundary Layers," NASA CR-3367, 1981.

10. Laufer, J., "Investigation of Turbulent Flow in a Two-Dimensional Channel," NACA Report 1053, 1951.

11. Daily, J.W., and Nece, R.E., "Chamber Dimension Effects on Induced Flow and Frictional Resistance of Enclosed Rotating Disks," *Trans. of ASME, J. of Basic Engg.*, V.82, 1960, pp.217-232.

12. Dibelius, G., Rodtke, F., and Ziemann, M, "Experiments on Friction, Velocity, and Pressure Distribution of Rotating Disks," D.E. Metzger and N.H. Afgan Eds., *Heat and Mass Transfer in Rotating Machinery*, 1984, pp.117-136.

13. Morrison, G.L., Johnson, M.C., and Tatterson, G.B., "Three-Dimensional Laser Anemometer measurements in an Annular Seal," *Trans. of ASME, J. of Tribology*, V.113, 1991, pp.421-427.

14. Morrison, G.L., Johnson, M.C., and Tatterson, G.B., "3-D Laser Anemometer measurements in a Labyrinth Seal," *Trans. of ASME, J. of Engg. for Gas Turbines and Power*, V.113, 1991, pp.119-125.

15. Athavale, M.M., Przekwas, A.J., and Hendricks, R.C., "A Numerical Study of the Flow Field in Enclosed Turbine Disk Cavities in Gas Turbine Engines," Presented at the Fourth International Symposium on Transport Phenomena and Dynamics of Rotating Machinery (ISROMAC-4), April 5-8, 1992, Honolulu, HI.

16. Athavale, M.M., Przekwas, A.J., and Hendricks, R.C., "Driven Cavity Simulation of Turbomachinery Blade Flows with Vortex Control," AIAA 31st Aerospace Sciences Meeting and Exhibit, January 1993.

17. Fuller, D.D., *Theory and Practice of Lubrication for Engineers*, pp.364-365, 1984.

18. Cameron, A., *Principles of Lubrication*, Longman's, pp.308-313, 1966.

19. Dietzen, F.J., and Nordmann, R., "Calculating Rotordynamic Coefficients of Seals by Finite-Difference Techniques," *Trans. ASME J. of Tribology*, vol. 109, pp. 388-394, 1987.

20. Kanemori, Y., and Iwatsubu, T., "Experimental Study of Dynamic Fluid Forces and Moments for a Long Annular Seal," *Trans. ASME, J. of Tribology*, v. 114, pp. 773-778, Oct. 1992.

21. Nelson, C.C., "Rotordynamic Coefficients for Compressible Flow in Tapered Annular Seals," *Trans. ASME, J. of Tribology*, v. 107, pp. 318-325, 1985.