# Particle Transport Analysis of Sand Ingestion in Gas Turbine Engines

**Klaus Brun**
e-mail: kbrun@swri.org

**Marybeth Nored**
e-mail: marybeth.nored@swri.org

Southwest Research Institute®,
Mechanical Engineering Division,
6220 Culebra Road,
San Antonio, TX 78238

**Rainer Kurz**
Solar Turbines, Inc.,
9330 Sky Park Court,
San Diego, CA 92123
e-mail: kurz_rainer_x@solarturbines.com

*Significant interest exists in the military and commercial aerospace industry to better predict and improve the durability of gas turbine jet engines that are operating in hostile desert environments, specifically, jet engines that see significant inlet sand or ash ingestion. This paper describes the development of a mixed CFD-empirical software tool that allows a detailed analysis of the kinematic and impact behavior of sand and other particulates in the near-field of turbomachinery blades and impellers. The tool employs a commercially available CFD solver to calculate the machine's transient flow field and then uses the output to determine a set of nondimensional coefficients in a set of empirical functions to predict the statistical probability of particles impacting on rotating or stationary surfaces. Based on this tool's output information, improved inlet air filtering techniques, optimized engine maintenance practices, and component designs can be realized. To determine the empirical coefficient and to validate the method, PIV testing was performed on an airfoil in a wind tunnel; then particle injection into a simple rotating impeller was tested on SwRI's high-speed compressor test rig. Results from these tests allowed optimizing of the model to reflect rotating machinery particle impact behavior more accurately.* [DOI: 10.1115/1.4004187]

*Keywords:* *gas turbine, blade erosion, particle transport, sand ingestion*

## Background

Jet engines are often operated in hostile environments that cause significant amount of ingestion of fine particulate matter, such as sand, gravel, and/or biological matter into the gas turbine principal flow path. Significant interest exists in the military and commercial aerospace industry to better predict and improve the durability of gas turbine jet engines that are operating in these environments, specifically, jet engines that see significant inlet sand ingestion in desert applications. For example, the military had a poor availability of their CH-47 helicopters during the operation Iraqi Freedom due to 75% to 95% premature engine failures related to sand ingestion. Improving this aircraft's availability would have a major impact on the military's operational costs and combat readiness.

There are three different areas in a gas turbine that must be evaluated individually for the effect of particle matter, such as sand in the flow path: The compressor, combustor, and turbine sections.

In the compressor, particle ingestion damage is mostly related to fouling (buildup of solid matter on the blades) and abrasive metal removal from the blades leading and trailing edges (rounding of the blades); i.e., blades are effectively sandblasted which leads to a deformation of the airfoil shape. The net consequence of the above is a general loss of compressor pressure ratio and overall gas turbine efficiency.

In the combustor, the local flame temperature can be as high as 3000°F. At these temperatures, sand will become a liquid silicone oxide (often called slag) that will deposit on the combustor injector and liner walls. These slag deposits will often lead to plugging of cooling holes on the liner and thus inadequate local flame cooling and wall hot spots, causing premature combustor liner metal failure.

In the hot turbine section, the sand particles generally create two separate problems: Sand particles can easily plug up film cooling holes/slots, and thus create film-cooling deficiencies. Also, the hot sand and silicone exhausting the combustor and entering the nozzle section can cause silicone deposits. The silicone build-up phenomenon is often encountered during sandblasting operation: Due to the high impact pressure and temperature of the sand particles, the silicone melts out of the sand and forms a solid film layer on the surface of nozzles and buckets. This film decreases the heat transfer (heat carry) ability of the metal and results in local overheating of the blades.

A final issue to consider is the blade cooling air that is bled for the compressor discharge and is then used as an internal cooling flow in the gas turbine hot section blades. Any solid matter, such as sand particles, that is carried into this cooling flow will deposit in the very fine blade cooling passages and plug them. Consequently, insufficient cooling flow will be available on the hot section blades, and premature high temperature blade metal failure will be the result.

The phenomenon of particulate ingestion damage in a gas turbine must be studied as a problem of particle-flow lag; i.e., the analysis of solid particle kinematic behavior in a high velocity rotating fluid. A solid particle of a given density and size that is carried by the air flow in a gas turbine, ideally follows the flow streamline and should never impact with any stators-rotor blades or walls. However, if during a sharp turn of the air flow (rapid direction change of the streamlines), the centrifugal and Coriolis forces on the particle exceed the aerodynamic drag and inertial forces, then the particle will deviate from the streamline and may impact on a blade, vane, wall, bucket, etc. and cause surface damage (or plug up a cooling hole). For a given gas turbine geometry, the probable particle impact regions, forces, and likely surface damage can be evaluated using particle lag theory and computational fluid dynamics.

This paper describes a set of application tools that is based on a mix of computational fluid dynamics (CFD) and a semiempirical particle transport model to predict gas turbine sand corrosion and fouling. The model has been verified using actual high-speed turbomachinery sand testing and laser particle image velocimetry on airfoil sections. The tools has already employed this particle lag analysis method to successfully predict the characteristic behavior and impact forces of solids in an axial air compressor for a refinery compression application.

Contributed by the International Gas Turbine Institute (IGTI) of ASME for publication in the Journal of Engineering for Gas Turbines and Power. Manuscript received April 27, 2011; final manuscript received April 29, 2011; published online October 28, 2011. Editor: Dilip R. Ballal.

The herein described turbomachinery particle flow analysis tool can be applied to any ground-based or aircraft propulsion jet engine to predict:

- the size and density of particles primarily responsible for damage and degradation in gas turbine. With this knowledge, the filter size and design can be optimized.
- the local impact force of sand particles on the turbomachine's internal blades, vanes, nozzles, walls, etc. Using these results, the coating life, metal abrasion, and wear can be predicted to improve scheduling of the turbomachine's maintenance intervals.
- the highest probability areas within gas turbine for fouling, deposit formation, and cooling flow interruptions; i.e., probable failure areas. Based on these results, the maintenance team can focus on high-risk areas for closer inspection.

The above analysis results allow one to develop a strategy to "harden" a gas turbine locally against the damaging effects of sand ingestion and/or to develop better maintenance techniques. For example, areas that have been identified to have the highest amount of particulate impacts and forces could be locally strengthened or a surface coating could be applied for special protection. Also, analysis results may show that only certain size and weight particles ingested into the gas turbine are responsible for the majority of damage; improved filtering techniques would allow to only target these particles rather than the currently employed "insect screen" total filtering. Finally, by identifying the areas that suffer most particulate damage, inspection and replacement strategies can be implemented to reduce cost of downtime and repair. By proper implementation of the analysis results into a turbomachinery filtration, inspection, maintenance, and hardening optimization strategy, significant cost savings and increased equipment availability can be achieved.

Thus, the primary aim of the herein described project was to develop a software tool that would allow detailed analysis of the kinematic behavior of sand and other particulates inside a jet engine gas turbine. Based on this tool's output information, improved inlet air filtering techniques, engine maintenance, and component designs were realized. These improvements were based on real data rather than the current simple approach of crude inlet filtering, component overdesign, and/or reactive internal part fixes to existing equipment.

In parallel with the analytical work of this project, particle injection into a simple rotating impeller was tested on SwRI's high-speed rotating machinery test rig to verify the method and optimize the model's implementation to reflect rotating machinery particle impact behavior more accurately. Thus, once the particle transport model had been fine-tuned, a generally applicable method was available to accurately predict the motion and impacting of sand and other impurities on the internal structures of an aircraft's gas turbine jet engine.

## Introduction

There are three different areas in a gas turbine jet engine that must be evaluated individually for the effect of particle matter in the flow path: the compressor, combustor, and turbine sections [1].

In the axial/radial compressor, particle ingestion damage is mostly related to fouling (buildup of solid matter on the blades) and abrasive metal removal from the blades' leading and trailing edges (rounding of the blades); i.e., blades are effectively sandblasted which leads to a shortening and deformation of the airfoil shape. The net result of the above is a general loss of compressor pressure ratio, output power, and overall gas turbine efficiency.

In the combustor, the local flame temperature can be as high as 3500°F. At these temperatures, sand forms a liquid silicone oxide (often called slag) that can deposit on the combustor injector and liner walls. The slag deposits often lead to plugging of cooling holes on the liner and; thus, inadequate local flame cooling and wall hot spots; premature combustor liner metal failure is the result.

In the hot turbine section, the sand particles generally create two separate problems: sand particles can easily plug up film cooling holes/slots and; thus, create film cooling deficiencies, which contribute to hot-section heat corrosion. Figure 1 shows the surface temperature on a G technology gas turbine first stage bucket increases with decreasing through-flow in the cooling holes. Also, the hot sand and silicone exhausting the combustor and entering the nozzle section can cause silicone deposits. The silicone build-up phenomenon is often encountered during the sandblasting operation. Due to the high impact pressure and temperature of the sand particles, the silicone melts out of the sand and forms a solid film layer on the surface of nozzles and buckets. This film decreases the heat transfer (heat carrying) ability of the metal and results in local overheating of the blades.

A final issue to consider is the blade cooling air that is bled from the compressor discharge and is used as an internal cooling flow in the gas turbine hot section blades. Any solid matter, such as sand particles that are carried into this cooling flow, will likely deposit inside the very fine blade cooling passages and plug them. Consequently, insufficient cooling flow will be available on the hot-section blades, and premature high temperature blade metal failure (hot corrosion) will be the result.

## Problem Statement

The phenomenon of particulate ingestion damage in a gas turbine can be studied as a problem of particle-flow transport (i.e., the analysis of solid particle kinematic behavior in a highly nonuniform and high velocity fluid flow). A solid particle of a given density and size that is carried by the airflow in a gas turbine ideally follows the flow streamline and should never impact with any stators/rotor blades or walls. However, if during a sharp turn of the airflow (rapid direction change of the streamlines) or during a rapid transient flow fluctuation, the centrifugal, inertial, and Coriolis forces on the particle exceed the aerodynamic drag forces, then the particle will deviate from the streamline and may impact on a blade, vane, wall, bucket, etc. and cause surface damage (or plug up a cooling hole). For a given gas turbine geometry, the probable particle impact regions, forces, and likely surface damage can be evaluated using particle lag transport theory and CFD.

As the fluid flow in a gas turbine is highly nonuniform, particle lag due to nonlinear fluid behavior becomes a major contributor to component degradation. Unfortunately, CFD and particle transport calculations are computationally intensive and sometimes numerically unstable. Thus, a more efficient method must be utilized to determine probable particle impact locations and velocities. This method should still rely on CFD analysis of the flow field but can utilize a simpler physical model to predict particle impact behavior.

Within the subject project, a software tool was developed to accurately predict the behavior of solid particles, such as sand,

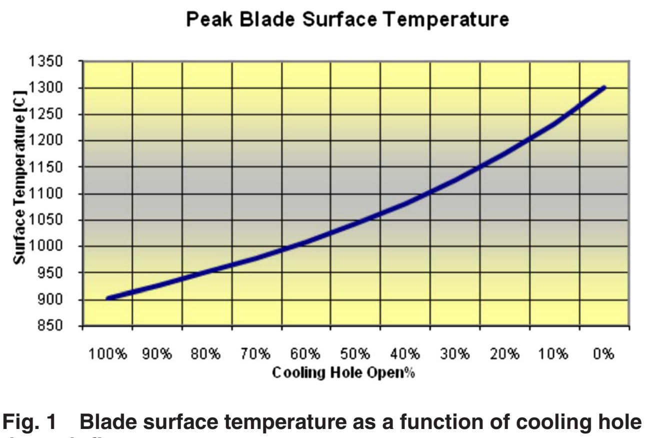


**Fig. 1 Blade surface temperature as a function of cooling hole through-flow**

dirt, and dust in a gas turbine jet engine. The program's analysis method was based on the derivation of the particle transport kinematic mechanics in a high velocity fluid, the implementation of a numerical solution of these equations based on a given velocity vector and pressure/density scalar field, and the interfacing with commercially available CFD codes. It was not within the project's stated goals to develop a new CFD code, but rather to employ a proven commercial turbomachinery CFD code as a tool for predicting particle kinematics within the highly complex flow field of an aircraft gas turbine.

**Project Tasks.** The following two tasks were implemented in parallel to achieving the project's objectives.

*Task 1 – Particle Transport Kinematics Analysis*

- derivation of transient particle transport equations and numerical model
- implementation of the numerical model and CFD interface development
- development of semiempirical model to determine particle behavior
- benchmarking of analysis method using simplified geometry
- calibration of model based on experimental test data
- application of code to first stage of centrifugal compressor (demonstration run)

*Task 2 – Experimental Verification*

- testing of particulate flow behavior in nonrotating flow (PIV in wind tunnel)
- testing of particulate flow behavior in rotating flow (high-speed compressor)
- validation of CFD and semiempirical model results using experimental results

Test work in Task 2 required the utilization of flow visualization (particle image velocimetry) and application of abrasive paints to determine impact behavior of particles. The analysis work in Task 1 was mostly completed utilizing a commercial CFD package, CFDesign by Blue Ridge Numerics.

**Literature Review.** Clearly, a number of researchers have investigated sand ingestion into gas turbines.

Solid particles may be transported into the gas turbine engine through the axial compressor as airborne ash particles, sand, other solid particulates, dust, and ice. The gas turbine combustor fuel system can also produce solid and molten particles when burning heavy oils or synthetic fuels. Particle transport paths and the effects of particle impact or deposition vary, based on the type and quality of material (synthetics versus ash), size distribution, and the particle size, as well as the gas flow path, operating conditions, and the blade material, and geometry. Previous research has centered on the modeling, design mitigation, and prediction of namely four degradation and performance-related effects: Compressor and turbine blade erosion, deposition in the combustor and hot sections leading to corrosion, impingement/deposition on GT blades leading to blockage of cooling passages, and reduction of the efficiency and surge margin due to blade tip clearance reduction.

In the area of blade erosion and advanced surface coating development, wind tunnel tests of blade material and coating erosion are well documented by Hamed et al. [2] —who provides an account of testing since 1960 over a range of temperatures, from ambient to 704 °C with aluminum oxide, fly ash, quartz, sand, and chromite. Tabakoff et al. [3] studied blade and coating erosion in an erosion wind tunnel to determine erosion rates and thermal barrier coating life reduction due to particle impact. Testing was performed over a range of typical hot section temperatures from 1600 to 2000 °F to show the escalation of erosion rate due to increased temperature. Sand particle ingestion in particular was shown to blunt the leading edge of rotor blades, reduce blade chords, and increase surface roughness [4]. This experimental work indicated that larger particles will cause increased erosion rates, but the particle size dependency is reduced when the particle impact velocity decreases. Richardson et al. [5] showed that the erosion effects on high pressure compressor blades tend to center on the outer 50% of the span, where notable reduction in blade chord length and thickness are significant. His work also showed the effect on airfoil surface roughness and a limit to the eroded surface roughness, whereby additional continued particle impacts and mass removal after 2000 cycles proved to have no effect on the blade roughness.

Recent erosion test research has focused on the significant interaction and importance of the particle rebound condition, since the particle trajectories within the gas turbine engine are heavily affected by the rebound velocity and magnitude. Tabakoff and Sugiyama's [6] laser Doppler velocimetry measures (on fly-ash type particles) found that impact angle is the primary factor affecting the restitution ratio. Departures from the mean restitution ratio were notable for the nonrounded particles. Sand particle ingestion was studied by Cowan et al. [7] through short pin fan array tests. The numerical simulations showed a dependency on the Stokes number of the particle. A smaller Stokes number particle size resulted in the highest deposition rate and least number of surface impacts (possibly due to a faster time to deposit and less resulting rebound impacts). The softening temperature was also shown to affect the deposition rate, whereby a lower softening temperature increased the number of deposits on the blades.

Regarding combustion/hot section deposition and related corrosion effects, it is important to consider the delivery of solid particles through the fuel system which may enter in solid and liquid form. Vaporization will likely occur in the hot section but later lead to deposition after condensation on cooler turbine surfaces in later stages of the turbine. The larger particles tend to collect due to inertial impact while smaller particles may be delivered by gas molecules at high temperatures. Extremely small particles may be transported by diffusion and also tend to collect on cooler surfaces [2].

Dunn et al. [8] investigated gas turbine operational effects when subjected to volcanic ash particles. This work showed images of the engine inlet characteristically known as "St. Elmo's glow," which appears due to the strikingly bright ring of light at the first rotor tip caused by dust particle ingestion. To cause material deposition in the hot section through thermally agitated gas particle transport of solids, Dunn reported the temperatures must be on the order of 2000 °F (1094 °C). Newer engines operating at these higher temperatures will experience both material deposition and erosion effects on the compressor and turbine blades.

In the area of deposition on the blades and blockage of cooling paths, considerable research has been done in predicting cooling hole blockage and design methods to counter this occurrence. Haase and Bons [9] studied cooling hole deposition in the context of synthetic fuel usage. Flow visualization testing was performed to show that large deposits in the front rows of film cooling holes help to promote a cavity and coolant airflow, which is more stable than smaller depositions throughout the cooling holes. Walsh et al. [10] studied sand ingestion on blockage of film cooling holes using a leading edge coupon over a range of sand particle size, amounts, and metal/coolant temperatures. Increased coolant velocity and pressure ratio were found to aid in passing particles through the coolant passage walls, causing less adherence to the surface. Metal temperature; however, was shown by Walsh to be the most significant parameter affecting particle adherence. Temperatures of above 1000 °C were found to promote melting of the sand particles and blockage of cooling holes, which was a limit close to the combustor/hole section deposition temperature limit observed by Dunn.

Design investigations to aid in reducing cooling hole blockage are numerous. Land et al. [11] investigated double-walled cooling geometries with impingement and film-cooling to combat ingested sand. The work examined if impingement cooling could be used to break up particles in cooling passages and minimize film-cooling hole blockage and found that a staggered arrangement of film-cooling and impingement air flow holes could aid in breaking down larger particles for transmission through the film-cooling holes. Musgrove et al. [12] also designed and studied a system of filtering sand or other small particles using a multiple louver array design. Optimization of the design resulted in the ability of the collector to develop an inlet vortex to create a favorable circulation pattern for particle collection.

The leading work in characterizing performance changes due to tip clearance reduction was performed by Dunn et al. [8]. They measured tip clearance reduction after erosion experiments using air entrained dust particles. The clearance reduction was reported to be greater than three times the initial tip clearance, which caused early surge in the machine, effectively eliminating the surge margin. Schmucker and Schaffer [13] found that erosion lead to 1% loss in tip clearance corresponded to a 7.5% reduction in surge margin and a 2% loss in efficiency. Separately, A. Ghenaiet et al. [14] focused a complete set of tests on sand ingestion in particular, using Lagrangian tracking methods for particles up to 1000 μm. Aerodynamic performance impacts due to blade profile loss and end wall loss were estimated to determine effects on adiabatic efficiency, pressure rise coefficient, and stall margin. The blade chord reduction was notably more significant in affecting performance compared to tip clearance loss. Relating turbomachine life to 10% efficiency drop, the authors showed that the time of loss decreased exponentially with sand concentration level, falling from 350 h to less than 100 h for sand particle concentrations of more than 100 mg/m$^3$.

Apart from a study of how these degradation mechanisms occur or means of protecting against them, it is equally important to understand the behavior of solid particles with a reliable prediction of the particle transport path. Reliable particle transport prediction allows a designer/operator to understand if deposition or erosion is likely to occur based on particle residence time and particle locations. Hussein and Tabakoff [15] provided the initial particle trajectory simulation effort in 1974, using experimentally obtained restitution ratios to simulate various particle trajectories. The work showed that the blade surface impacts tend to increase with increasing particle diameter and increased velocities at the axial compressor stage inlet. Since this early research, three-dimensional flow effects and viscous forces have been incorporated into many Eulerian–Lagrangian models of particles in the flow stream. A more complete summary of leading work in radial inflow turbines and axial compressors is provided by Hamed et al. [2].

**Equations of Motion.** The set of equations that govern fluid flow are known as the Navier-Stokes equations (Brun and Kurz [16]). Like all Newtonian conservation-based equations, they consist of a mass continuity, force balance, and energy balance equations. They are a set of higher order, coupled, nonlinear, and nonhomogenous partial differential equations for which a closed form analytical solution can only be obtained for a limited set of very basic problems.

Numerical solutions to the Navier-Stokes equations can be obtained with a reasonable accuracy using modern, commercially available CFD codes. However, as these codes are intended to predict the fluid flow field only, they are generally not capable of solving transient particle transport kinematics problems. Some CFD codes offer simple single particle transport modules and/or "tracer" visualization techniques, but these are essentially just visualization of the flow streamlines or streak lines and do not predict how a multitude of particles would behave as they are carried by a highly unsteady flow. Thus, to analyze particle movements within an unsteady fluid flow, it is first necessary to determine the transient flow field (using CFD codes) and then apply time-dependent particle transport equations to model individual particle kinematics within the three-dimensional flow field.

The equations that govern relative particle movements within a fluid flow are a summation of forces on the particle moving in the local frame of reference. By applying the inertial drag (from the CFD model), centrifugal, buoyancy, and Coriolis forces to the particle in the moving frame at small time steps, the transport of the particle relative to the flow can be simulated. Namely,

$$\left[\begin{matrix} F_p = m_p a = F_{\text{Drag}} + F_{\text{Centrifugh}} \\ +F_{\text{Coriolis}} + F_{\text{Buoyancy}} \end{matrix}\right]_{\text{Relative}} \tag{1}$$

For small incremental steps in *an absolute frame* of reference, the centrifugal and Coriolis forces are effectively inertial forces, and the subject equation becomes:

$$\left[m_p \frac{d\nu}{dt} = f_{\text{Drag}} + f_{\text{Buoyancy}}\right]_{\text{Absolute}} \tag{2}$$

A transient numerical solution for this equation requires computation of both the drag and buoyancy terms as a function of the fluid flow field (determined from the CFD code) and the motion of the particle relative to the absolute frame of reference. As the analysis has to be performed in a complex three-dimensional flow field with very small transient time steps to capture a highly unsteady flow field, the solution to this equation is numerically intensive and difficult. Furthermore, there is very little experimental data available in the public domain that would allow one to compare analysis results with proven test data. Thus, to verify and calibrate a particle transport model, it is critically necessary to perform a number of laboratory tests on an actual high-speed rotating turbomachine, such as a centrifugal or axial compressor.

**Analysis Approach.** Within the subject project, a CFD and semiempirical approach was utilized to determine particle behavior in turbomachines. This approach is based on using CFD to determine the flow field without the influence of particles and then to utilize the 3D flow field results to determine surface impact velocities using a semiphysical model. Namely, once a flow field for a rotating machine was determined using a commercial CFD solver, the following approach was taken.

The motion of a particle can be determined based on a simple model that balances inertial, centrifugal, Coriolis, and drag forces on the particle. Namely, a model based on:

$$m\frac{d\vec{V}}{dt} = F_{\text{Centrifugh}} + F_{\text{Coriolis}} = \vec{F}_{\text{Drag}}$$

with some determinable velocity vector impact coefficients, appears to adequately predict the motion of sand particles around a stationary airfoil. Nondimensional (Buckingham-$\pi$) analysis shows that the velocity vector impact (VVI) of a particle near an airfoil must follow the functional form:

$$\gamma_I = \frac{R^2\rho_p\dot{V}_p}{\mu_f\Delta V} \tag{4}$$

$$\gamma_C = \frac{R^2\rho_p V_{\hat{t}}^2}{\mu_f r\Delta V} \tag{5}$$

$$\gamma_O = \frac{R^2\rho_p V_{\hat{r}} V_{\hat{t}}}{\mu_f r\Delta V} \tag{6}$$

$$\Gamma_I = \frac{R\rho_p\dot{V}_p}{\rho_f\Delta V^2} \tag{7}$$

$$\Gamma_C = \frac{R\rho_p V_{\hat{t}}^2}{\rho_f r\Delta V^2} \tag{8}$$

$$\Gamma_O = \frac{R\rho_p V_{\hat{r}} V_{\hat{t}}}{\rho_f r\Delta V^2} \tag{9}$$

$$V = A\gamma_I + B\gamma_C + C\gamma_O + D\Gamma_I + E\Gamma_C + F\Gamma_O \tag{10}$$

where $A$, $B$, $C$, $D$, $E$, and $F$ are VVI coefficients that have to be determined from comparison of the experimental results of sand impacting location on the airfoil surfaces with the CFD results near the airfoil surfaces. These nondimensional coefficients correspond to inertial, Coriolis, and centrifugal forces (denoted by $I$, $C$, and $O$, respectively) on a particle divided by the drag forces on the particle. Coefficients that are near unit or above signify that the nondrag forces dominate on the particles and that the particles will likely not follow the streamlines. Thus, by calculating the coefficients locally near the blade surfaces from the CFD flow field results, one can determine probable regions of impacting particles. Experimental results compared to CFD analysis showed that if individual coefficients actually exceeded 0.5, or if the sum of all coefficients exceeded 0.8, then significant particle impacts are seen. The impact density of the particles appears to behave linearly with the VVI coefficients.

As the fluid behaves differently depending on the flow being either laminar ($\mathrm{Re} < 500$) or turbulent ($\mathrm{Re} > 500$), separate nondimensional parameters were derived for each of the flow regimes, where $\gamma$ and $\Gamma$ correspond to laminar and turbulent, respectively. However, the analysis results showed that most relevant particle forces near the blades were in the turbulent flow regime and that the laminar nondimensional coefficients could be neglected (i.e., only the $\Gamma$ parameters were found to be significant). Thus, the critical VVI coefficients for the analysis are only $D$, $E$, and $F$.

One should note that the impact velocity is a vector and that the nondimensional coefficients must consequently be individually evaluated for each direction ($x$, $y$, $z$ in the stationary frame; the normal, bi-normal, and tangential directions relative to the moving blade in the rotating frame). Also, because of the linearized equation form assumption, the velocity vector V and VVIs will be valid only for the immediate vicinity around an airfoil (which is the primary region of interest of this project).

To determine the impact velocity, it was assumed that the particle velocity will correspond to the average flow velocity just streamwise upstream from the particle location. Although this assumption is likely to be somewhat conservative (i.e., impact velocities will be overpredicted) to meet the objective of this project, this approach is generally preferred.

**Analysis Implementation.** The following individual tasks must thus be completed to implement the particle impact analysis method described above.

For a given stationary or rotating geometry, the flow field is calculated using a CFD code. The CFD code provides the complete 3D velocity field as an output.

Based on the velocity field, the VVI coefficients $D$, $E$, and $F$ are calculated for each flow direction near the surfaces of the blades of interest.

Results of these calculations are then plotted as a surface contour plot to identify the regions on the blades of highest probability of particle impacts. Figure 2 shows a typical (not to scale) map of VVI probability regions on the leading edge of an airfoil.

For the highest probability regions ($\mathrm{VVI} > 0.5$), the impact velocities are then calculated from CFD results using local velocity vectors streamwise upstream of the impact locations.

## Particle Analysis in Nonrotating Flow

Prior to performing experimental and CFD work on particulate behavior in the rotating frame, it was determined to develop a set of experiments to verify the approach in the stationary frame. The geometry for analysis selected was NACA 009 airfoil, which is a blade shape commonly utilized in the gas turbine industry for stationary flow elements. Sand behavior around the blade was analyzed using both CFD and experimental flow visualization.

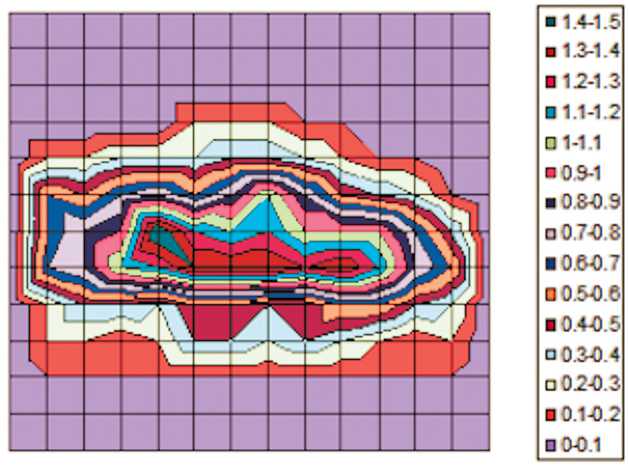


**Fig. 2 VVI total sum indicating probability of particle impacts on leading edge of airfoil**

**Experimental Setup.** To analyze how various sizes and specific density sands behaved around a NACA 009 airfoil, an airfoil was mounted in a small wind tunnel, and a simple particle image velocimetry (PIV) system was devised. The PIV system consisted of a 300 mW Argon-Ion laser, a mirror mounted on a small high-speed DC motor, and a 35 mm camera. In this simple arrangement, the laser beam is aimed at the rotating mirror (rotating at 11,000 rpm) to generate a light sheet in the wind tunnel. Particles, such as sand grains, flowing through the wind tunnel, will scatter light as they cross the light sheet. A slow exposure photograph of this light scatter allows one to trace the movement of single particles at fixed intervals corresponding to the rotational speed of the laser beam in the light sheet. Thus, velocity vectors for a 2D section of the entire flow field can quickly be determined.

For the analysis of sand flow over a NACA 009 airfoil, three different sand grain sizes, three angles of attack, and two airflow velocities were tested. Velocity vectors were then compared to a CFD analysis of the same airflow.

**Results of Particle Behavior in Stationary Frame.** Sand kinematic behavior was studied around NACA 009 airfoils for various flow incidence angles. This study was performed to evaluate the relative influence of streamline curvature and particle size on the motion of particles in the airflow, as well as to test some basic particle kinematic theories. For this task, a small wind tunnel was constructed and a 500 mW Argon-Ion laser was employed to visualize particles carried by the flow using a basic particle image velocimetry (PIV) technique. Figure 3 shows a typical example of flow visualization results. Three sand grain sizes were tested:

- coarse 0.5 mm average particle size
- medium 0.2 mm average particle size
- fine 0.05 mm average particle size

All three sands had approximately equal specific gravities of approximately 2.1. Experimental results showed that smaller

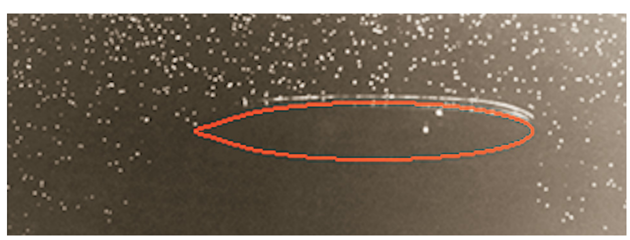

**Fig. 3 Particle image velocimetry on a NACA 009 airfoil**

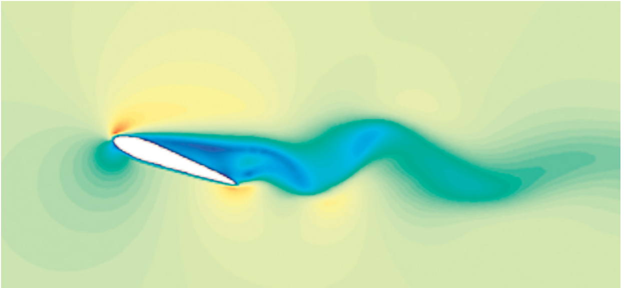

Fig. 4 Simple 2-D CFD model of flow around airfoil

particles tend to follow the flow streamlines more closely than larger particles.

**CFD Comparison.** In parallel to the experimental work, a simple 2D CFD model was employed to determine the transient streamlines (velocity and local curvature) of the flow around the airfoil (see Fig. 4).

Based on the CFD results, the above-described VVI coefficients were determined. Typical results are shown in Table I for the 0.2 mm particle sand.

Comparison of the results in Table I with the PIV measurements showed that the highest impact blade impact velocities and probability of impact were at the leading edge. These results were consistent with PIV observations and experimental results in the public domain. One should note that the F-VVI term was zero for all cases. This term corresponds to Coriolis forces, which are not acting in a nonrotating flow field.

## Particle Analysis in Rotating Flow

**Motion in the Rotating Turbomachinery Reference Frame.** When expanding the previous model to the rotating frame, particular care has to be given to the inertial terms, as the particles in the fluid will also be affected by global centrifugal and Coriolis forces. To develop this model for the rotating frame, experimental sand injection work has been performed in a high-speed compressor at SwRI. CFD results can then be correlated to the experimental results with the aim of determining the VVI coefficients as a generalized model for sand transport near airfoils in a high-speed turbomachine.

**CFD Modeling Approach.** CFD work was performed to provide a baseline for experimental results. A high-speed centrifugal compressor onsite at SwRI was modeled using a 3D solid modeling package. Only one stage of the two-stage compressor was modeled, as this single impeller is the focus of experimental work (see Fig. 5).

This geometric representation was then imported into a commercial CFD code. Using expected experimental parameters, a series of CFD simulations were conducted to determine flow field characteristics at the leading edges of the impeller blades. The velocity components were taken from the CFD analysis to compute the nondimensional flow parameters previously defined in this project summary.

A series of models have been analyzed using various permutations of mesh density, boundary conditions, and turbulent flow

Table 1 CFD analysis of open loop compressor wheel

| Location (mm) | D-VVI | E-VVI | F-VVI | Sum VVI |
|---|---|---|---|---|
| 0.0 (Center line) | 0.8 | 0.6 | 0.0 | 1.4 |
| 0.1 | 0.4 | 0.7 | 0.0 | 1.1 |
| 0.2 | 0.1 | 0.5 | 0.0 | 0.6 |

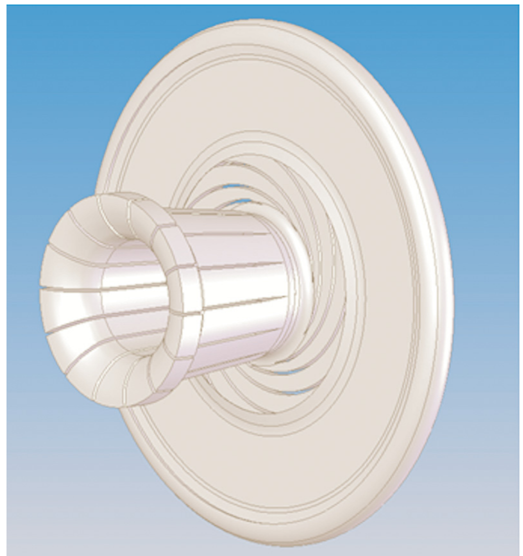

Fig. 5 Single-stage impeller CFD model

models. The data presented here is from a simulation that is largely representative of all the runs performed. The captures below (Fig. 6) depict CFD output of flow velocity inside the impeller gas volume. It should be noted that the images represent geometry of the fluid volumes and not the solid components of the compressor. The software package being used for this analysis allows for flow characteristics to be determined at specific locations. One area of particular interest being studied is the leading edge of the impeller blades. The vector plot shows the direction of flow in this area.

The nondimensional parameters previously discussed in this report were calculated based on the CFD results. For properties of sand, desert sand from Qatar was ascertained and analyzed. Average density and radius values were used for all calculations. This CFD analysis was repeated for a number of flow and head conditions of the compressor as shown in Table II.

Some parametric studies of the CFD analysis were also performed to develop a qualitative understanding of the relative physical influences of each of the VVI coefficients. For example, computations demonstrated that the simple inertial terms were dominated by centrifugal and Coriolis terms. While the centrifugal parameters did not vary significantly at small distances from the leading edge of the impeller blades, the Coriolis parameters showed significant dependence of position. Figure 7 is a representative plot showing the Coriolis terms as a function of radial distance from leading edge of a rotating compressor blade.

**Testing of Particles in Compressor.** Experiments were performed using SwRI's high-speed centrifugal compressor test stand. A variable speed drive electric motor was used to drive the centrifugal compressor at speeds up to 10,000 rpm, and sand was injected into suction of the compressor. The experiments were used to determine where sand impacted the impeller blades. Only one stage of the compressor was used for this analysis. The surfaces of the impeller exposed to airflow were painted with an abradable paint prior to each test run. The impeller was inspected after each test run to determine where sand impacted the blades.

Measured quantities of sand and other particles were injected into the suction flange of the compressor upstream of the impeller and mixed using a preswirler. As the compressor was operated in an open-loop arrangement, the sand particulates passed only once through the compressor before being discarded. This allowed also for easy control and consistency of the quantity of sand passed through the compressor for each test. After all particles had passed through the impeller, the hardware was disassembled and visually

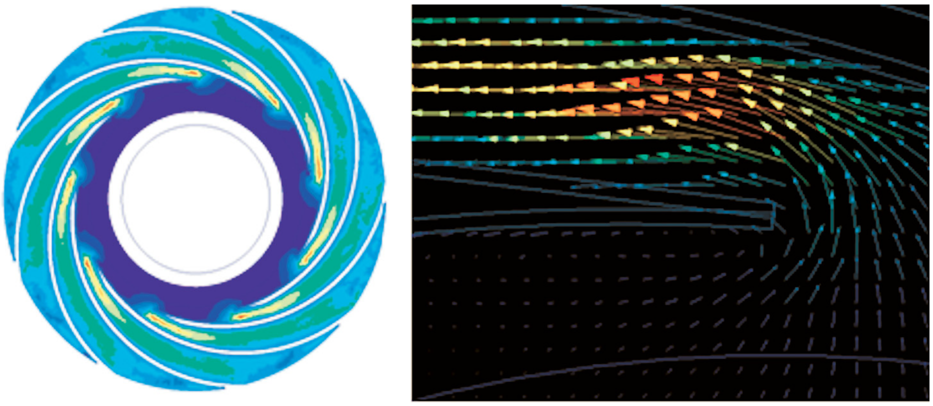

Fig. 6 CFD results — velocity magnitude plot and vector plot near leading edge

Table 2 0.2 mm sand on NACA 009 airfoil at aero incidence flow

| Case | Speed (rpm) | Flow (acfm) | Pressure Ratio |
|---|---|---|---|
| 1 | 5000 | 8000 | 1.05 |
| 2 | 5000 | 7000 | 1.15 |
| 3 | 10,000 | 16,000 | 1.05 |
| 4 | 10,000 | 14,000 | 1.15 |

inspected for paint abrasion. From the visual inspection, the main particle impact locations were measured, and the impact density was inferred. Figure 8 shows leading edge abrasion of paint at on the compressor impeller inlet.

Three separate test runs were performed. The first used sand ($\rho = 1.694$ g/mL) with a compressor speed of 10,000 rpm. The second test used diatomaceous earth ($\rho = 0.325$ g/mL), and the third test again used sand. These last two experiments were run with the compressor operating at 5000 rpm.

**Experimental Results.** Analysis of the experimental results allowed the comparison of predicted VVI coefficients from the CFD results with actual particle impact distribution of inside a high-speed turbomachine. For each test, all abrasion locations inside the impeller were documented by their location, size, and relative paint removal (visual only). These locations and sizes were then directly compared to a map of the sum of VVI coefficients and individual VVI coefficients for each directional component determined from the CFD results. The map and the photograph could then be overlapped to determine whether the CFD code reasonably well predicted the highest probability impact locations of sand for a given grain size and operating conditions. Figure 9 shows a typical photograph and (manually) overlapped numbers of local VVI coefficients on the suction side of the internal flow passages of the compressor. Good agreement between VVI coefficients and local abrasion is seen in this figure.

A similar analysis was performed for all test results for the impeller sand ingestion testing. For each location that was identified in the test to show significant sand abrasion, the CFD analysis results were overlapped and compared.

**Results of Rotating Frame Analysis.** Generally, good agreement was found between the predicted highest probability impact locations (from the VVI coefficients) and the actually observed test results for all operating conditions of the compressor. It is difficult to define an absolute uncertainty of this type of measurement, as the inspections are performed visually, and the comparison is somewhat qualitative in nature. However, results showed that whenever any of the individual VVI coefficients normal to the surface exceeded 0.5, or the sum of the VVI coefficients normal to the surface (in the rotating frame) exceeded 0.8, then a measurable area of particle impacts was identified in *all* test cases. Thus, if the model predicts an area where sand will impact, then there is a very high probability of actual sand abrasion.

Nonetheless, for a number of cases it was determined that the VVI coefficients underpredicted particle impact locations (i.e., particle impact locations that were experimentally observed were not properly predicted). This corresponded to approximately 20% of all particle impact locations (i.e., the model has a miss rate of

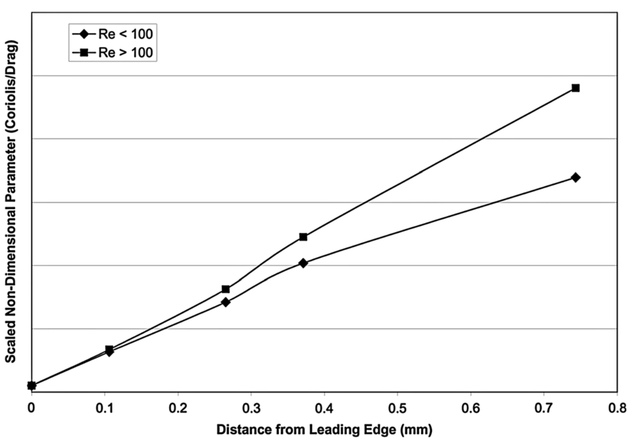


Fig. 7 Coriolis VVI terms as function of radial distance from leading edge

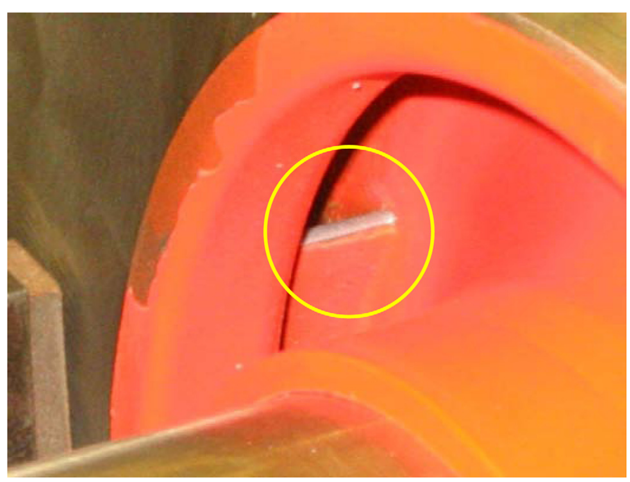

Fig. 8 Impeller blade leading edge after sand ingestion experiment

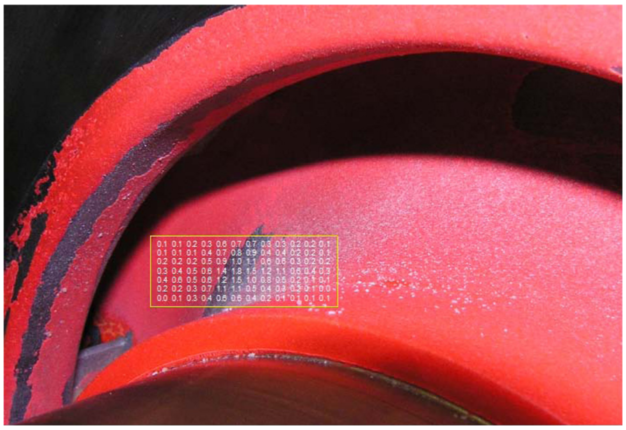

**Fig. 9 Sand abrasion in centrifugal impeller with local VVI coefficients**

approximately 20%). This incomplete prediction capability is likely due to the uneven flow particle stream of the sand that was entrained into the centrifugal impeller geometry. In real life gas turbine applications, it is likely that the model would miss a smaller number of impact locations than the laboratory tested and have an improved miss rate of approximately 20%.

The rotating frame analysis did not allow for a direct determination of impact velocities from the testing results (as did the PIV), but the impact density from the test results provided some indication of the strengths of relative impacts of the particles. As previously described, local impact velocities were calculated from the average streamwise upstream flow velocities for all high probability particle impact areas. Again, (at least qualitatively) good agreement between the test results and the CFD predictions were found.

## Summary and Conclusions

A software tool and semiphysical (or semiempirical) model was developed to predict the behavior of solid particles, such as sand, dirt, and dust in a gas turbine jet engine. The program's analysis method was based on the derivation of the particle transport kinematic equations in a rotating high velocity fluid, the implementation of a numerical solution of these equations based on a given velocity vector field, and the interfacing with commercially available CFD codes. This model's approach is based on using CFD to determine the flow field without the influence of particles and then to utilize the 3D velocity flow field results to determine probable particle impact locations and surface impact velocities using a semiphysical model of influence coefficients. Comparative tests of sand flow in a wind tunnel around a NACA 009 airfoil and sand flow inside a high-speed centrifugal compressor showed good agreement of the CFD and semiempirical model predictions. The model properly predicted likely particle impact locations with a miss rate of approximately 20%. This miss rate is likely due to effects related to the laboratory testing and would be smaller in real applications. Details of the test setup and some of the test results and the CFD comparisons were presented.

The above analysis results allow one to develop a strategy to "harden" a gas turbine locally against the damaging effects of sand ingestion and/or to develop improved maintenance techniques. For example, areas that have been identified to have the highest amount of particulate impacts and forces can be locally strengthened or a surface coating can be applied for special protection. By identifying the areas that suffer most particulate damage, pro-active inspection and replacement strategies can be implemented to reduce cost of downtime and repair. The method and tools developed in this project have been successfully utilized in two commercial projects over the last 12 months and will continue to be used in other commercial applications.

## Nomenclature

$A,B,C,D,E,F$ = velocity vector impact coefficients
$R$ = radius from machine centerline
$a$ = particle acceleration (m/s$^2$)
$m$ = particle mass (kg)
$\underline{p}$ = pressure (N/m$^2$)
$\vec{F}$ = Force vector (N)
$\underline{V}$ = Velocity vector component (m/s)
$\vec{V}$ = Velocity vector (m/s)
$f$ = local force (N)
$t$ = time (s)
$v$ = local velocity (m/s)
$\gamma$ = laminar particle impact functions
$\rho$ = fluid density (kg/m$^3$)
$\mu$ = fluid viscosity (Pa s)
$r$ = streamline radius (m)
$\Gamma$ = turbulent particle impact functions
$b$ = binormal
$I$ = inertial
$f$ = fluid
$n$ = normal
$r$ = radial
$t$ = tangential
$o$ = centrifugal
$C$ = Coriolis
$p$ = particle

## References


[1] Simonis, J. C., 2004, "The Silent Enemy: Effects of Sand and Fine Debris From Iraq and Afghanistan on Army Aviation Assets," 6th International Filtration Conference.
[2] Hamed, A., Tabakoff, W., and Wenglarz, R., 2006, "Erosion and Deposition in Turbomachinery," J. Propul. Power, **22**(2), pp. 350–360.
[3] Tabakoff, W., Grant, G., and Ball, R., 1974, "An Experimental Investigation of Certain Aerodynamic Effects on Erosion," AIAA Paper No. 74–639.
[4] Grant, G., and Tabakoff, W., 1975, "Erosion Prediction in Turbomachinery Resulting From Environmental Particles," J. Aircr., **12**(5), pp. 471–478.
[5] Richardson, J. H., Sallee, G. P., and Smakula, F. K., 1979, "Causes of High Pressure Compressor Deterioration in Service," Proceedings of AIAA/SAE/ASME Joint Propulsion Conference, Las Vegas, NV.
[6] Tabakoff, W., and Sugiyama, Y., 1980, "Experimental Methods of Determining Particle Restitution Coefficients."
[7] Cowan, J. B., Tafti, D. K., and Kohli, A., 2010, "Investigation of Sand Particle Deposition and Erosion Within a Short Pin Fin Array," ASME Paper No. GT2010-22362, Proceedings of the ASME Turbo Expo, Glasgow, UK.
[8] Dunn, M. G., Baran, A. J., and Miatech, J., 1996, "Operation of Gas Turbine Engines in Volcanic Ash Clouds," J. Eng. Gas Turbines Power, **118**, pp. 724–731.
[9] Haase, K., and Bons, J. P., 2010, "Velocity Measurements Around Film Cooling Holes With Deposition," ASME Paper No. GT2010-22358, Proceedings of ASME Turbo, Glasgow, UK.
[10] Walsh, W. S., Thole, K. A., and Joe, C., 2006, "Effects of Sand Ingestion on the Blockage of Film-Cooling Holes," ASME Paper No. GT2006-90067, Proceedings of ASME Turbo Expo, Barcelona, Spain.
[11] Land, C. C., Joe, C., and Thole, K. A., 2010, "Considerations of a Double-Wall Cooling Design to Reduce Sand Blockage," J. Turbomach., **132**, p. 031011.
[12] Musgrove, G. O., Barringer, M. D., Thole, K. A., Grover, E., and Barker J., 2009, "Computational Design of a Louver Particle Separator for Gas Turbine Engines," ASME Paper No. GT2009-60199, Proceedings of ASME Turbo Exp, Orlando, FL.
[13] Schmucker, J., and Schaffler A., 1994, "Performance Deterioration of Axial Compressors Due to Blade Defects," Propulsion and Energetic Panel, Symposium, Rotterdam, the Netherlands, April 25– 28.
[14] Ghenaiet, A., Tan, S. C., and Elder, R. L., 2005, "Prediction of an Axial Turbomachine Performance Degradation Due to Sand Ingestion," Proc. Inst. Mech. Eng., Part A, **219**, pp. 273–287.
[15] Hussein, M. F., and Tabakoff, W., 1973, "Dynamic Behavior of Solid Particles Suspended by Polluted Flow in a Turbine Stage," J. Aircr., pp. 434–440.
[16] Brun, K., and Kurz, R., 1999, *Gas Turbine Theory*, textbook published by Solar Turbines, San Diego, CA, in three editions, ISBN 0-615-11751-1.