FEA (Femap and NX Nastran)

Engineering Analysis of Armored Seat Pallets for HH-60G Apache Helicopters: The UH-60L SAR Martin-Baker seat pallet was modified to accommodate a 5 mm thick ballistic grade, hardened steel plate. The original all aluminum design was extensively redesigned to handle the 72 crash landing load cases as required by US Army for structural integrity. Prior engineering analysis work was followed as detailed in LSF00571 for the HH-60G Martin-Baker seat pallet installation stress analysis report. In the revised design, a lightweight aluminum pallet is attached via 18 bolts to a hardened steel plate. The aluminum component was meshed with 8-node bricks and the steel plate with plate elements. The seat structure was modeled using beam elements and was attached to the pallet via seat inserts that slide within rails that are cut into the aluminum pallet. The loading scenario consists of a 225 lb crewman that is subjected to various acceleration loads up to 20g in any one direction. The model was run in full contact model to capture the mechanical behavior between the seat inserts and pallet and the pallet against the armored plate. Results were interpreted under design safety factors that corrected the linear stress values for plastic damage based on MIL-HDBK-5H for 7075 aluminum alloy sheet plate. For example, the plastic bending ultimate allowable for 7075-T7351 plate is 131 ksi (e/D = 2.0) given a yield stress of 59 ksi. Design changes were made based on the stress results and an engineering report created for submittal to the client and review by the US Army. Results were accepted with no comment.

High-Temperature Ceramic Plasma Tube Analysis: An aluminum oxide (Al_2O_3) plasma tube is fabricated by brazing several arc cast Molybdenum focusing rings within the inner tube. Upon cooling to room temperature, the CTE differences between the alumina and moly components have the potential to create high tensile stresses in the ceramic component. Given the specialized geometry of the plasma tube, a detailed finite element analysis was performed. Results from this analysis were used to guide the design of new Mo internal components that drastically reduced the stress levels in the AL_2O_3 tube. A fracture mechanics study for brittle materials indicated at these reduced stress levels, the component would have no problem surviving installation handling and cycling thermal loading. The part is in production today.

LED Glass Panel Inspection Line: This project analyzed a large glass panel conveyor line having a length of approximately 80 meters. Inspection stations were spaced along the line where traveling cameras would be used to monitor the quality of the glass panel. These cameras would move in a non-stochastic manner but would nevertheless set up low and high-frequency vibrations in the line. These vibrations would then deteriorate the camera imaging process and cause other measurement challenges. Modal analysis was performed on key components of the line and major natural modes were shifted to limit camera induced vibration. A transient, modal-superposition analysis was also conducted to check the transmittance of the structure. It was then determined that by mass tuning of the structure, almost all of the camera induced vibration could be eliminated. The final structure was then build based on these analysis results. This line is now operating at several factories and is reported to be much quieter than its predecessors.

Large, Industrial U-Joint Stress Analysis: A 10" diameter U-Joint was analyzed to determine its limit load capacity. Full contact was enforced between the yoke, pin, internal block and drive pin sub-assemblies. Material selection studies were done to determine what would be the best mix of hardened, nitrided steels or a low-alloy tool steel (H1) or a flame carburized 8630. Results from the stress analysis were used to refine an existing design through the use of a bigger drive pin with a larger diameter internal tightening bolt. The whole assembly was pre-loaded and it was noted that bolt preload was an essential engineering variable that should be closely monitored. Fatigue and limit load analysis gave hard predictions between torque and life expectancy. Based on these numbers, the U-Joint was put into production and has operated without failure.

Wind Turbine Tower and Transmission Hub Structural Analysis: A small 5 KW wind turbine tower was analyzed for wind and seismic loading per UBC and ASCE 7-02. The tower was a conical steel tube built up in two sections. A complete rainflow counting fatigue analysis was performed on the tower per UBC specifications. Along with the tower analysis, the turbine power transmission shaft and blade attachment hub were analyzed for peak wind loading and steady-state fatigue damage. Results from the stress analysis work led to several design changes to improves its fatigue life.

National Institute of Science and Technology (NIST) Investigation of Respirator Fit and Function: This was an in-depth detailed project to study the fit and function of a fireman type respirator mask. LS-DYNA was used to perform the fit study of the mask against a simulated human head. The silicone material model was easy to obtain while that for human tissue required some investigation. The best material model fit for human skin was that of a soft rubber compound within an elastic membrane. A transient flow study within the nose cup of the respirator mask was conducted using CFdesign. Inhalation and expiration studies were conducted using standard respiratory breathing curves. Transient flow results indicated that the standard respirator mask creates some turbulence during inhalation that might cause breathing difficulty. Research papers are in process to elaborate upon these results with a shared co-authorship.

ASME Tube and Shell Heat Exchanger Vessel: Based on an initial design as developed by TEMA and ASME codes, a large pressure vessel was analyzed under ASME Section 8, Division 2 specifications with a complete seismic and buckling analysis. An interesting twist to this vessel analysis was that it was based on the client's existing vessel and was woefully under-designed for its intended application. Although one may take analysis work for granted, it is very easy to produce wrong numbers. Predictive Engineering takes great pride that it has never delivered bad results to a client and that in over 15 years of service, not one component analyzed by Predictive has failed due an analysis error.

Sliding and Contacting Auto Body Repair Clamps: General stress analysis work was done on a variety of small sliding clamps for the auto body repair industry. The models would often be composed of several small cast parts that would interact. The FE models would enforce contact between the adjacent parts and then loaded to the required rating. Once the design was optimized and approved for casting, initial samples of the cast part would be tested. The use of finite element modeling has allowed this one company to avoid all initial part failures and potential for downstream lawsuits.

Submarine Analysis Work via ABS and ASME Codes: Predictive engineering has certified two large, manned commercial submarines and a third experimental submarine destined for the Mariana Trench. We have extensive experience in ABS and ASME PVHO codes that allows us to guide the client toward the most optimized design for a manned submersible. Complete FE analysis can be done with a nonlinear buckling calculation to validate the submarine design. Our work has been strain gauged and validated under ABS surveyor requirements.

Transmission Shaft Stress Analysis: Stress analysis on several different types of transmission shafts have been done at Predictive Engineering. Complete transmission boxes have been simulated with interacting gear contact, shaft bearings and the total resultant load transfer to the supporting frame. Motor shaft work has also been done for a motorcycle engine, gear box, and other mechanical equipment.

Normal Modes Analysis of Scanning Electron Microscope Wafer Holder: Although the use of linear dynamics is common place in the world of precision equipment, the development of high accuracy, predictive FE models can be difficult. This work involved the creating of a very detailed FE model of a wafer holder having more than a 1,000,000 elements. Normal modes results were experimentally checked and found to be within 1%. Subsequent follow on designs were explicitly base on the FE work and have been put into production with no downstream vibration performance problems.

CTE Thermal-Stress Analysis of Composites: Fiber reinforced epoxy materials whether reinforced with graphite or Kevlar or glass are prone to developing high internal stresses due to CTE mismatches. The worst combination is that of laminates with Kevlar and graphite fibers. The mismatch between CTE's and the high-strength of the fibers can create high internal stresses in the laminate. Stress analysis work was done on an advanced CFRP composite that was coupled to sections containing blends of Kevlar and graphite layers. Results showed that the client could safely use the structure within the specified temperature range. This was done through the use of 2-D plate models and a complete 3-D analysis of individual layers. The Femap composite laminate modeler was instrumental in generating the FE model.

Electronic Connectors and Springs: The electronics industry enforces strict requirements on connector and contact spring performances. Although these devices are very simple mechanical elements, accurate analysis work is required to obtain useful fatigue results. Work has been performed for several major electronic device manufacturers in the design optimization of connector springs and clips, hook devices, and small mechanical parts. The majority of these parts were fabricated out of fully hardened Be-Cu or precipitation-hardened, martensitic stainless steel (SS 302).

ASME Pressure Vessel Analysis: Extensive experience has been earned in the analysis of dozens of simple to complex pressure vessels. Vessels have been certified under ASME Section VIII, Division 1 and 2 with every possible configuration of tube sheets, nozzle connections, partitions and operations. Work has also been executed under NQA-1 for extremely complex nuclear waste recycling vessels in seismic environments. Buckling analysis was done via ASCE and ASME requirements. In some cases, the buckling resistance of the vessel was certified by UG-2 exception using a non-linear approach to the structure. A complete rainflow counting fatigue analysis was also performed on all structural components, weld joints and piping structures.

Solar Panel Structural Design: Structural design was performed on a broad range of extruded aluminum sections to obtain an optimized design to support gale force wind loads (up to 50 lbf/ft^2). These panels are typically destined for residential and commercial roof tops. As part of the analysis, fatigue predictions were made on the aluminum components and fracture predictions for the glass cover sheet of the panel.

Power Spectral Density (PSD) Analysis, Separation Shock and Pyro Shock: A finite element model was constructed to simulated a broad range of military transport conditions (captive carry), launch (separation shock) and delivery (Pyro Shock) following MIL-STD-810e with reference to Method 514.4 and 516.4. The model was analyzed via PSD and Response Spectrum analysis modes. A fully non-linear transient model (LS-DYNA) was used for the separation shock analysis. Results from this work were used to validate the design of a critical piece of military armament.

Mechanical engineering analysis on a broad range of clamps and hooks used in the auto body repair industry. These clamps often involve multiple parts with contact behavior between highly stressed components. Stress analysis results are used to guide product development and optimize the tool design for weight and biometrics. Not only must the tool have a high strength to weight ratio, it must also fit comfortably in the hand of the user. All stress results are verified on production prototypes. Outstanding correlation between FEA results and tool load carrying capability has been demonstrated over seven years of product development. Modeling notes: FEA models are routinely created from complex sculpted Pro/E geometry files.

A plastic throttle pedal assembly structurally analyzed using the finite element analysis method. A free body diagram was developed to map the force transfer between the pedal arm and the body components. FEA was then performed on each separate part. By performing this analysis as a piece-part job, it was not necessary to implement a nonlinear contact algorithm allowing the analyses to run efficiently and quickly. Modeling notes: FEA models were imported from SDRC I-DEAS iges geometry.

Research program into the application of fracture mechanics toward the fatigue crack growth prediction of cast components. Leveraging experimental fatigue crack growth data provided by the company, fracture mechanics principles were applied in the analytical and FEA calculation of stress intensity factors (Kic). These stress intensity factors were then used to predict fatigue crack growth based on a modified Paris Law crack growth model in experimental castings. A three-dimensional (3-D) crack growth FEA model was also developed to extract full-field crack growth information. The final report showed good correlation between analytical and experimental life-cycle predictions.

Structural analysis to optimize the world's first 100% plastic house and foundation. Finite element analysis was performed on the roof, walls, and foundation structural components. Analysis challenges were found in accurately capturing the large deflection, stress-stiffening behavior of the roof structure and in developing an accurate foundation model. The foundation was particularly tricky due to the non-linear contact between the simulated floor joist and ground connections. Analysis results are being used to drive the design process toward more structurally optimized shapes utilizing less plastic while achieving higher strengths. Insights gained during this modeling effort show that extrusions will work as well as pultrusions for most continuous shapes.

Multi-component FEA model of a ultrasonic transducer head for a medical equipment start-up company. The model included the PZT ceramic transducer, foam backing, brass support structure and a polyethylene cap. Stress and deflection results were obtained based on pressure loading across the face of the transducer head. Based on material property data for the PZT ceramic, the transducer head was certified for manufacturing.

Modal frequency analysis performed on an optical thermal imaging pod used by major aircraft and helicopter manufacturers. The assembly included a mixture of aluminum castings, forgings, and electric sub-assemblies. The analysis model was then excited through a sinusoidal sweep under a prescribed acceleration loading. Harmonics were identified within the structure and compared to experimental shaker table results. Good correlation was shown between the FEA and shaker table results. The final report substantiated that the internal electrical components of the pod would be relatively unaffected by external harmonic excitation.

A broad range of analyses techniques used to virtually engineer stoker grates and their sub-assemblies for the world's largest manufacturer of these critical components within power generation furnace boilers. Stoker grates sit underneath almost every power generation boiler in the world. Their purpose is to support the fuel load (coal, wood chips, food processing waste, etc.), to provide a combustion air stream, and finally, to remove the burnt residue (ash). Vibrating the ash transport mechanism is the massive stoker bed via an oscillating drive. Structural issues arise due to vibration harmonics and temperature induced stresses. Direct transient finite element analysis was used to investigate piping stresses within the stoker grate. Models were built for the complete range of stoker grates using a complex medley of plates, beams, and spring elements. All of these models were subjected to direct transient, direct frequency and modal frequency dynamic analyses. Results from this work were used to optimize the stoker designs and to ensure extended service life. Additionally, thermally induced stresses in large castings were also investigated. Residual stresses arising from thermally induced plastic deformation were found to significantly affect the structural performance, and design changes were implemented in these castings.

Physics based kinematics model of impact hammer used to pulverize coal. Dynamic derived forces were then used to structurally optimize a wear-resistant impact hammer. The goal was to lower the stresses in the hammer allowing the use of a more abrasion-resistant cast iron. Design optimization through the use of finite element analysis facilitated the development of a novel impact hammer using an A-R cast iron that was hitherto unthinkable. A fracture mechanics assessment was also included as part of this investigation.

Forensic FEA work performed for a major supplier of after-market auto parts. A complete suspension module was idealized into a finite element model to allow the correct application of torsional boundary conditions to the sway bar component under investigation. Plate elements were used to model the rear sway arms while solid elements were used to allow the construction of a detailed model of the welded sway bar structure. The two FEA structures were connected together using rigid links and spring elements to simulate the coupling affects of bolts and rubber bushings. FEA results were used to validate new designs and to optimize the final design candidate. Field testing validated the modeling results and the part is now in production.

FEA services provided to the world's market leader in the manufacturing of large, complex, high-quality structural investment castings for the aerospace market. A very large complex investment casting model was analyzed for structural integrity. The wax pattern was modeled using 10-node tetrahedral elements with the investment shell modeled via a surface skin of shell elements. The assembly was then supported through risers and stiffeners attached to a steel platform. This complex assembly was then evaluated under multiple loading conditions. Stress results allowed greater confidence in the integrity of the final production casting.

FEA modeling and optimization work to develop the next generation of Stekel mill coiling drums. Stekel mills are gaining in popularity as a cost-effective alternative to multi-stand steel mills for the production of high-grade plate and sheet steels. A coiling drum is subjected to high stresses and high temperatures as it coils thick plate during the reversing Stekel mill operation. This project work was performed for a large industrial casting operation and it end user steel mill client. Several designs were virtually evaluated for high temperature deflection and strength characteristics. At the end of many design iterations, a new interior rib design was developed that provided 2x greater stiffness and lowered notch stresses at the slot opening. This coiling drum is now in service and performing as designed.

Aircraft landing gear slider. Finite element analysis was performed on a landing gear structure for a major aircraft landing gear manufacturer. Aerospace analysis work requires extreme attention to model construction, mesh quality, and analysis documentation procedure. The engineering idealization of the landing gear part was not trivial. To correctly account for contact behavior between the parts, gap elements were extensively used. With the employment of gap elements, the analysis procedure becomes nonlinear. Typically this is not a hindrance but due to the models large size (approximately 600,000 DOF) run time was a significant factor in the model,s construction.

LS-DYNA

Shock Fixture Tooling with LS-DYNA: The client faced a difficult problem to validate samples of a delicate electronic device that would be subjected to a small explosive fuze. Prior methods involved the sacrifice of an expensive production part to validate each electronic device. Analysis work was first conducted to simulate the explosive shock pulse of the fuze. This was then validated against experimental work. With a calibrated shock pulse, a novel fixture design was developed that allowed the simultaneous testing of eight electronic devices with a single fuze. This result allowed the client to significantly cut their QA costs and speed up the manufacturing process.

Power Spectral Density (PSD) Analysis, Separation Shock and Pyro Shock: A finite element model was constructed to simulated a broad range of military transport conditions (captive carry), launch (separation shock) and delivery (Pyro Shock) following MIL-STD-810e with reference to Method 514.4 and 516.4. The model was analyzed via PSD and Response Spectrum analysis modes. A fully non-linear transient model (LS-DYNA) was used for the separation shock analysis. Results from this work were used to validate the design of a critical piece of military armament.

Drop Testing of Advanced Composite Satellite Terminal Receiver: Full scale drop testing was done of a lightweight composite satellite receiver. Experimental drop testing had shown failures in three key components. A drop test model was constructed using Femap and then analyzed using LS-DYNA. Two of the three failures were perfectly replicated with the FE model showing that the third part should never have failed. Metallurgical examination showed pre-existing casting flaws that had caused the part to prematurely fail. The structure was optimized with a complete redesign of one part from aluminum to a graphite composite structure. The engineering report for this project was submitted for external review by an opposing team of consultants per U.S. Military requirements. The drop test standard for this work was MIL-STD-810e.

Electron Beam Weld Simulation in AlBeMet 162: This project arose due to significant weld induced cracking of thick section (25 mm) high aluminum content beryllium materials (60% Be / 40% Al). In the client’s welding process, the electron-beam would make multiple passes over the same spot to create a vacuum tight weld zone. Due to geometry constraints and EB energies, cracks would occur near the end of the welding process. LS-DYNA was used to simulate this process and was able to accurately predict cracking in the client’s product. The FE model used a traveling energy source that was tuned for the electron-beam welding process. The FE model was able to weld distinct blocks together and also to incorporate welding chills as needed. As the structure cooled down subsequent to welding, residual plastic strains would occur and drive the formation of high elastic tensile stresses. Good correlation was shown between these high tensile stress locations and cracking damage in the part. The welding process was then modified in the LS-DYNA model and it was shown that these elastic stresses could be significantly reduced. Final laboratory work by the client confirmed these numerical results.

Elastic-Plastic Deformation of BGA Lead-Free Solders for Electrical Connector Evaluation: BGA chips by their design use small lead balls to make the final electrical connection to the printed circuit board (PCB). To test these chips, an electrical circuit must be made from the tester to the lead ball. However, it is often a bit more difficult than it sounds since the lead ball must be plastically deformed sufficiently to break through a thin oxide layer and create a robust, low-impedance electrical circuit. This type of mechanical interaction was simulated using LS-DYNA. A Be-Cu spring was pushed against the BGA such that the edges of the spring connector would cut into the lead ball. For this simulation, a lead-free solder composition was used. Simulation results showed that high forces would be required to create sufficient plastic damage on the surface of the lead ball. Given the requirement to simultaneous test hundreds of BGA connections during each test, the force requirements were deemed excessive and the project was cancelled. It should be noted that experimental work using micro-load cells nicely correlated the LS-DYNA results.

Drop Testing and Firing Simulation of an Infra-Red Targeting Scope: Objectives for this analysis work were to determine if the cast magnesium casing would crack upon drop testing onto plywood from three feet and to evaluate the integrity of the internal electrical components and focusing mechanisms during a live firing event. Both models were built with Femap and analyzed using LS-DYNA. The drop testing model was rather simple with the Mg casing modeled with plate elements and the internal components such as lenses, battery pack and circuit boards modeled simply to capture the right center-of-mass and stiffness behaviors. The drop test model accurately predicted that the infra red scope would pass the drop test. The live firing simulation was much more complex and was driven by structural component failures within the scope during the shock event. The LS-DYNA model was able to pin-point the damage mechanism but the actual solution involved a number of fixes that are too lengthy to detail in this brief note. For example, printed circuit board flex (PCB) during the firing event was one failure mechanism that was corrected through the use of additional support pins but a complete internal revision was eventually required.

Compressive Buckling Load Limit for a Large Glass Sphere for Deep Diving Manned Submersible: Engineers that know something about the mechanics of materials realize that high-quality glass is one of true material wonders of the world. Under pure compression, glass can elastically withstand stresses up 500,000 psi and fused silica in excess of 1,000,000 psi. The challenge to engineering with glass and other brittle materials is to maintain the structure under compressive loading. Brittle structures do not fail due to shear loading or pure KIIc fracture growth. In all documented cases of catastrophic failure of brittle structures some sort of tensile stress existed in the structure to start the crack. In hydrostatic loading conditions due to water pressure, most submersible structures are dominantly under pure compressive stress. However, due to openings in these structures via nozzles or hatches, the continuity of the structure is breeched and low level tensile stresses can developed. This problem was eliminated in a unique design proposed by DOER Marine. A 68” glass sphere would have a diametrically opposed 16” openings drilled into the sphere. These openings would act as entry hatches for the three person crew. During operation, a ceramic or titanium plug would provide stress continuity between the glass sphere and the hatch. This design was validated using LS-DYNA to determine its ultimate buckling load. The material model was designed to fail at tensile loads of 500 psi and initiate plastic deformation at 500,000 psi (the start of densification of high purity glass). The wall thickness of the sphere was tapered from 4” at the hatch to 2.5” at the equator. The final tweak for the buckling analysis was to perturb the mesh to account for manufacturing tolerances and localized dimensional imperfections of the sphere. LS-DYNA has a unique approach (*PERTURBATION) where the nodes can be shifted via a number of functions or just in a spectral manner. To bracket the analysis, the sphere was assumed to be out-of-round by 1.0”. The sphere was shown to buckle at a dive pressures of around 60,000 psi. At the depth of the Marianas Trench, the sea pressure is 16,570 psi (a depth of 35,798 ft). A fracture mechanics study was also done on the proposed glass sphere to consider sub-critical crack growth around embedded inclusions. Given the overwhelming compressive stress field during operating, no crack growth would be predictive. These findings were confirmed by external experts hired by DOER Marine. The project was halted in mid-course due to a lack of funding by external sources.

Drop Testing of Large Nuclear-Waste Containers: The U.S. Department of Transportation (DOT) specification 49 CFR 173 calls out a rigorous series of testing procedures for containers that are used in the transport of hazardous waste. One such requirement is that the container survive a drop test from a specified height as determined by the weight of the container. As the weight of the container increases, the drop height decreases. As an example, a 50,000 lbf container must survive a drop from 12" onto its most vulnerable corner. Over the years, as the sophistication of computer programs has increased and the accuracy of the real world simulations has improved, the DOT has allowed the substitution of conservative numerical results for actual drop testing. A numerical standard in the industry for drop testing is LS-DYNA from LSTC (see www.LSTC.com). This explicit/implicit FEA code is quite amazing in its ability to accurately capture extreme nonlinearities in a transient event. Although its bread and butter is the simulation of car crashes, the drop testing of large steel containers and the high speed impact analysis of cars are quite similar. Both involve dynamic events where multiple contacting surfaces must be handled quickly and large plastic strains dealt with in a realistic manner.

Large Strain Flexibility Analysis of Nylon 12 Watch Band: Engineering plastics can be divided into two broad camps: high-strength, low-flexibility (e.g., Polystyenes, PMMA) or low-strength and high flexibility (e.g., rubbers, polyamides).

LS-DYNA is well suited for the modeling of any type of plastic due to its capability to handle almost any type of material constitutive behavior. In work for a major sports equipment manufacture, a series of Nylon 12 watch bands were analyzed through a complete set of movements that simulated the putting on and taking off of the watch band. The material model for Nylon 12 used LS-DYNA's *mat_simplfied_rubber/foam and was constructed using vendor supplied compression and tensile data. The performance of the watch bank was simulated and checked for fatigue damage. The client accepted the results and put the product into major production.

CFD (CFdesign)

Wind loading on Airport Noise Adsorption Wall: Wind loading conditions up to Beaufort Scale 7 were simulated on a large mobile wall structure using CFdesign. Wind load forces were directly mapped from CFdesign to Femap for structural analysis. Wind loads were checked against their upper limits using ASCE 7-02. The wall was designed to fit within a shipping container and is to be fabricated using standard steel tubing. The analysis work followed Universal Building Code (UBC) specifications for structures near human occupancy with a safety factor of 2x. Extreme wind loading conditions were also modeled and a nonlinear buckling analysis was performed to ensure that the wall would not collapse.

Thermal Fluid Analysis of Buried Pipe: This was a very interesting CFD analysis of buried plastic pipes under about eight feet of soil. The CFD work was coupled with a stress analysis models to facilitate a diagnosis of a pipe rupture problem. The CFD model was used to generate thermal profiles in buried piping vaults and in adjacent piping runs (SCH 80 PVC piping). Temperature profiles were then mapped onto the FE model to make predictions about piping stresses. The stress results correlated extremely well with observed in-field piping failures.

National Institute of Science and Technology (NIST) Investigation of Respirator Fit and Function: This was an in-depth detailed project to study the fit and function of a fireman type respirator mask. LS-DYNA was used to perform the fit study of the mask against a simulated human head. The silicone material model was easy to obtain while that for human tissue required some investigation. The best material model fit for human skin was that of a soft rubber compound within an elastic membrane. A transient flow study within the nose cup of the respirator mask was conducted using CFdesign. Inhalation and expiration studies were conducted using standard respiratory breathing curves. Transient flow results indicated that the standard respirator mask creates some turbulence during inhalation that might cause breathing difficulty. Research papers are in process to elaborate upon these results with a shared co-authorship.

HVAC CFD Modeling of Large Power Plant Buildings: Gas turbines for power generation, even with the best co-generation sub-systems, create significant heat loading within the power plant building. To complicate this matter, new noise standards for power plant buildings has required that these buildings be closed to the free flow of air from the outside. In essence, you have a heat generation source within a closed environment. To obtain tolerable internal operating temperatures through-out the year, a full computational fluid dynamics (CFD) study was done on two large power plant buildings. One building was a standard large hall configuration with the steam co-generation turbine at one end of the building the hall containing two large gas turbines. The other building used one massive gas turbine with a large HRSG (Heat Recovery Steam Generator) feeding to a steam generator. The building also contained two aux. boilers. In both buildings, cooling control was the biggest challenge since doors could not be opened during summer due to the noise limitation specification. As such, all cooling was by air conditioners. To limit the cost of air conditioning, only areas where human occupancy might occur were cooled letting other areas be hotter than normal but not to such an extent that safety was compromised. The CFD models were very large running in the range of 10 to 15,000,000 elements. Air flow was managed by placement of air conditioners and vents to the outside since makeup air was part of the equation. Winter conditions were also evaluated with building skin temperatures as low as 1.4 F. The buildings have been placed into service and are operating within the stated specifications as designed.

CFD Thermal Analysis of Speaker Enclosure: Compact stereo devices generate significant amounts of heat due to the high power generation of the amplifiers and face difficult cooling situations due to their tight enclosures, human safety concerns and that fan cooling due to noise generation is not permitted. CFD studies were thus focused on spreading the heat into surrounding structures or increasing the natural convection heat flow through larger heat sinks or better placement of these sinks. At times, the CFD work indicated that the design was simply unfeasible and a major rework of the speaker device was required. Most of these models were run as coupled thermal-fluid mode where air buoyancy (natural convection) was coupled to the solid chip components of the model. This is also known as conjugate heat transfer or conduction-convective heat transfer. In all, a desktop I-Pod stereo system, an amplified boom-box and several powered desktop speaker systems were successfully analyzed for thermal performance. Geometry was prepared from Pro/E and all analysis work done with CFdesign.