Finite Element Analysis — A Numerical Tool for Machinery Vibration Analysis

Robert J. Sayer, Sayer & Associates, Inc., Medina, Ohio

Finite Element Analysis (FEA) is a numerical technique that can be used to approximate the structural dynamic characteristics of vibrating mechanical systems, the understanding of which is paramount to any root-cause failure study involving excessive vibrations. The Finite Element technique can be used for structural dynamic studies of existing equipment or to evaluate the dynamic characteristics of machines and structures prior to fabrication. FEA models can be used to approximate "stress stiffening" effects in rotating mechanical components (e.g., fan wheels, pump rotors, motor shafts, etc.).

Part of any root-cause vibration study should include a solution that addresses the identified problems. Solutions for vibration problems involving the excitation of a natural frequency (resonance) are frequently difficult to obtain based solely upon experimental data. Although an intuitive interpretation of the experimental data may suggest stiffening a component of a machine, the details and the effectiveness of a modification could be difficult to access. Furthermore, in many cases it is difficult to determine the effect of a modification on other natural frequencies that may not currently be a problem, but could become one after implementing the modification.

The finite element method is a numerical technique that can be used to approximate the modal parameters (natural frequencies and mode shapes) of complex structural-mechanical systems. A finite element model is typically much more detailed than an experimental model (operating deflection shape or experimental modal analysis). Finite element models contain thousands of degrees of freedom (DOFs) while most experimental models include hundreds of DOFs or less. The half-symmetry finite element model of a grinding mill included in Figure 1 contains over 100,000 DOFs. Because of this level of refinement, a finite element model provides vibration analysts with an unparalleled approach to evaluating the effect of even the finest details on the structural dynamic characteristics of any mechanical design.

The finite element method focuses on calculating the behavior and response of a continuum consisting of an infinite number of points. In a continuum problem, a field variable such as displacement or velocity contains an infinite number of possible values, since it is a function of each point in the continuum. The task of solving the continuum problem is simplified using a finite element representation that divides the continuum into a finite number of subdivisions called elements. The elements are connected at nodal points into a mesh or finite element model.

The stiffness and inertial properties of each individual element are defined by a mathematical displacement or interpolation function (linear, quadratic, etc.), the elastic material properties (modulus of elasticity, Poisson ratio, and material density), the size of the element and its connectivity and relationship to all other elements in the model. The assemblage of all elements produces an estimation of the stiffness and inertial properties of any complex structural-mechanical system.

It is important to realize that the finite element method is an approximate numerical technique. The accuracy of a solution obtained by finite element analysis depends on several factors, the most important of which are:

1. Degree of refinement of the finite element mesh
2. Appropriateness of the finite element types used to model a machine/structure
3. Boundary conditions used at the limits of the finite element model.

Figure 2 is included as an illustration of the effect that mesh refinement has on the accuracy of a finite element solution. Three separate finite element models were developed to estimate a complex double curvature mode shape of a beam. Each model contained more elements than the previous. The first model was constructed from only two linear elements (3 nodes), the second model used four elements (5 nodes) and the third model used eleven elements (12 nodes).

The first model (only two elements) provided a grossly inaccurate estimation of the complex mode shape. It does not contain a sufficient number of degrees-of-freedom to describe the desired mode. Since this model contains only one internal nodal point, located at mid-span of the beam, it is not capable of estimating the complex mode shape. Rather, it suggests a mode resembling the simple flexural mode of the beam instead of the complex double curvature mode. The inadequate number of elements used in the first model to represent the flexural characteristics of the beam resulted in visual aliasing.

The second finite element model provided a better approximation of the complex mode shape. This model had nodes located at the quarter points of the beam and was able to approximate the double curvature characteristics of the mode shape, albeit not as smoothly as the third finite element model. In addition to providing a better evaluation of the mode shape, the third finite element model also provides a more accurate estimation for the mode's natural frequency. The number of elements required to provide good results is dependent upon many factors, including the complexity of the mechanical system, the complexity of the mode shape being studied, element types, material properties, nonlinearities, etc.

Boundary conditions also effect the accuracy of the finite element solution. Boundary conditions are mathematical constraints applied to the finite element model to account for items that are not directly included in the model. For example, the modal characteristics of a rotor are usually estimated by a finite element analysis of the shaft and impeller. The bearing conditions are entered into the model as either rigid constraints or elastic springs. The accuracy of the finite element solution is dependent upon the accuracy of the bearing stiffness used in the analysis.

For cases where equipment and structures exist and are in operation, experimental data can be used to correlate finite element models, providing insight into the nature of boundary conditions. Impact tests can identify natural frequencies and suggest the amount of motion at bearings, information that can be used to calibrate a finite element model. The refinement of the model and the numerical value of boundary conditions can be modified until good correlation is achieved between the numerical calculations and the experimental results.

The appropriateness of the finite element type used to describe a structural or mechanical component can also effect the accuracy of the finite element solution. For instance, it may be inappropriate to use thin plate elements to model a thick concrete machine foundation. An assemblage of three dimensional solid elements would provide a more accurate solution.

Based on a paper presented at the National Technical Training Symposium and 27th Annual Meeting of the Vibration Institute, New Orleans, LA, July 2003.

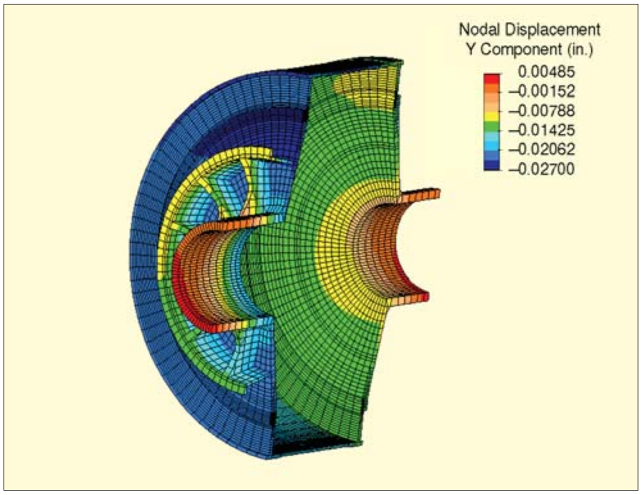

Figure 1. FEA model of grinding mill.

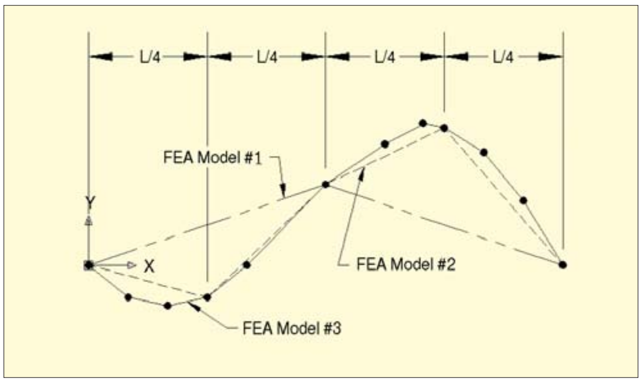

Figure 2. FEA results for beam mode shape.

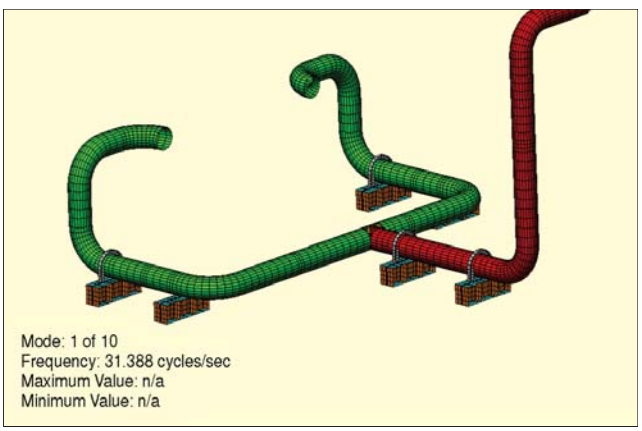

Figure 3. FEA model of piping systems.

Structural Dynamic Modification Studies

Because a finite element model provides a more detailed mathematical description of a mechanical system than an experimental model, it is well suited for structural dynamic modification studies. An experimental model is typically a stick diagram that vaguely resembles the mechanical system being studied. The evaluation of structural modifications using experimental models is very limited and should be done only with an understanding of the approximations involved.

Figure 3 contains a three-dimensional view of a finite element model of a piping system. The piping and supports were experiencing excessive vibrations and fatigue failures due to the excitation of resonance by pressure pulsations produced by a reciprocating compressor. This piping system was analyzed using both experimental and numerical (finite element) techniques.

Figure 4 is an experimental modal analysis (EMA) model of the same piping system. The finite element model is more detailed than the EMA model. Although the experimental model was sufficient to describe the modal parameters of the piping system (natural frequencies and mode shapes), it did not contain enough resolution to accurately estimate the effects of proposed structural modifications. The actual stiffness of the pipe supports were affected by the local flexural deformation of the beam used as a pipe support. A finite element model was necessary to investigate local stiffening of pipe supports that would have been difficult with the experimental model.

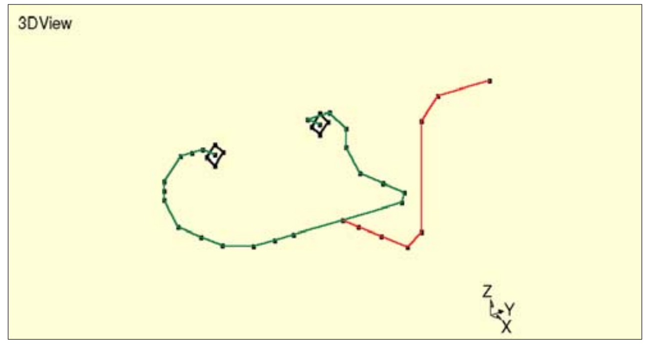

Figure 4. EMA model of piping sytem.

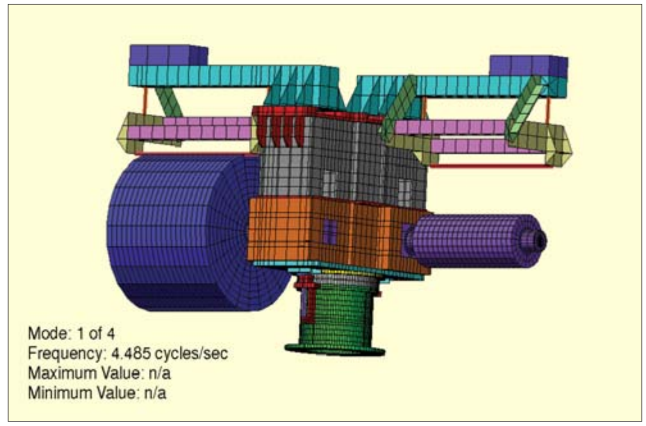

Figure 5. FEA model of winder.

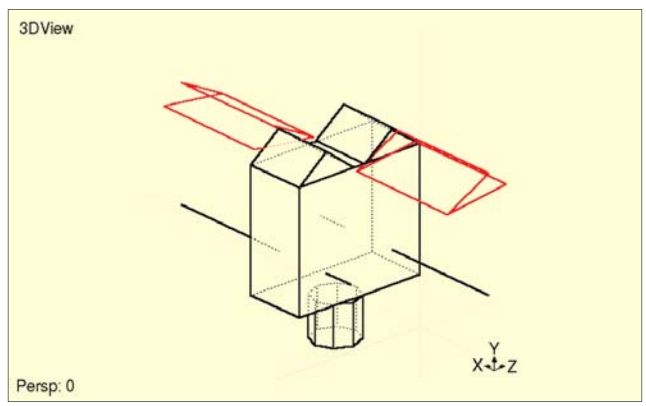

Figure 6. EMA model of winder.

Another example of the difference between finite element and experimental modal models, related to a structural dynamics modification study, can be seen in a comparison of Figures 5 and 6. These figures represent models of a winder machine that had been experiencing problems relative to the excitation of natural frequencies.

The process line that the winder serviced operated at a constant speed. The rotational speed of the winder varied during the winding process since the diameter of the coil was constantly changing. Several resonances were excited during the winding process as the frequency of the dynamic force produced by the rotating elements of the machine coincided with natural frequencies.

Figure 5 is the finite element representation of the structural-

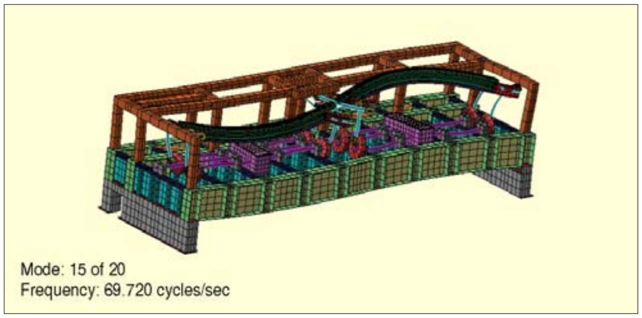

Figure 7. Mode shape of proprietary machine.

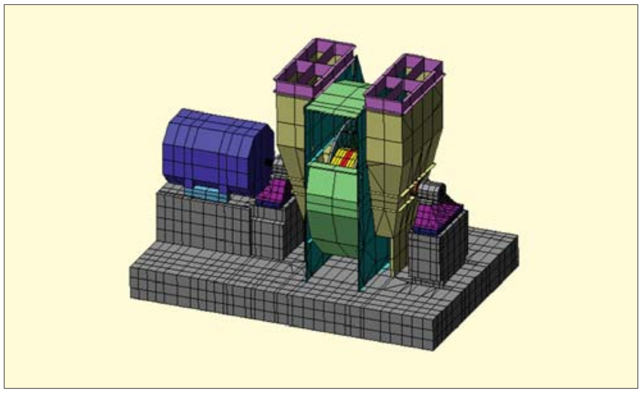

Figure 8. FEA model of fan foundation.

mechanical system. The experimental modal model is contained in Figure 6. The degree of refinement of the finite element model is clearly superior to that of the experimental model. Although the experimental modal model was sufficient to experimentally determine the natural frequencies and mode shapes of the winder mechanism, it did not contain the refinement necessary to accurately evaluate the expected effect of proposed modifications.

The experimental model was sufficient to serve as a valuable root-cause analysis tool. However, the finite element model was required to develop design modifications directed at removing the resonance excitation problem.

Structural Dynamic Design Audits

The finite element method provides the vibration analyst with a numerical technique to evaluate the structural dynamic characteristics of a machine and/or structure during the design stage, prior to fabrication. Natural frequencies and mode shapes can be predicted to ensure that equipment does not operate at or near any resonant frequencies.

The stiffness of supporting structures/foundations can be evaluated. Areas of structural weakness or probable resonances can be addressed during the design stage where changes are easily accomplished. Once a structure or machine is installed, structural modifications may be limited by access restrictions, difficult to implement due to the nature of the problem (e.g., soil-foundation interface stiffness), and very expensive.

An analysis that supports the structural dynamic design of a mechanical system can minimize potential problems in the field and reduce the cost of a project. The cost to modify a machine or structure in the field (after installation) can be very expensive. Vibration problems originating from the resonant excitation of equipment can result in delayed plant start-up and increased maintenance outages. The cost of lost production can be tremendous, evaluated in some cases as millions of dollars per day.

Figure 7 contains the results of a finite element model developed to assist with the design of a proprietary machine. The finite element model was used to evaluate the structural dynamic characteristics of the proposed machine design, identify potential problem areas, and provide corrective changes during the design stage. This is definitely a pro-active tool,

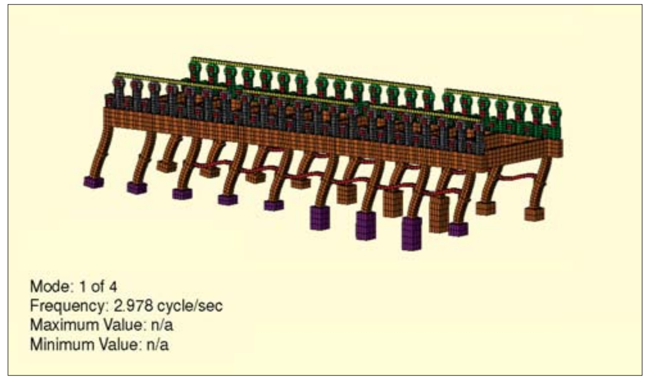

Figure 9. Mode shape of paper machine.

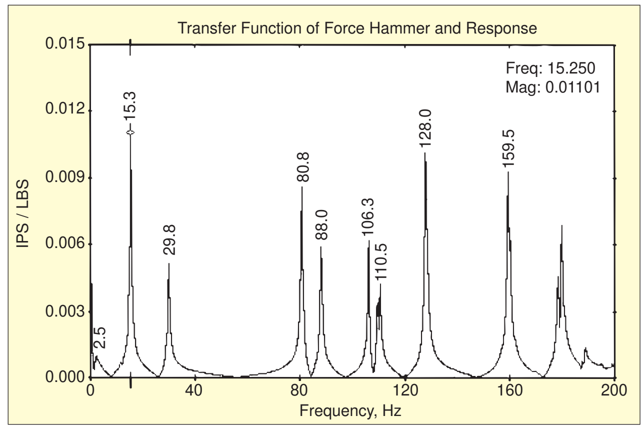

Figure 10. Impact test results for fan.

providing root-cause analysis prior to fabrication and installation.

Finite element evaluation during the design stage is recommended for any machine that could have a significant impact on the productivity of an operation. The cost of a numerical study is insignificant compared to that of delayed project completion, lost production and/or equipment repair or replacement after installation.

Finite element modal evaluations are not limited to mechanical equipment. In many cases, although the structural dynamic characteristics of a machine may be acceptable, the structural supporting system or foundation under that machine experiences resonance problems, leading to excessive vibrations, premature mechanical failures, and productivity issues.

Figure 8 includes a finite element model developed to estimate the modal characteristics of a foundation for a large industrial fan. Problems involving soil-foundation interfaces are very difficult and extremely expensive to correct after the foundation has been installed. The finite element method provides a technique whereby the vibration analyst can evaluate the foundation for possible deficiencies during the design stage, thus circumventing resonance problems during a phase where modifications can be made on paper.

Modal Studies of Large Structures

In some cases, the mechanical system of concern may be very large, making experimental modal testing impractical. Finite element analysis can be used to provide a numerical estimate of the modal characteristics of these large structural/mechanical systems.

An example is a dryer section of a paper machine that exhibited sensitivity to machine speed. Vibrations of a large reinforced concrete structure that supported the dryer increased nearly exponentially with increasing machine speed, supporting the theory that excessive vibrations could be a result of resonant excitation of the structure. However, the structure was too large to excite experimentally.

A finite element model, shown in Figure 9, was developed

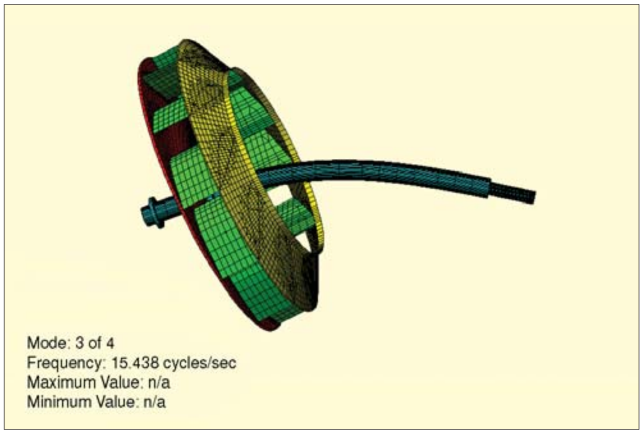

Figure 11. FEA results for fan rotor.

to assist in the root-cause analysis of the vibration problem and to serve as a basis upon which the effect of structural dynamic modifications could be studied. The finite element model confirmed that excessive vibrations were the result of the excitation of natural frequencies as the paper machine reached certain line speeds. The finite element model was then used to develop modifications to minimize resonance excitation. More importantly, the finite element model was used to evaluate the probability of resonant problems for future machine expansions involving increases in line speed of the paper machine.

FEA of Rotating Components

Impact tests performed to identify the natural frequencies of rotating components of machines are performed while the machine is "at rest." During operation, as the machine part rotates, the natural frequencies can change due to centrifugal stiffening and gyroscopic effects. The amount of increase due to centrifugal stiffening and gyroscopic effects can vary significantly and is dependent upon several factors including the rotational speed, stiffness and mass of the rotor.

For cases where an impact test identifies the "at rest" natural frequency of a rotor to be less than the operating speed, it is important to evaluate the effect of centrifugal stiffening and gyroscopic effects to insure that the rotor does not become resonant during operation. Excitation of the natural frequencies of a rotor, such as a fan wheel and shaft, can lead to sudden and catastrophic failure.

Figure 10 contains the results of an impact test performed to identify the "at rest" natural frequencies of a centrifugal fan rotor, including shaft and fan wheel. According to the impact test, the principal natural frequency of the fan rotor was 15.3 Hz (918 rpm). The nominal operating speed of the fan was 20 Hz (1200 rpm). Since the "at rest" natural frequency was less than the operating speed of the fan, it became necessary to evaluate the effects of gyroscopic and centrifugal load stiffening.

A finite element model, shown in Figure 11, was developed to determine the modal parameters of the fan rotor. It was correlated to the experimental data, providing bearing stiffness information that was used in the boundary conditions for the numerical model. The number and type of elements (refinement of the model) were modified until agreement between numerical predictions and experimental data was reached. The final correlated finite element model provided an estimate of 15.4 Hz (924 rpm) for the "at rest" natural frequency. The numerical estimation was very close to the experimental data. The subsequent stress-stiffened finite element analysis indicated that the expected increase in natural frequency during operation would not be sufficient to remove the fan from possible resonant excitation. Either the shaft or fan wheel would have to be modified. The finite element model was then used to develop an acceptable modification.

The finite element method can also be used to estimate stress levels in rotating equipment during operation. This includes

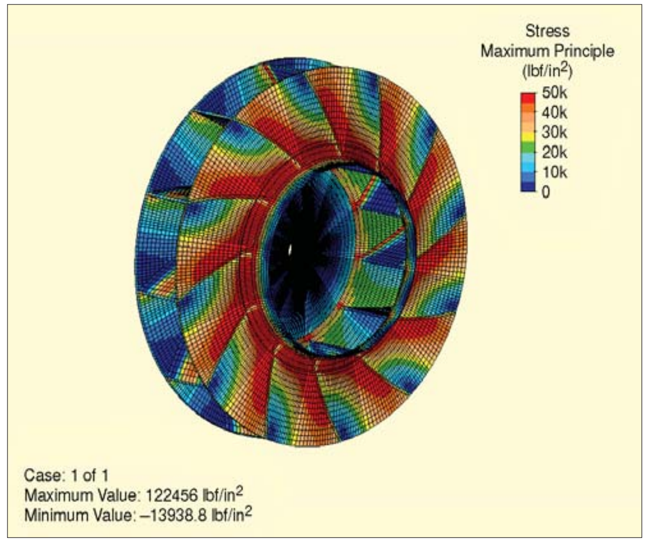

Figure 12. Centrifugal stress in fan wheel.

both centrifugal stresses and stresses due to the application of dynamic loading, such as unbalanced forces. The stress results can then be input into a fatigue algorithm to predict the life of a mechanical component. Centrifugal stresses are generally associated with low cycle fatigue – being applied and removed once per start-up. Stresses due to dynamic forces produced by rotating equipment are typically associated with high cycle fatigue – being cycled once or more per revolution. Figure 12 contains the centrifugal stress contours estimated by a finite element model of a fan wheel. The finite element analysis indicated massive yielding in the inlet shroud at each blade, confirming the suspected failure mechanism of the wheel.

Conclusion

Experimental methods such as spectral analysis, operating deflection shape testing, and experimental modal analysis can be used in any root-cause study to identify the source and nature of a vibration problem. However, using the experimental data to formulate solutions to the problems identified during the root-cause study can, in some cases, be very difficult. The finite element method provides vibration analysts with a powerful numerical tool that can be used to evaluate the effectiveness of proposed solutions to vibration problems, regardless of the complexity of the modification. Because a finite element study uses numerical models that have a greater degree of resolution and refinement than experimental models, results obtained from a finite element analysis may be more accurate than those of any experimental model.

Finite element analysis is a valuable tool for evaluating the structural dynamic characteristics of machines and structures during the design stage, even prior to prototyping. The method can be used to estimate operating stress levels and fatigue life of mechanical components and to estimate the natural frequencies and mode shapes for equipment and supporting structures.

Acknowledgment

The author wishes to acknowledge Mr. Arthur R. Crawford, who has been an inspiration and mentor in incorporating experimental data into numerical finite element analysis. In many cases, the accuracy of a finite element analysis would be questionable without the experimental confirmation of boundary conditions and structural nonlinearities. Mr. Crawford's contributions have been instrumental to the success of many structural dynamic studies.

The author can be contacted at: sayerinc@bright.net.