

## STRUCTURAL ANALYSIS SKILLS

\*\*\*This document is held up to a basic level to represent a sample for our both theoretical background & software capabilities/skills. (Click on each link to see the detailed explanations)

### Pre&Post Processing

- [Linear – Nonlinear Static Stress Analysis](#)
  - [Buckling Analysis](#)
  - [Enhanced FEM Modeling of CAD Parts](#)
- [Meshing of CAD Parts/Assemblies According to the Customer Specified Mesh Qualities](#)  
(warpage, skewness ratio, mesh size, interior quad-tria angles, aspect ratios etc.)  
(Special skills on 2D Sheet metal shell meshing – Full control of the mesh)
- [Automated Reporting Excel-Word Macro Setup for Large Databases with up to 10.000 & more Loadcases](#)
  - [Automatic Stress Analysis load/loadcase creation/update via command files.](#)
- [Automatic Preprocessor & Solver command files' combined Analysis Setup for Automated Stress Analysis Procedure.](#) (Following steps performed sequentially at one click: Database run for FEM input file, solver solution, Attachment of the analysis output file to the database, automatic stress reporting at specified elements/nodes, combined hand calculations, listing of the specified critical calculation solutions with the relevant loadcase names)
- [Automatic Stress Analysis FEM Model Creation, Stress Analysis & FEM modifications with Iterational Algorithms, where Parametrization of the CAD Parts are applicable.](#)

### Solver Parameters

- [Enhanced Control of the Fatal Errors Occuring via FEM Modeling](#)  
(Listing of the fatal errors & determining the solution procedure either by FEM Model Preprocessor modifications or Solver's own parameters)
- [Integrated Solutions to Solver/Preprocessor or Computer Hardware Capability Problems for Large FEM Model files](#) (100-200 MB - megabytes) according to the analysis needs.

### CAD Parameters

- Basic CAD modifications on the original CAD data delivered by the customer for FEM modeling purposes.

### **Linear – Nonlinear Static Stress Analysis**

- ✓ Linear-Nonlinear Static Stress Analysis FEM model creation, application of boundary conditions.
- ✓ Basic Structural Thermal Analysis.
- ✓ Application of Structural Thermal loads for Linear Static Analysis either by thermal field functions or “nodal thermal load applying” command files created from Thermal Analysis reports via macros.
- ✓ All kinds of 1D, 2D & 3D element usage involved according to the modeling needs. (Spring, Gap, Bar, Beam, Rod, Quad, Tria, Hex, Wedge, Tet & etc.)
- ✓ FEM theory held up to a quite good level for FEM modeling and stress reporting purposes & decisions.
- ✓ Residual Stress reporting via Nonlinear Static Analysis Loadcases.
- ✓ Stress Reporting for Fatigue Analysis & Simple Fatigue Life Calculation applicable with the delivery of the fatigue formula to be applied for the FEM model by the customer.
- ✓ Finite element analysis via Preprocessor and Solver programs & Stress reporting of the target elements either by Mathcad, Excel, Combined Excel/Visual Basic macros or Combined Mathcad/Excel/Visual Basic macros.
- ✓ Necessary Hand calculations’ setup via Excel, Combined Excel/Visual Basic macros & Combined Mathcad/Excel/Visual Basic macros. (Fastener/Joint calculations including bearing, nut pull through, joint combined RF; critical von mises/max principal/max shear/max tension-compression & etc.)
- ✓ Loadcase elimination of the relevant FEM model to shorten the Solver analysis runtime (where elimination of the loads is applicable & the whole FEM model analysis in solver program take more time than the limited project time – for instance a FEM model with 100.000 elements and 10.000 loadcases take months to complete the analysis only in most solvers. This procedure has great advantage in such big FEM model applications).

[GO TO THE TOP OF THIS DOCUMENT](#)

### **Buckling Analysis**

- ✓ Basic Buckling Analysis according to the boundary conditions of the FEM section.
- ✓ Buckling analysis of “a section of the FEM model” via “Linear Static Analysis nodal results of the boundary nodes of the selected section”.
- ✓ Hand calculation buckling analysis by “the nodal force results” of the interfaces of a specific section with respect to the relevant Stress Manuals. (Combination of Hand Calculation Buckling & Linear Static Analysis)
- ✓ Buckling hand calculation applicable for rectangular flanges and webs with “4 edges hinged, clamped”, “3 edges clamped-1 edge free”, “3 edges hinged-1 edge free” & similar boundary conditions.
- ✓ Crippling analysis of several beam sections. (I, Z, J, T, V, multicorner sections & etc. ) with respect to the relevant Stress Manuals.
- ✓ Column/Panel buckling analysis with respect to the relevant Stress Manuals.

[GO TO THE TOP OF THIS DOCUMENT](#)

### **Enhanced FEM Modeling of CAD Parts**

- ✓ CAD geometry import to preprocessor program with the appropriate file format.
- ✓ Simplification of the CAD geometry according to the analysis criterias.
- ✓ Midsurface extraction if the CAD model is sheet metal & is to be modeled with 2D elements (Quad,Tria).
- ✓ Solid meshing of the CAD parts where needed. (Hex, Tet, Wedge elements)
- ✓ Good understanding of the capabilities of the mesh elements. (Spring, Gap, Bar, Beam, Rod, Quad, Tria, Hex, Wedge, Tet & etc.)
- ✓ Differences between different 1D element types & different 2D element types well identified by hand calculations & applications.
- ✓ Necessary modifications on the FEM model where 1D to 2D element conversion is needed. (for instance: connection of 1D beam element to 2D modeled (with quad elements) beam element with relevant MPC application at the connection points etc.)
- ✓ Good control on the mesh (Meshing performed using all the tools of the preprocessing program used)
- ✓ Mesh Quality Checks with respect to the customer meshing criterias.
- ✓ Good control on the grouping of the geometry, FEM elements, nodes & their Ids where these have to be setup in “a specific format requested the customer”.

[GO TO THE TOP OF THIS DOCUMENT](#)

### **Meshing of CAD Parts/Assemblies According to the Customer Specified Mesh Qualities**

- ✓ CAD geometry simplification in the preprocessor program.
- ✓ Meshing of 2D – 3D parts decided logically & all geometry modified for this final logic.
- ✓ CAD part meshed with the modified final geometry.
- ✓ Checks performed according to the customer specified mesh qualities & necessary modifications applied to the final mesh. (warping, skewness ratio, mesh size, interior quad-tria angles, aspect ratios etc.)
- ✓ For assemblies, exact representation of the original assembly is the main target and consideration on carefully taking both the FEM theory and the real-life behaviour of the working assembly into account.
- ✓ Test run of the assemblies to achieve exact stress load paths & Necessary modifications implemented on the final FEM to eliminate any faulty modeling causes.

[GO TO THE TOP OF THIS DOCUMENT](#)

**Automated Reporting Excel-Word Macro Setup for Large Databases with up to 10.000 & more Loadcases**

- ✓ Good hold of large databases from FEM modification start to end.
- ✓ Automatic Stress Reporting macros' setup for the specified elements of any FEM analysis.
- ✓ Automatic multiple "solver output files' attachments to the FEM databases" & reportings via Excel macros.
- ✓ Stress & any kind of result extraction from the output text files of the solver where extraction from output text file is more efficient than "database attachable results file of the solver. (Depending upon the type of result)
- ✓ Automatic results table creation via preprocessor program, solver program, Excel & Word.
- ✓ Automatic stress reporting applicable where the parts of the stress report can be created parametrically for a repeating stress analysis procedure.

[GO TO THE TOP OF THIS DOCUMENT](#)

**Automatic Stress Analysis load/loadcase creation/update via command files.**

- ✓ Application of any parametric load data matrice sent by the customer (Force, Moment, Pressure, Temperature etc.)
- ✓ Combining of the input load data “sent by the customer” with the loadcase creation/update command files of the preprocessor.
- ✓ All possible combinations of the loads in a loadcase with the necessary preprocessor command file creation algortihms.
- ✓ Nodal/Element field applications included.
- ✓ Parametric Field function applications available where needed.

[GO TO THE TOP OF THIS DOCUMENT](#)

### **Automatic Preprocessor & Solver command files' combined Analysis Setup for Automated Stress Analysis Procedure**

- ✓ Analysis procedure from Preprocessor>mathematical FEM model file creation to Excel>Stress reporting
  - Parametric mathematical FEM model file creation
  - Parametric Solver analysis with the relevant Solver Parameters
  - Detection of Fatal errors if any.
  - Parametric Solver output file(database analysis results file) attachment to the database.
  - Parametric specific stress results (report file) extraction.
  - Parametric Excel table creation (or Parametric Hand calculation algorithm execution on the results of the analysis)
  - All above steps at one click after the correct setup of the whole algorithm.
  - Checks to approve the automation of the analysis procedure.
- ✓ Any preprocessor command file can be parametrized and created rapidly via macros according to the loadcase names, multiple application nodes'/elements' Ids, transformation coordinate systems, stress reporting types, preprocessor reporting modules and etc. (almost all combinations available)

[GO TO THE TOP OF THIS DOCUMENT](#)



**Automatic Stress Analysis FEM Model Creation, Stress Analysis & FEM modifications with Iterational Algorithms, where Parametrization of the CAD Parts are applicable.**

- ✓ Applicable mostly on the parametrical FEM/CAD models.
- ✓ Whole analysis procedure from FEM Model Creation (with all the element & material properties referenced from other files) to final Analysis results both for linear, nonlinear static & buckling analysis can be automatized.
- ✓ The analysis results of the FEM may be read by an optimization algorithm depending upon the design & critical stress analysis regions of the CAD part.
- ✓ An update on the input parameters (FEM Geometry data, material data, mesh quality data & etc.) of the FEM model can be implemented automatically right after the first run of the analysis.
- ✓ The analysis for the modified FEM model can be applied.
- ✓ Depending upon the relationship between the input parameters & outputs from the analysis, a suitable algorithm can be setup to find the optimum (final) design of the CAD part.

[GO TO THE TOP OF THIS DOCUMENT](#)

### **Enhanced Control of the Fatal Errors Occuring via FEM Modeling**

- ✓ Problem handling in solver program where fatal errors occur due to modeling or faulty boundary conditions' application.
- ✓ Removal of singularities occuring during the analysis.
- ✓ Good preprocessor/solver combined logic to solve the numerical problems faced.
- ✓ For macros (which use both preprocessor & solver programs as tools) that take 150-200 hours to extract the final results of the whole stress analysis procedure, effective control on the preprocessor/solver configuration files is achieved (In massive data projects).

[GO TO THE TOP OF THIS DOCUMENT](#)

**Integrated Solutions to Solver/Preprocessor or Computer Hardware Capability Problems for Large FEM Model files (100-200 MB - megabytes) according to the analysis needs.**

- ✓ Control on the various parameter applications both in Preprocessor&Solver programs to use the whole computer hardware capability.
- ✓ Solver configurations' control whenever needed.

[GO TO THE TOP OF THIS DOCUMENT](#)