

***A powerful FEM system***



***for all CAD systems***

## ***With MEANS V12 to the optimal product***

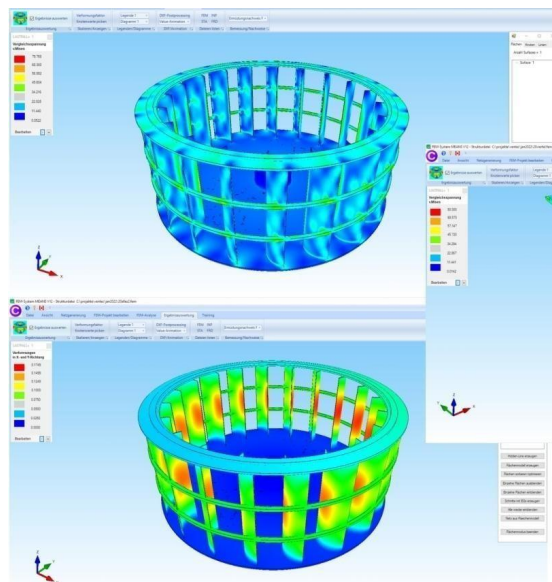
The finite element method (FEM) is the method of choice, for example to optimize individual workpieces or entire assemblies with regard to the ratio of weight to stability via simulation. This method saves a lot of money, since it makes it possible to reduce real tests, for example on automobiles, airplane wings or tools, to an absolute minimum. A particularly powerful representative of this type of software is MEANS, which has proven itself in practical use since 1989.

Space-filling computer giants are no longer necessary for the preliminary analysis of CAD models using the finite element method, since even normal PCs from discounters now have a sufficiently high level of performance to master such calculations. A high computing power is a mandatory prerequisite, because the formulas and calculation processes behind this type of program have it all.

It is necessary to determine the deformation of a part or the complete area of a component when subjected to temperatures and forces. Due to the fact that a tight network of triangles is placed on the area to be examined for the simulation, a large number of calculation steps have to be carried out, with the starting conditions for the next triangle resulting from the results of the previous calculation of a triangle that has already been processed.

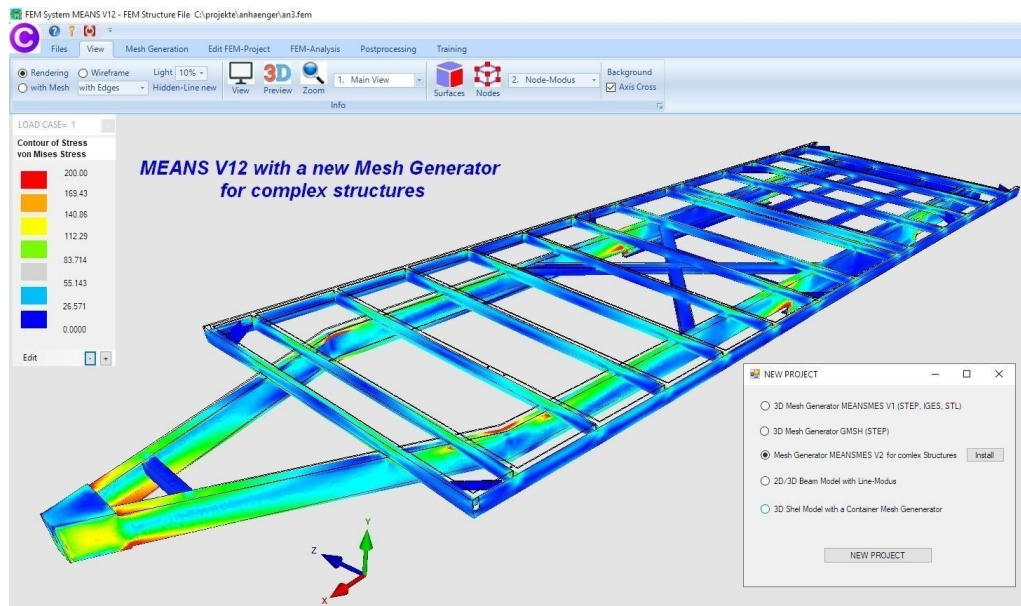
In this way, the effects of the temperatures and forces acting on the component are determined piece by piece. At the end of the calculation, the system outputs a false-color image and an associated color table that provide information about the calculation result.

The MEANS program, which supports the neutral formats DXF, STL, IGS and STEP and can calculate both 2D and 3D models, is ideal for professional purposes. It has a modern ribbon user interface, so that the operation is not a mystery.

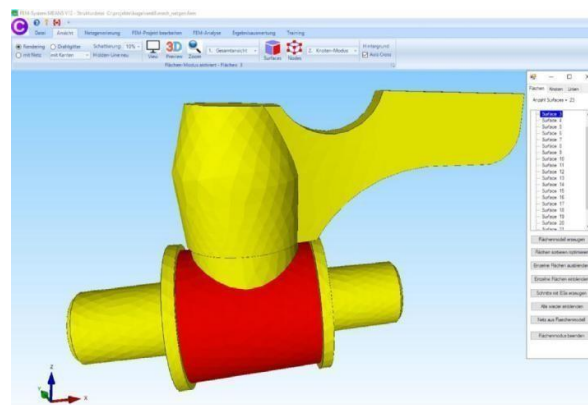


***FEM-Simulation eines Ventilators unter Fliehkraft***

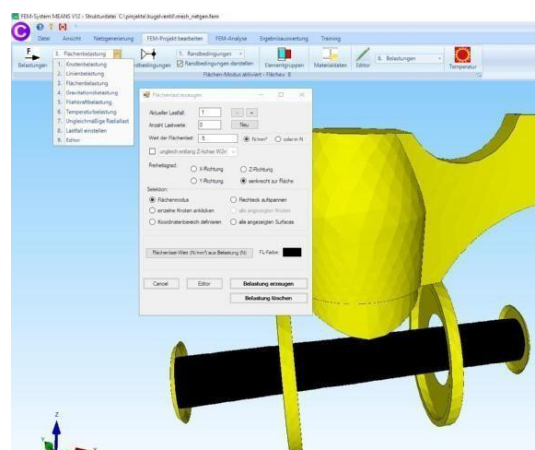
***mit FEM-System  
MEANS V12  
www.femcad.de  
www.fem-infos.com***



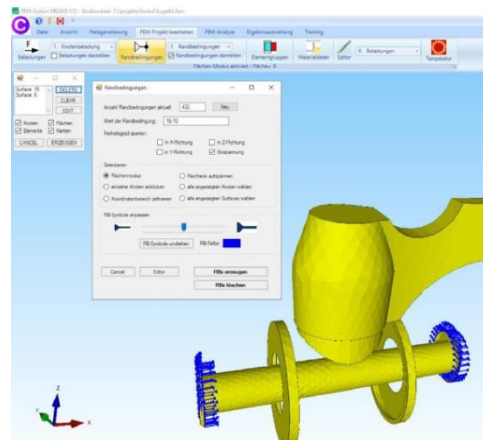
After the mesh generation must the FEM mesh in the MEANS format exported become. Through this it becomes possible to create a surface model that allows surfaces to be assigned for loads or boundary conditions select.



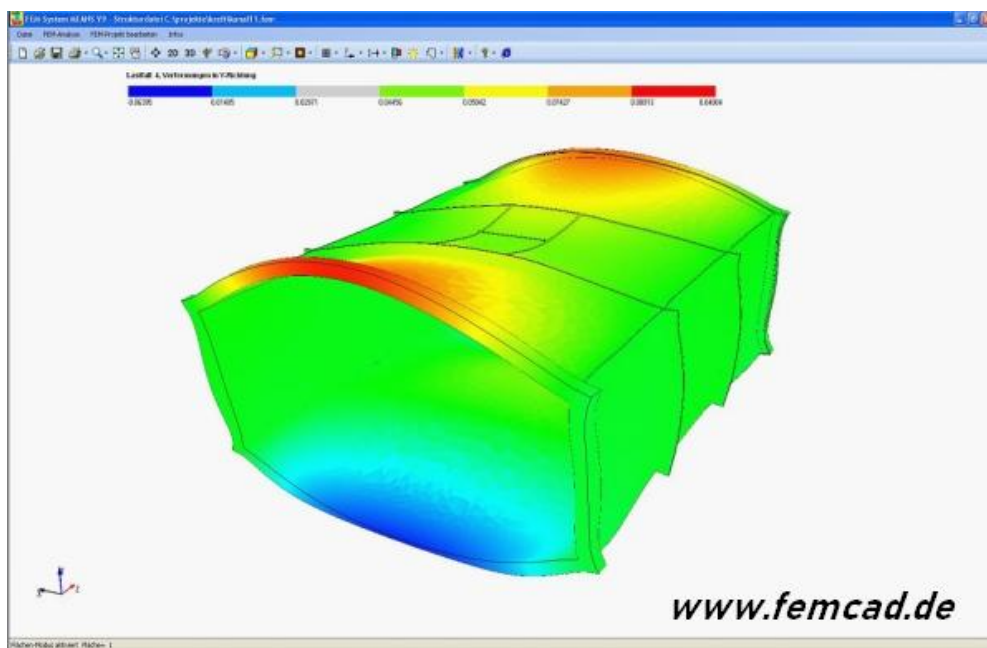
If, for example, the pressure load on the inner tube is to be determined in this example, outer surfaces can be hidden in order to expose the inner tube. The surface load can then be generated, for which some information must be entered in various dialog boxes.



Of course, boundary conditions can also be taken into account so that the FEM calculation is as realistic as possible. Fixed supports as well as spring constants and elastic beddings can be used.



Before a simulation, it is also important to specify material data, after all, steel behaves differently than aluminum, for example, when exposed to temperature and force. A material database is available in MEANS for this purpose. After all data has been entered, the simulation can be started. The result, for example, is the deformation in the X, Y and Z directions with a deformation factor.



It remains to be noted that MEANS can be used in many different ways. This means that tasks in statics can be solved just as easily as natural frequencies, buckling loads, temperatures, shape optimization or finding optimal contact conditions between several components.

#### Contact details:

Engineering office HTA software - Maiwaldstraße 24 - 77866 Rheinau

Telephone 07844-98641 - Website [www.femcad.de](http://www.femcad.de) - Email [info@femcad.de](mailto:info@femcad.de)