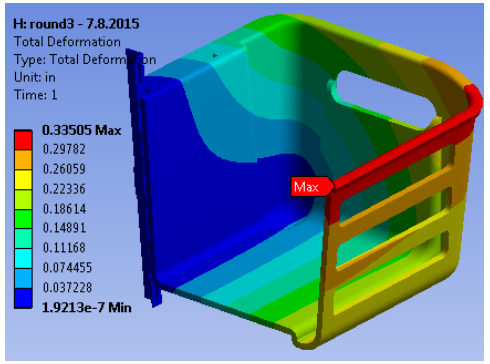
Introduction of FEA

Basic concept, application, capabilities, etc.

1. What is FEA

- **Finite Element Analysis (FEA)** is a numerical method to find approximate solution of equations (from mathematics):

$$\begin{cases} a_1x_1 + b_1x_2 + c_1x_3 + \cdots = F_1 \\ a_2x_1 + b_2x_2 + c_2x_3 + \cdots = F_2 \\ a_3x_1 + b_3x_2 + c_3x_3 + \cdots = F_3 \\ \quad \cdots\ \cdots\ \cdots\ \cdots\ \cdots\ \cdots\ \cdots\ \cdots\ \cdots\ \cdots \\ \quad \cdots\ \cdots\ \cdots\ \cdots\ \cdots\ \cdots\ \cdots\ \cdots\ \cdots\ \cdots \end{cases}$$



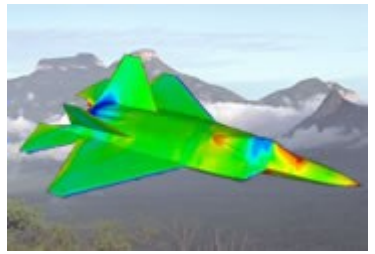
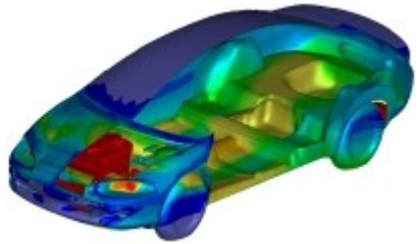
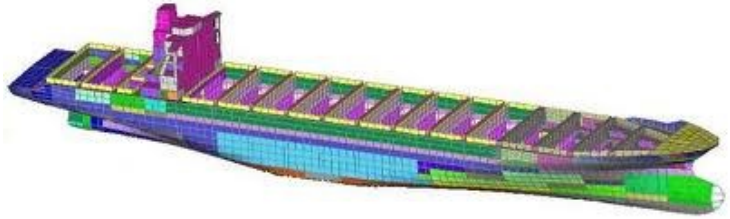
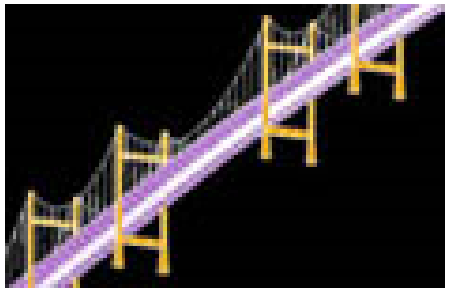
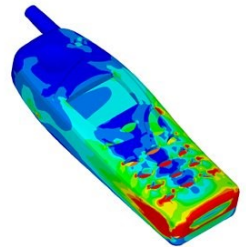
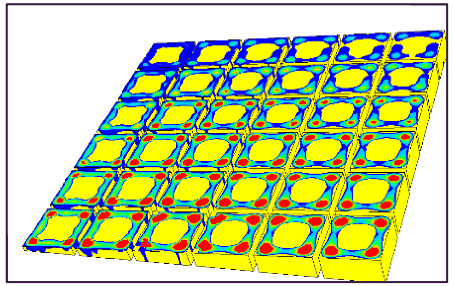
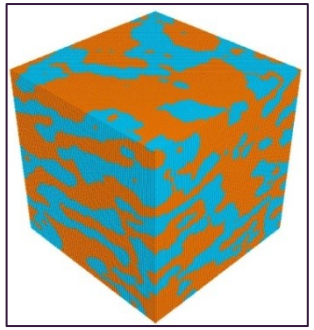
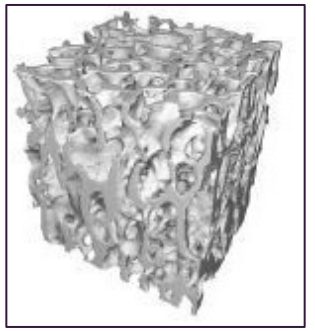
- FEA is a numerical simulation tool used to predict structure response to the given load/testing condition, optimize the product design, determine the structure strength, deformation, temperature, weak area, etc. (from applications)

2. What FEA Can Do

- Find out potential design issues before the prototype stage.
  - ✓ FEA does virtual lab test on all kind of products, simulate and predict the behavior of structures under given load conditions.
  - ✓ Predict stress, deflection, safety factor, temperature, buckling deformation, fatigue failure, etc.
  - ✓ Evaluate the structure strength, find the weak point and potential design issues, optimize the structure design, etc.
- FEA can do simulation that can't be done by real lab and prototype test
  - ✓ Extreme condition: High/low temperature; Test Equipment Unavailable; Schedule Impractical; Long time; special location (Outer space, deep sea).
- Save time & cost

3. Application fields:

- Aerospace
- Automotive
- Construction
- Civil Engineering
- Chemical Engineering
- Consumer Products
- Electronics
- Heavy Equipment
- Machine Parts
- Medical
- MEMS
- Plastic Engineering
- Power Generation
- Sports
- Tooling
- Transportation
- Etc.

4. Why choose FEA Simulation

- Analysis and predict product performance start from the concept stage
- Find out design defects before prototype, and provide design direction for design/product engineers
- Optimize the product structure to achieve the best performance with the lowest cost
- Save time, material and cost. Contribute to faster and less expensive design cycle
- Enhance design and provide better insight into critical design parameters
- Simulate test that can't be done by real lab and prototype test

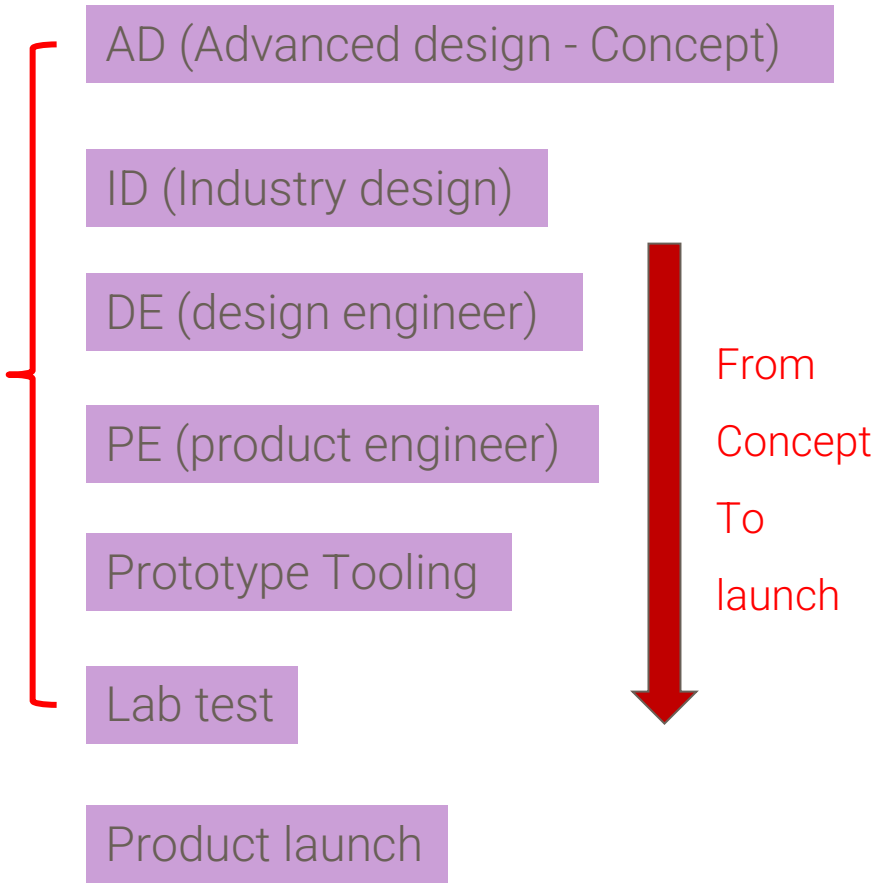
5. Our FEA Simulation Tools

- ANSYS Workbench
  - ✓ For all kinds of FEA structure stress analysis and thermal analysis.
  - ✓ Simple, medium and advanced FEA projects.
  - ✓ Linear/nonlinear, dynamic, transient thermal, etc.
- ANSYS AIM
  - ✓ For simple and medium FEA structure stress and thermal projects.
  - ✓ For simple CFD steady analysis project.
  - ✓ Easier to learn and use this tool. Good for new FEA user.
- Solidworks Simulation
  - ✓ Can run some simple and medium FEA/CFD project.
  - ✓ Useful for Solidworks user.
  - ✓ Accuracy may not be good as the ANSYS FEA.

6. Role of FEA

- From concept to product launch, FEA provides technical support for AD, ID, ED, PE, tooling and lab test.

- FEA helps engineers find potential design issues, determine right material, optimize product size, thickness, etc. Save time and cost.



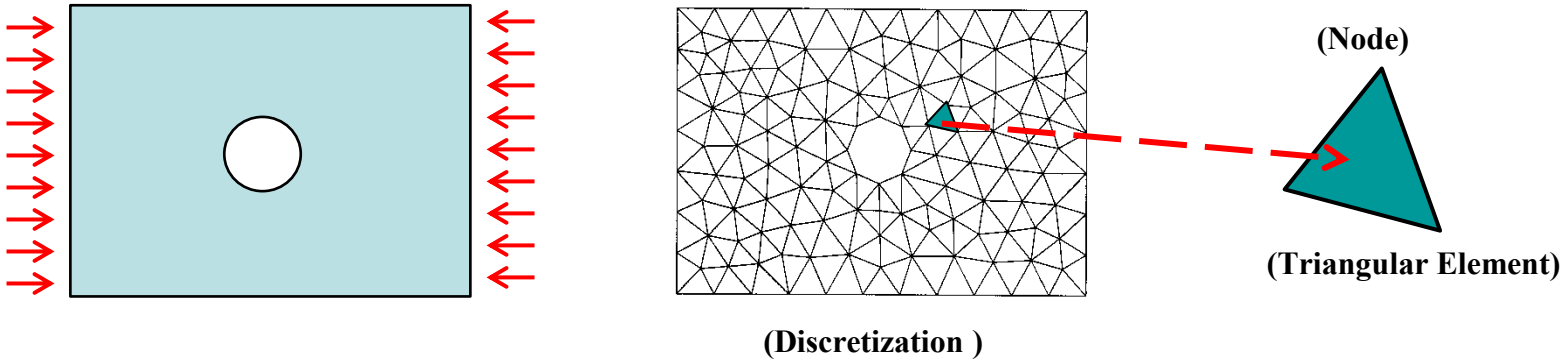
7. Learning FEA

- Training for basic analysis tools:
  - ✓ About 2 hours training, everyone could use FEA software for basic analysis
  - ✓ About 10-20% FEA projects could be done by basic trained engineers
  - ✓ Simple FEA is helpful for design/product engineers to understand and find design problem quickly

- Training for advanced analysis tools:
  - ✓ For insight into understanding FEA, people need to have advanced degree in engineering field, take 10-15 FEA related courses, and 5-10 year experience
  - ✓ Nobody could fully use FEA for all industry fields
  - ✓ Many researchers are still working to develop FEA for more accurate and faster analysis

8. Basic concept of FEA

1. Discretize the body into a finite number of element subdomains



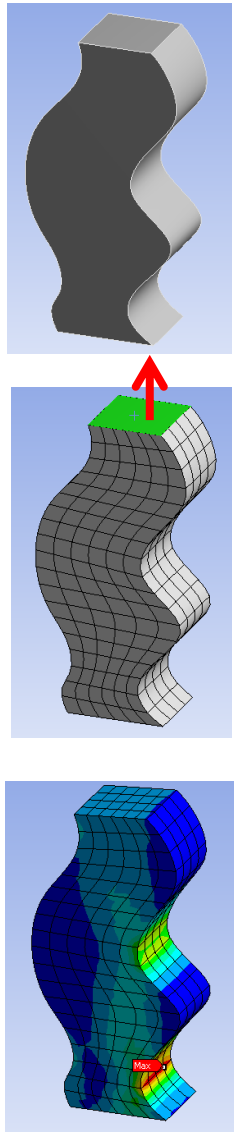
2. Generate equations over each element in term of nodal values. (partial differential and integral equations)
3. Numerically calculate above equations, obtain the result of deflection, stress, safety factor, etc.

$$\begin{cases} a_1x_1 + b_1x_2 + c_1x_3 + \cdots = F_1 \\ a_2x_1 + b_2x_2 + c_2x_3 + \cdots = F_2 \\ a_3x_1 + b_3x_2 + c_3x_3 + \cdots = F_3 \\ \cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots \\ \cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots\cdots \end{cases}$$

9. How to RUN FEA

Three main steps in FEA program:

- **Step 1: Pre-processing (Input)**
  - Geometry Model (Solidworks)
  - Material Data
- **Step 2: Run FEA**
  - Boundary Condition
  - Meshing
  - Analysis Type: Static, Dynamic, etc.
  - Solve
- **Step 3: Post-processing (Output)**
  - Review Analysis Result
  - Verification

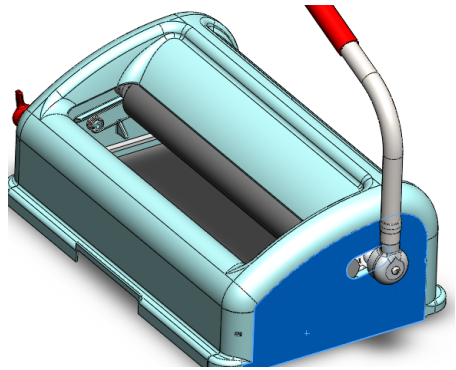


9-1. Step 1: Pre-processing

Three main steps in FEA program:

Input Geometry

- Create and simplify CAD model in CAD software:
  - ✓ Delete fillets, holes, drafts, etc. that are not structurally significant
  - ✓ Contact the interface between parts of assembly
- Import CAD model to FEA program



Input Material Data

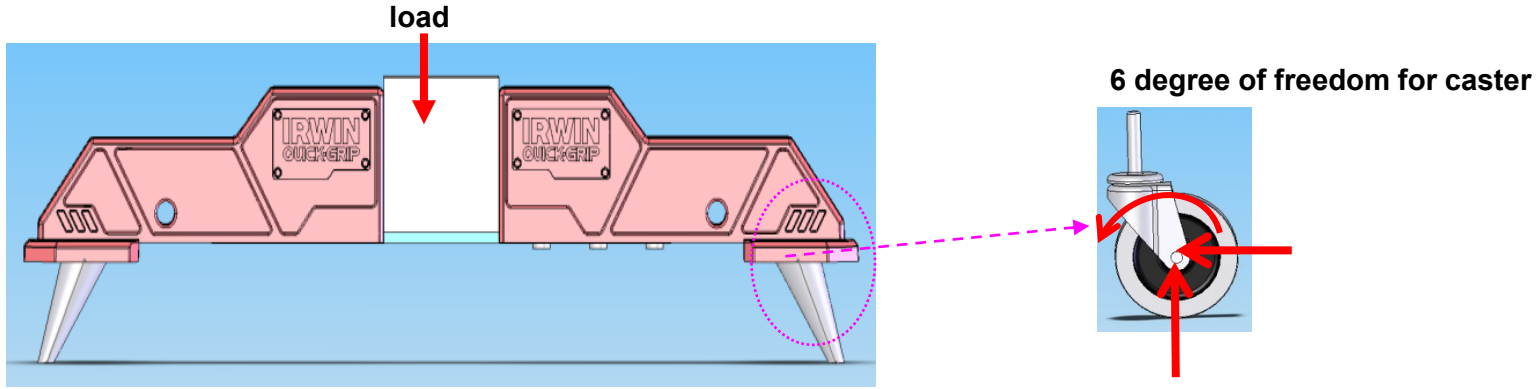
- Modulus
- Poisson's Ratio
- Yield strength
- Density …

Note: The material data is very important for FEA analysis, it is directly related to the analysis result.

9-2. Step 2: Run FEA

Boundary Condition

- Definition: the physical constraint applied to the boundary of domain
- It includes displacement, force, velocity, temperature, etc.

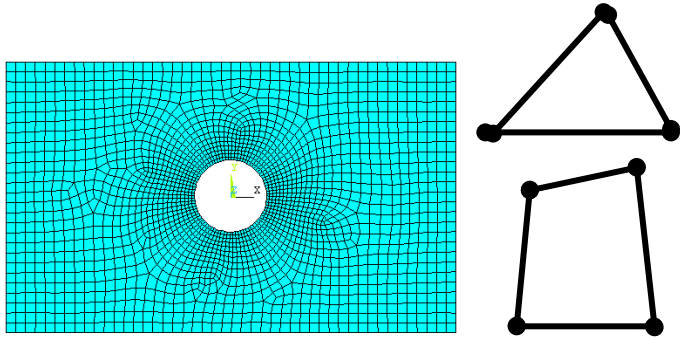


- Make sure the applied boundary condition is the same as the real condition
- Always consider the worst case for boundary conditions
- Don’t over constrain the boundary conditions

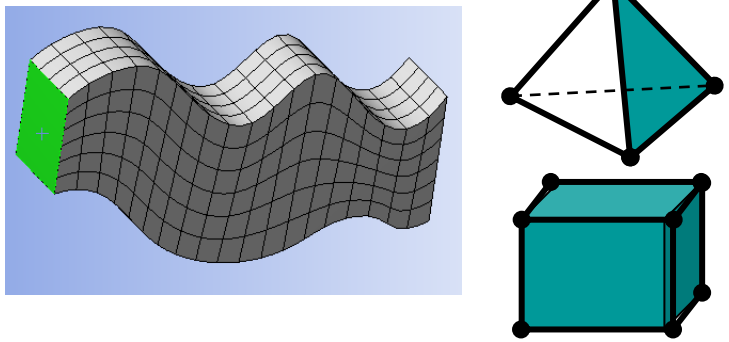
9-2. Step 2: Run FEA

Meshing

- Two-Dimensional Elements
  Triangular, Quadrilateral
  Plates, Shells, 2-D Continual



- Three-Dimensional Elements
  Tetrahedral, Rectangular Prism (Brick) 3-D Continual

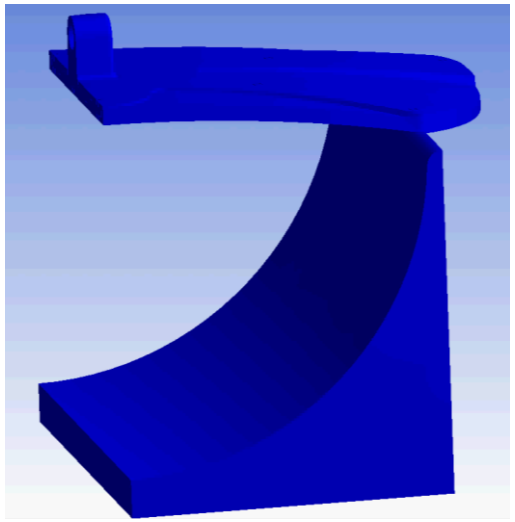


- **Note:** Different element type and shape will affect the FEA running speed and accuracy

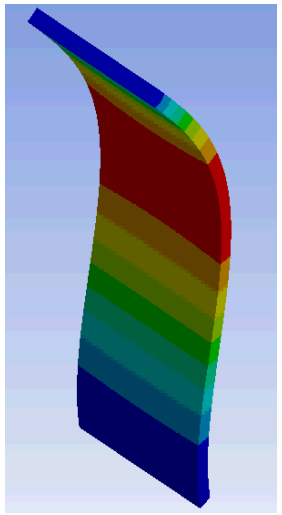
9-2. Step 2: Run FEA

Analysis type and solve

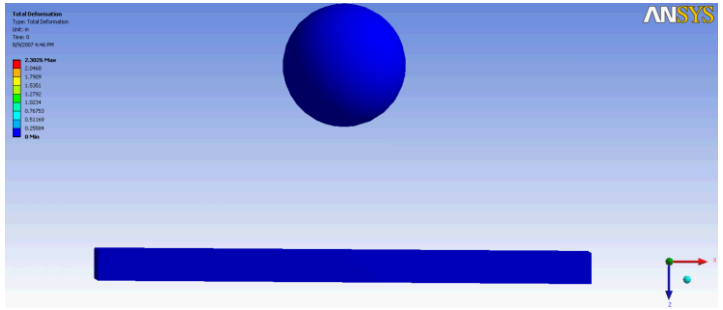
- Static analysis (linear & nonlinear)
- Buckling analysis
- Dynamic analysis (impact)
- Thermal analysis
- Fatigue analysis



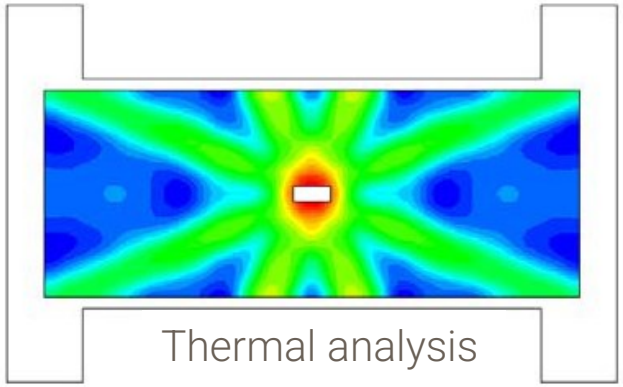
Nonlinear static analysis



Buckling analysis



Impact analysis



Thermal analysis

9-3. Step 3: Post Processing

Review analysis result:

- Review analysis results including stress, strain, deformation, safety factor and others
- Make design decisions based on the FEA results

Verification:

- Do the reaction forces balance the applied loads?
  - ✓ Compare with the results calculated by hand
- Where is the maximum stress located?
  - ✓ If it is at a singularity position, the value is generally meaningless
- How do the analysis results compare with the theory result?
- Compare FEA results to test results, if available

10. Accuracy of FEA result

- For medium complexity project, the analysis error is between 10% - 20%

	Deflection, temperature	Stress, safety factor
Analysis error	10%	20%

- Analysis errors are coming from:

Error Percentage	Error Types	Notes
10%	Material data & Geometry	1) Linear material data is simplified for faster FE analysis. 2) simplified model
5%	FEA processing	1) element type, 2) mesh density, 3) boundary conditions, 4) analysis solver, etc.
5%	Others	1) molding process method, 2) irregularities on the real product, 3) different test conditions and environmental factors, etc.

11. Analysis Time on FEA

- For medium complexity FEA project

Analysis type	Model	Time	
		Running time	Expected turnaround time
Static	One part	2 hours	< 10 hours
	Assembly	4 – 8 hours	10 – 30 hours
Dynamic	One part	10 – 20 hours	1 -1.5 week
	Assembly	20 – 40 hours	2 - 3 weeks

Note: Run time does not include pre-/post-processing, i.e., setting up problem, re-running to obtain convergence, analyzing and creating report, etc.

12. Time vs. Accuracy

- To obtain more accurate analysis results:
  - ✓ Increase mesh density
  - ✓ Accurate material data
  - ✓ Use original solid model
  - ✓ Choose appropriate solver and solver type

- To make FEA run faster:
  - ✓ Simplify analysis model
  - ✓ Optimize mesh density

13. What we need to run FEA

- Accurate 3D CAD model (Solidworks, Creo, .igs, or.stp format files)
- Accurate material data:
  - ✓ Static: modulus, Poisson's ratio, yield strength.
  - ✓ Dynamic: rated stress-strain curve, Poisson's ratio
  - ✓ Thermal: thermal expansion, thermal conductivity, modulus, Poisson's ratio, yield strength
  - ✓ Fatigue: S-N curve, modulus, Poisson's ratio, yield strength.
- Detailed load conditions.
- What is the main concern from the analysis?

14. Conclusion

- FEA can do all kinds of product performance tests before make real prototype and lab tests
- FEA can obtain accurate results on simple analysis (error < 10%)
- FEA can obtain reasonable result on complex analysis (error < 30%)
- For more accurate FEA result, need to receive:
  - ✓ Accurate CAD data
  - ✓ Accurate material data
  - ✓ Accurate boundary conditions
  - ✓ Patience

15. Our Analysis Ability

- FEAmax can perform all types of FEA simulations as below

- Static/Dynamic analysis
- Linear/Nonlinear analysis
- Buckling analysis
- Drop test
- Fatigue Stress analysis
- Steady thermal analysis
- Transient thermal analysis
- Modal analysis
- Vibration (seismic) analysis
- Harmonic Analysis
- Spectrum Analysis
- Transient Impact Analysis
- Fracture Analysis
- Electromagnetics Analysis
- Composite Analysis
- Acoustic Analysis

15. About FEAmax LLC

- Since 2005, FEAmax has provided expert engineering services from concept to production, specializing in product development using state-of-the-art CAD/CAE, FEA, CFD, Moldflow, Prototype Testing, and Manufacturing tools.
  - ✓ CAD design/drafting services.
  - ✓ Finite Element Analysis (FEA)
  - ✓ Computational Fluid Dynamic (CFD)
  - ✓ Moldflow Analysis
  - ✓ Lab test and material analysis
  - ✓ Manufacturing & Global Sourcing.