Introduction to FEA

Won Hyun Park

FEA Pitfalls

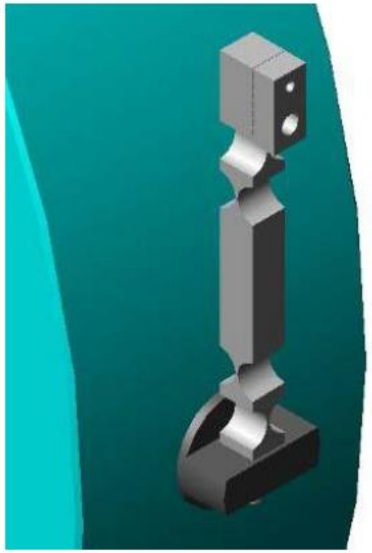
Tony Rizzo (Bell Labs) quotes:

- "Some engineers and managers look upon commercially available FEA programs as automated tools for design. In fact, nothing could be further from reality than that simplistic view of today's powerful programs. **The engineer who plunges ahead, thinking that a few clicks of the left mouse button will solve all his problems, is certain to encounter some very nasty surprises."**
- "**With the exception of a very few trivial cases, all finite element solutions are wrong**, and they are likely to be more wrong than you think. **One experienced analysis estimates that 80% of all finite element solutions are gravely wrong, because the engineers doing the analyses make serious modeling mistakes."**
- "Finite element analysis is a very powerful tool with which to design products of superior quality. Like all tools, it can be used properly, or it can be misused. **The keys to using this tool successfully are to understand the nature of the calculations that the computer is doing and to pay attention to the physics**."

FEA Theory

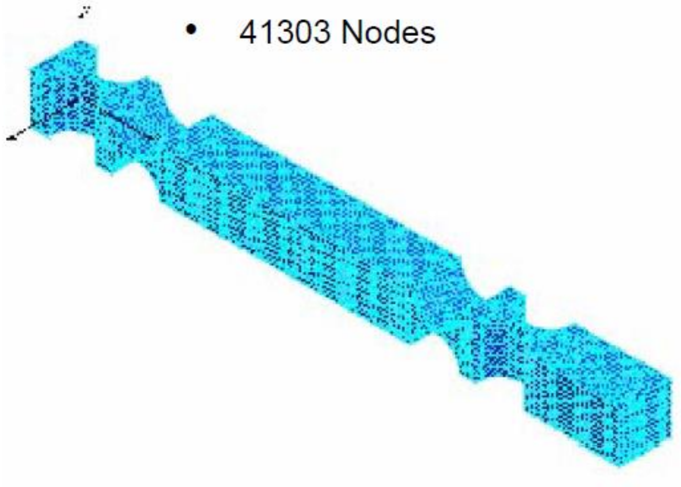
- Finite element method – numerical procedure for solving a continuum mechanics problem with acceptable accuracy.
- Subdivide a large problem into small elements connected by nodes.

Flexure Solid Model



Flexure FEA Model

- 123399 DOF
- 25795 TRETA4 Elements
- 41303 Nodes



- FEM by minimizing the total potential energy of the system to obtain primary unknowns - the temperatures, stresses, flows, or other desired

FEA example for spring

- **Equilibrium : Minimum of Potential energy**

  (Assume 1D problem : x axis)

$$\Pi = strain\ energy\ - work = \frac{1}{2}\sigma\varepsilon\ - Fx$$

$$\frac{\partial \Pi}{\partial x} = 0$$

For a spring,

$$\Pi = \frac{1}{2}kx^2\ - Fx\ , \qquad \frac{\partial \Pi}{\partial x} = kx - F = 0$$

$$\rightarrow \quad \boldsymbol{F = kx}$$

General FEA formula

The total potential energy can be expressed as:

$$\Pi = \frac{1}{2}\int_{\Omega} \sigma^T \varepsilon dV - \int_{\Omega} d^T b dV - \int_{\Gamma} d^T q dS$$

The total potential energy of the discretized individual element:

$$\Pi_e = \frac{1}{2}\int_{\Omega_e} u^T \left(B^T E B\right)^T u dV - \int_{\Omega_e} u^T N^T p dV - \int_{\Gamma} u^T N^T q dS = 0 .$$

$\frac{\partial \Pi_e}{\partial u} =$ O gives: **F= K u**, where K is stiffness Matrix, [K].

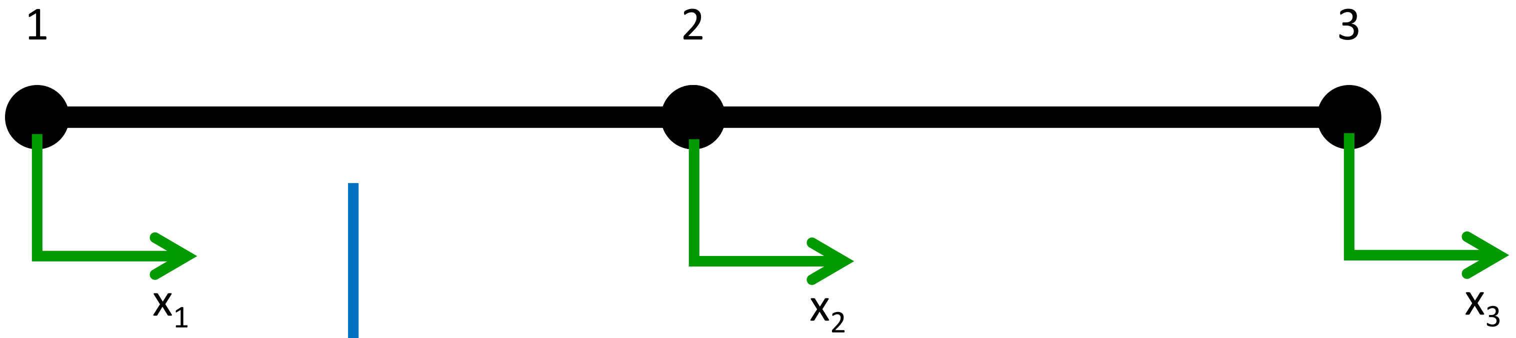
FEA Solution

- **Simple Hook's law**

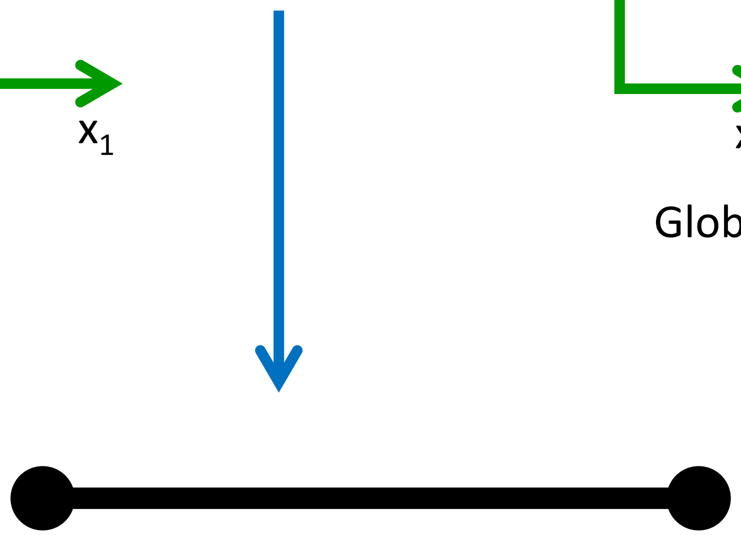
$$[F] = [K]\cdot[u]$$

$$[u] = [K]^{-1}\cdot[F]$$

System stiffness matrix : 1D example

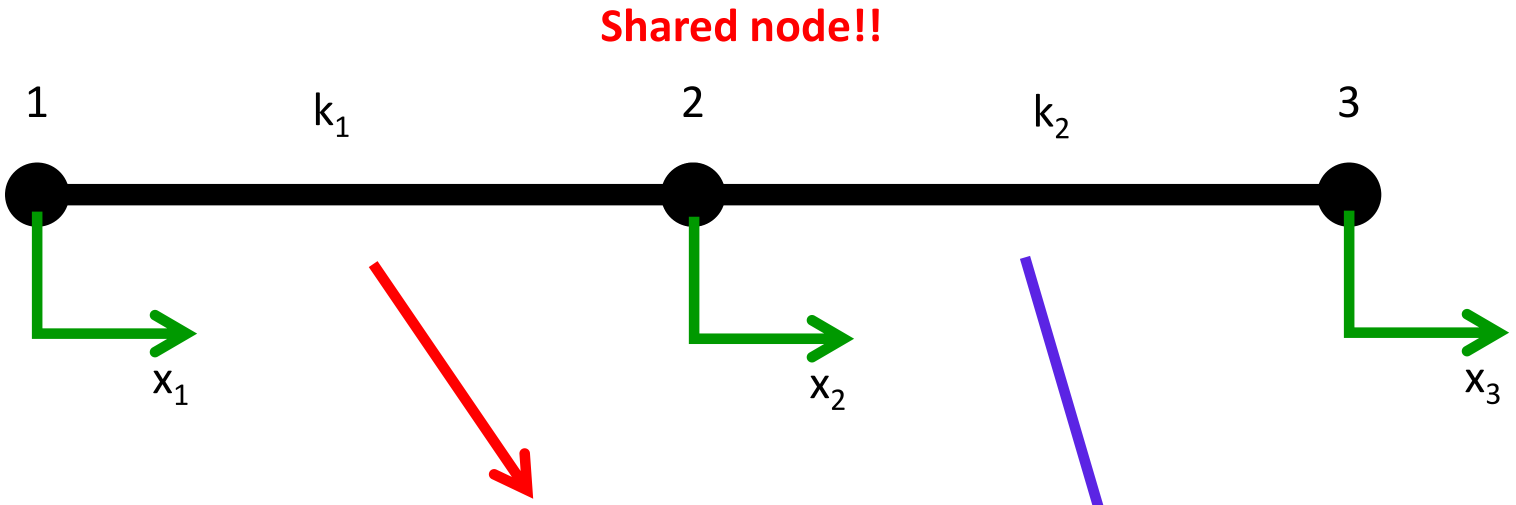


Global stiffness matrix = 3 x 3



Element stiffness matrix = 2 x 2 = $\begin{pmatrix} k & -k \\ -k & k \end{pmatrix}$

How to build the stiffness matrix

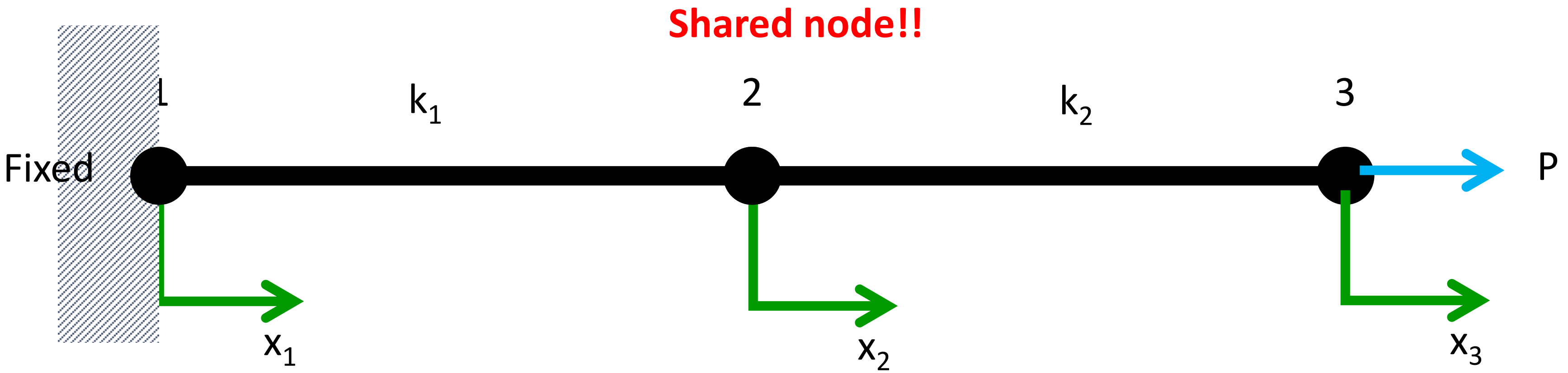


Global stiffness matrix = 3 x 3

$$\begin{pmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1 + k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{pmatrix}$$

B.C and solve

- **Boundary condition**



$$\begin{pmatrix} 0 \\ x_2 \\ x_3 \end{pmatrix} = \begin{pmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1 + k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{pmatrix}^{-1} \begin{pmatrix} 0 \\ 0 \\ P \end{pmatrix}$$

Nodes

"What are they"

A node is simply a coordinate location in space where a DOF (degree of freedom) is defined.

Nodes – Properties and Characteristics

- Infinitesimally small
- Defined with reference to a global coordinate system
- Typically nodes are defined on the surface and in the interior of the component you are modeling
- Form a grid work within component as a result of the mesh
- Typically define the corners of elements
- Where we define loads and boundary conditions
- Location of our results (deformation, stress, etc.)
- Nodes are the byproduct of defining elements

Elements

"What are they"

An element is a mathematical relation that defines how a DOF of a node relates to the next.

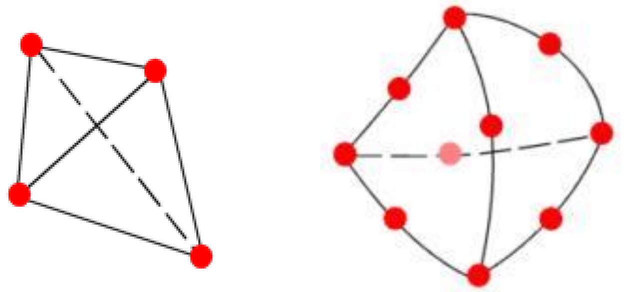
Elements – Properties and Characteristics

- Point, 2D and 3D elements
- Define a line (1D), area (2D) or volume (3D) on or within our model
- Dimensions define an "Aspect Ratio"
- A set of elements is know as the "mesh"
- Mesh shape and density is critical to the analysis
- Typically have many options that may be preset for the user
- Elements are typically what we define

SolidWorks Simulation

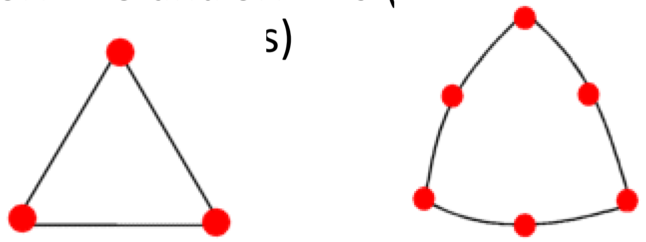
Solid Mesh

- TETRA4 and TETRA10 (4 &*10 node tetrahedron solid elements*)



Shell Mesh

- SHELL3 and SHELL6 (3 & 6 node thin
s)



Problems, Pitfalls and Tips

Requires planning and element / DOF knowledge

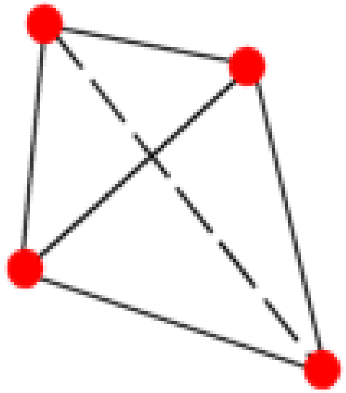
- The element defines the number of active DOFs.

TETRA4 & TETRA10 Elements

- 3 translational DOF per node
- 1 DOF per node for thermal
- TETRA4 (linear) TETRA10 (parabolic)
- Supports adaptive “P” method

SHELL3 & SHELL6 Elements

- 6 DOF per node (3 translational + 3 rotational)
- 1 DOF per node for thermal
- Membrane and bending capabilities
- Uniform thickness element
- SHELL3 (linear) SHELL6 (parabolic)
- Supports adaptive “P” method

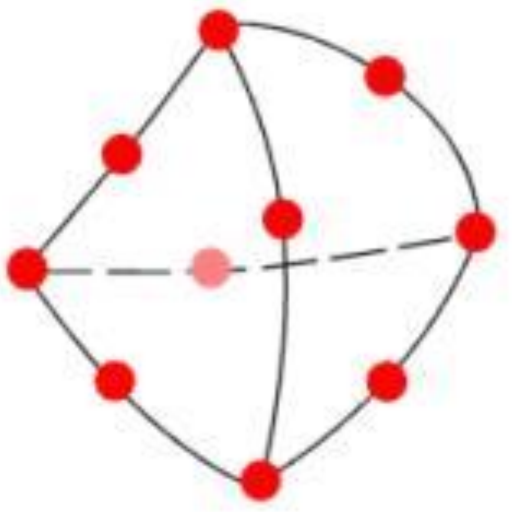


4-node tetrahedral mesh

The *unit reference tetrahedron has corners* at `{0,0,0}`, `{1,0,0}`, `{0,1,0}`, `{0,0,1}`

$$
\mathbf{K}^e = \hat{E}\begin{bmatrix}
4-6\nu & 1 & 1 & -2\hat{\nu} & -\tilde{\nu} & -\tilde{\nu} & -\tilde{\nu} & -2\nu & 0 & -\tilde{\nu} & 0 & -2\nu \\
1 & 4-6\nu & 1 & -2\nu & -\tilde{\nu} & 0 & -\tilde{\nu} & -2\hat{\nu} & -\tilde{\nu} & 0 & -\tilde{\nu} & -2\nu \\
1 & 1 & 4-6\nu & -2\nu & 0 & -\tilde{\nu} & 0 & -2\nu & -\tilde{\nu} & -\tilde{\nu} & -\tilde{\nu} & -2\hat{\nu} \\
-2\hat{\nu} & -2\nu & -2\nu & 2\hat{\nu} & 0 & 0 & 0 & 2\nu & 0 & 0 & 0 & 2\nu \\
-\tilde{\nu} & -\tilde{\nu} & 0 & 0 & \tilde{\nu} & 0 & \tilde{\nu} & 0 & 0 & 0 & 0 & 0 \\
-\tilde{\nu} & 0 & -\tilde{\nu} & 0 & 0 & \tilde{\nu} & 0 & 0 & 0 & \tilde{\nu} & 0 & 0 \\
-\tilde{\nu} & -\tilde{\nu} & 0 & 0 & \tilde{\nu} & 0 & \tilde{\nu} & 0 & 0 & 0 & 0 & 0 \\
-2\nu & -2\hat{\nu} & -2\nu & 2\nu & 0 & 0 & 0 & 2\hat{\nu} & 0 & 0 & 0 & 2\nu \\
0 & -\tilde{\nu} & -\tilde{\nu} & 0 & 0 & 0 & 0 & 0 & \tilde{\nu} & 0 & \tilde{\nu} & 0 \\
-\tilde{\nu} & 0 & -\tilde{\nu} & 0 & 0 & \tilde{\nu} & 0 & 0 & 0 & \tilde{\nu} & 0 & 0 \\
0 & -\tilde{\nu} & -\tilde{\nu} & 0 & 0 & 0 & 0 & 0 & \tilde{\nu} & 0 & \tilde{\nu} & 0 \\
-2\nu & -2\nu & -2\hat{\nu} & 2\nu & 0 & 0 & 0 & 2\nu & 0 & 0 & 0 & 2\hat{\nu}
\end{bmatrix}
$$

$\hat{E} = E/(12(1-2\nu)(1+\nu))$, $\tilde{\nu} = 1 - 2\nu$ and $\hat{\nu} = 1 - \nu$



10-node tetrahedral mesh

$$
Ke= \left(\begin{array}{ccccccccccccccc}
447 & 324 & 72 & 1 & -6 & -12 & 54 & 48 & 0 & 94 & 66 & 36 & -152 & -90 & 12 \\
324 & 1032 & 162 & 24 & -104 & -42 & 24 & 216 & 12 & 60 & 232 & 84 & -180 & -32 & 72 \\
72 & 162 & 339 & 0 & -30 & -35 & 0 & 24 & 54 & 24 & 60 & 94 & -24 & 36 & -8 \\
1 & 24 & 0 & 87 & -54 & -36 & 18 & -24 & 0 & 10 & -18 & -12 & -32 & -54 & 12 \\
-6 & -104 & -30 & -54 & 132 & 54 & -12 & 72 & 12 & 0 & 76 & 36 & 36 & 268 & 72 \\
-12 & -42 & -35 & -36 & 54 & 87 & 0 & 24 & 18 & 0 & 36 & 46 & 48 & 108 & 76 \\
54 & 24 & 0 & 18 & -12 & 0 & 108 & 0 & 0 & -36 & -12 & 0 & 72 & 12 & 0 \\
48 & 216 & 24 & -24 & 72 & 24 & 0 & 432 & 0 & -24 & -144 & -48 & 24 & 288 & 48 \\
0 & 12 & 54 & 0 & 12 & 18 & 0 & 0 & 108 & 0 & -24 & -36 & 0 & 24 & 72 \\
94 & 60 & 24 & 10 & 0 & 0 & -36 & -24 & 0 & 204 & 108 & 72 & 104 & 60 & 24 \\
66 & 232 & 60 & -18 & 76 & 36 & -12 & -144 & -24 & 108 & 492 & 216 & 48 & 308 & 96 \\
36 & 84 & 94 & -12 & 36 & 46 & 0 & -48 & -36 & 72 & 216 & 312 & 24 & 120 & 140 \\
-152 & -180 & -24 & -32 & 36 & 48 & 72 & 24 & 0 & 104 & 48 & 24 & 1416 & 648 & 144 \\
-90 & -32 & 36 & -54 & 268 & 108 & 12 & 288 & 24 & 60 & 308 & 120 & 648 & 3936 & 864 \\
12 & 72 & -8 & 12 & 72 & 76 & 0 & 48 & 72 & 24 & 96 & 140 & 144 & 864 & 1416 \\
55 & 48 & 0 & -83 & 54 & 12 & -90 & 72 & 0 & -26 & -30 & -12 & -392 & -312 & -48 \\
42 & 112 & -6 & 90 & -260 & -90 & 36 & -360 & -36 & -24 & -68 & -12 & -336 & -1424 & -96 \\
-12 & -30 & 19 & 12 & -54 & -83 & 0 & -72 & -90 & 0 & 12 & 10 & 0 & 96 & -248 \\
-311 & -180 & -24 & 19 & -18 & -12 & -198 & -144 & 0 & 58 & 54 & 36 & 232 & 456 & 144 \\
-252 & -992 & -126 & 0 & -32 & -18 & -72 & -792 & -36 & 36 & 88 & 36 & 336 & 352 & 96 \\
-24 & -90 & -275 & 0 & -18 & -17 & 0 & -72 & -198 & 24 & 36 & 58 & 96 & 192 & 232 \\
-431 & -288 & -96 & 11 & -6 & -12 & 18 & 24 & 0 & -350 & -234 & -132 & 136 & 216 & 48 \\
-306 & -1040 & -234 & 6 & -28 & -6 & 12 & 72 & -12 & -216 & -860 & -324 & 216 & 256 & -144 \\
-132 & -306 & -395 & -12 & 6 & 11 & 0 & -24 & 18 & -96 & -252 & -386 & 48 & -288 & -152 \\
95 & 84 & 24 & -59 & 18 & 12 & -18 & -48 & 0 & -98 & 18 & 12 & -680 & -648 & -240 \\
60 & 128 & 30 & 72 & -272 & -126 & -24 & -72 & -12 & -36 & -392 & -180 & -504 & -1568 & -432 \\
24 & 42 & 59 & 48 & -126 & -167 & 0 & -24 & -18 & -24 & -180 & -242 & -240 & -576 & -680 \\
148 & 84 & 24 & 28 & -12 & 0 & 72 & 72 & 0 & 40 & 0 & -24 & -704 & -288 & -96 \\
114 & 448 & 84 & -42 & 148 & 60 & 36 & 288 & 72 & 36 & 268 & 72 & -288 & -2384 & -576 \\
36 & 96 & 148 & -12 & 48 & 64 & 0 & 144 & 72 & -24 & 0 & 4 & -96 & -576 & -848
\end{array}\right.
$$

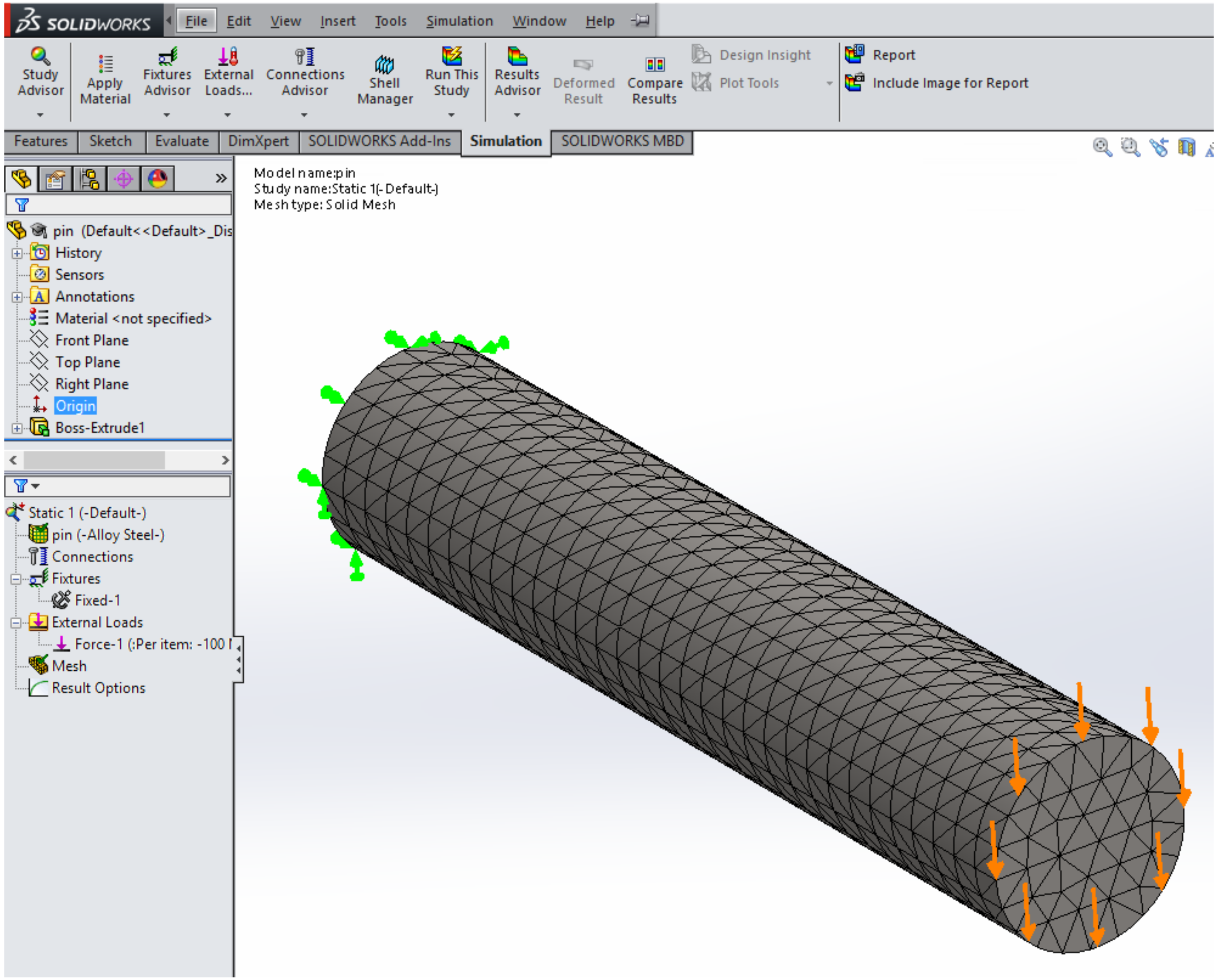
$$
\left.\begin{array}{ccccccccccccccc}
55 & 42 & -12 & -311 & -252 & -24 & -431 & -306 & -132 & 95 & 60 & 24 & 148 & 114 & 36 \\
48 & 112 & -30 & -180 & -992 & -90 & -288 & -1040 & -306 & 84 & 128 & 42 & 84 & 448 & 96 \\
0 & -6 & 19 & -24 & -126 & -275 & -96 & -234 & -395 & 24 & 30 & 59 & 24 & 84 & 148 \\
-83 & 90 & 12 & 19 & 0 & 0 & 11 & 6 & -12 & -59 & 72 & 48 & 28 & -42 & -12 \\
54 & -260 & -54 & -18 & -32 & -18 & -6 & -28 & 6 & 18 & -272 & -126 & -12 & 148 & 48 \\
12 & -90 & -83 & -12 & -18 & -17 & -12 & -6 & 11 & 12 & -126 & -167 & 0 & 60 & 64 \\
-90 & 36 & 0 & -198 & -72 & 0 & 18 & 12 & 0 & -18 & -24 & 0 & 72 & 36 & 0 \\
72 & -360 & -72 & -144 & -792 & -72 & 24 & 72 & -24 & -48 & -72 & -24 & 72 & 288 & 144 \\
0 & -36 & -90 & 0 & -36 & -198 & 0 & -12 & 18 & 0 & -12 & -18 & 0 & 72 & 72 \\
-26 & -24 & 0 & 58 & 36 & 24 & -350 & -216 & -96 & -98 & -36 & -24 & 40 & 36 & -24 \\
-30 & -68 & 12 & 54 & 88 & 36 & -234 & -860 & -252 & 18 & -392 & -180 & 0 & 268 & 0 \\
-12 & -12 & 10 & 36 & 36 & 58 & -132 & -324 & -386 & 12 & -180 & -242 & -24 & 72 & 4 \\
-392 & -336 & 0 & 232 & 336 & 96 & 136 & 216 & 48 & -680 & -504 & -240 & -704 & -288 & -96 \\
-312 & -1424 & 96 & 456 & 352 & 192 & 216 & 256 & -288 & -648 & -1568 & -576 & -288 & -2384 & -576 \\
-48 & -96 & -248 & 144 & 96 & 232 & 48 & -144 & -152 & -240 & -432 & -680 & -96 & -576 & -848 \\
376 & 0 & -96 & -152 & -192 & 0 & -116 & -72 & 48 & 292 & 144 & 0 & 136 & 288 & 96 \\
0 & 928 & 0 & -96 & 256 & 96 & -72 & -176 & 72 & 216 & 736 & 216 & 144 & 256 & -144 \\
-96 & 0 & 376 & 96 & 192 & 136 & 48 & 72 & -116 & -48 & 72 & 148 & 0 & -288 & -152 \\
-152 & -96 & 96 & 1048 & 576 & 192 & 292 & 168 & -48 & -308 & -144 & -96 & -680 & -672 & -288 \\
-192 & 256 & 192 & 576 & 2176 & 288 & 192 & 736 & 168 & -144 & -224 & -72 & -480 & -1568 & -528 \\
0 & 96 & 136 & 192 & 288 & 760 & 0 & 120 & 148 & -96 & -72 & -164 & -192 & -480 & -680 \\
-116 & -72 & 48 & 292 & 192 & 0 & 984 & 648 & 144 & -152 & -72 & 48 & -392 & -408 & -48 \\
-72 & -176 & 72 & 168 & 736 & 120 & 648 & 2208 & 432 & -216 & 256 & 144 & -240 & -1424 & -48 \\
48 & 72 & -116 & -48 & 168 & 148 & 144 & 432 & 984 & 48 & 144 & 136 & 0 & 48 & -248 \\
292 & 216 & -48 & -308 & -144 & -96 & -152 & -216 & 48 & 696 & 216 & 144 & 232 & 504 & 144 \\
144 & 736 & 72 & -144 & -224 & -72 & -72 & 256 & 144 & 216 & 1056 & 432 & 288 & 352 & 144 \\
0 & 216 & 148 & -96 & -72 & -164 & 48 & 144 & 136 & 144 & 432 & 696 & 96 & 144 & 232 \\
136 & 144 & 0 & -680 & -480 & -192 & -392 & -240 & 0 & 232 & 288 & 96 & 1120 & 432 & 192 \\
288 & 256 & -288 & -672 & -1568 & -480 & -408 & -1424 & 48 & 504 & 352 & 144 & 432 & 3616 & 864 \\
96 & -144 & -152 & -288 & -528 & -680 & -48 & -48 & -248 & 144 & 144 & 232 & 192 & 864 & 1408
\end{array}\right)
$$

eigs of Ke = {8809.45, 4936.01, 2880.56, 2491.66, 2004.85, 1632.49, 1264.32, 1212.42, 817.907, 745.755, 651.034, 517.441, 255.100, 210.955, 195.832, 104.008, 72.7562, 64.4376, 53.8515, 23.8417, 16.6354, 9.54682, 6.93361, 2.22099, 0, 0, 0, 0, 0, 0}

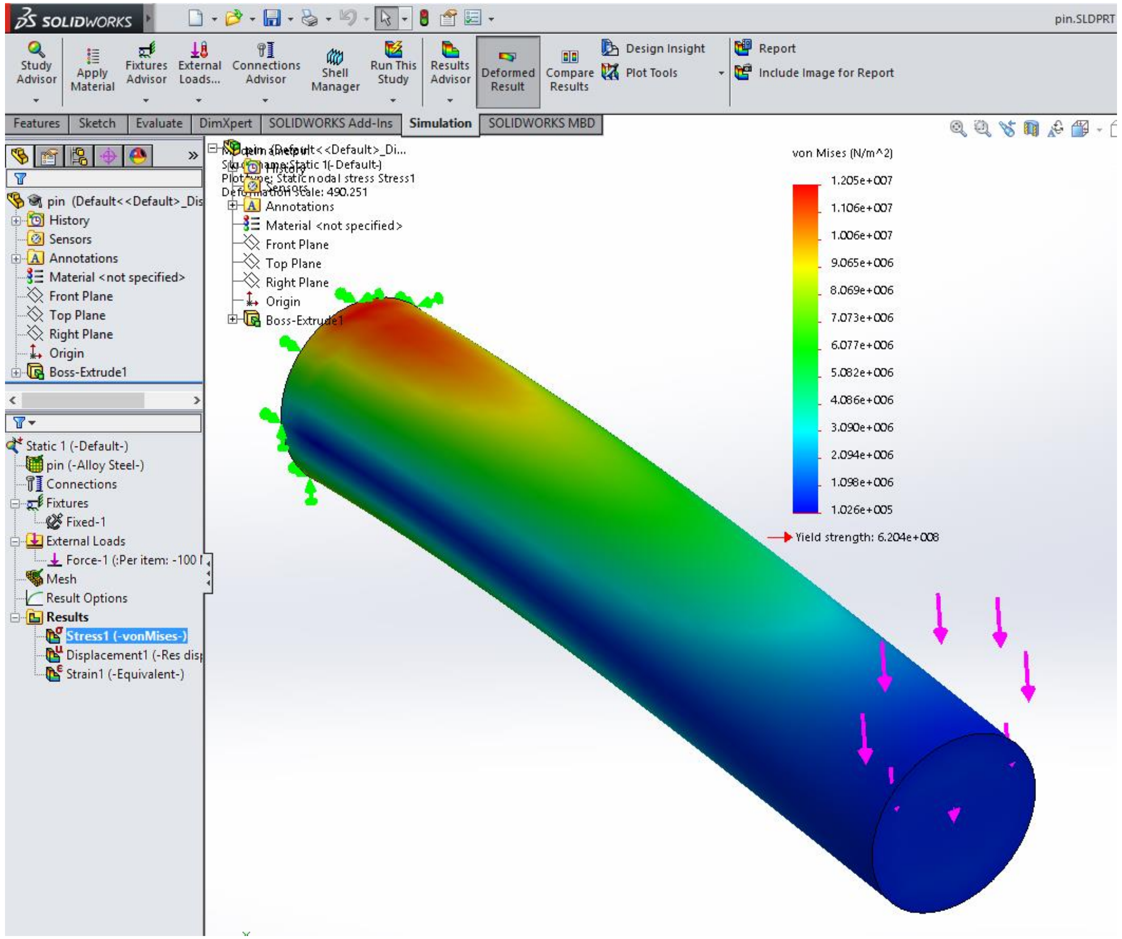
FEA procedure

- **1. Identify the problem, sketch the structure and loads.**
- **2. Create the geometry with the FE package solid modeler or a CAD system.**
- **3. Apply material properties.**
- **4. Mesh the model.**
- **5. Apply boundary conditions (constraints and loads) on the model.**
- **6. Solve numerical equations.**
- **7. Evaluate the results.**

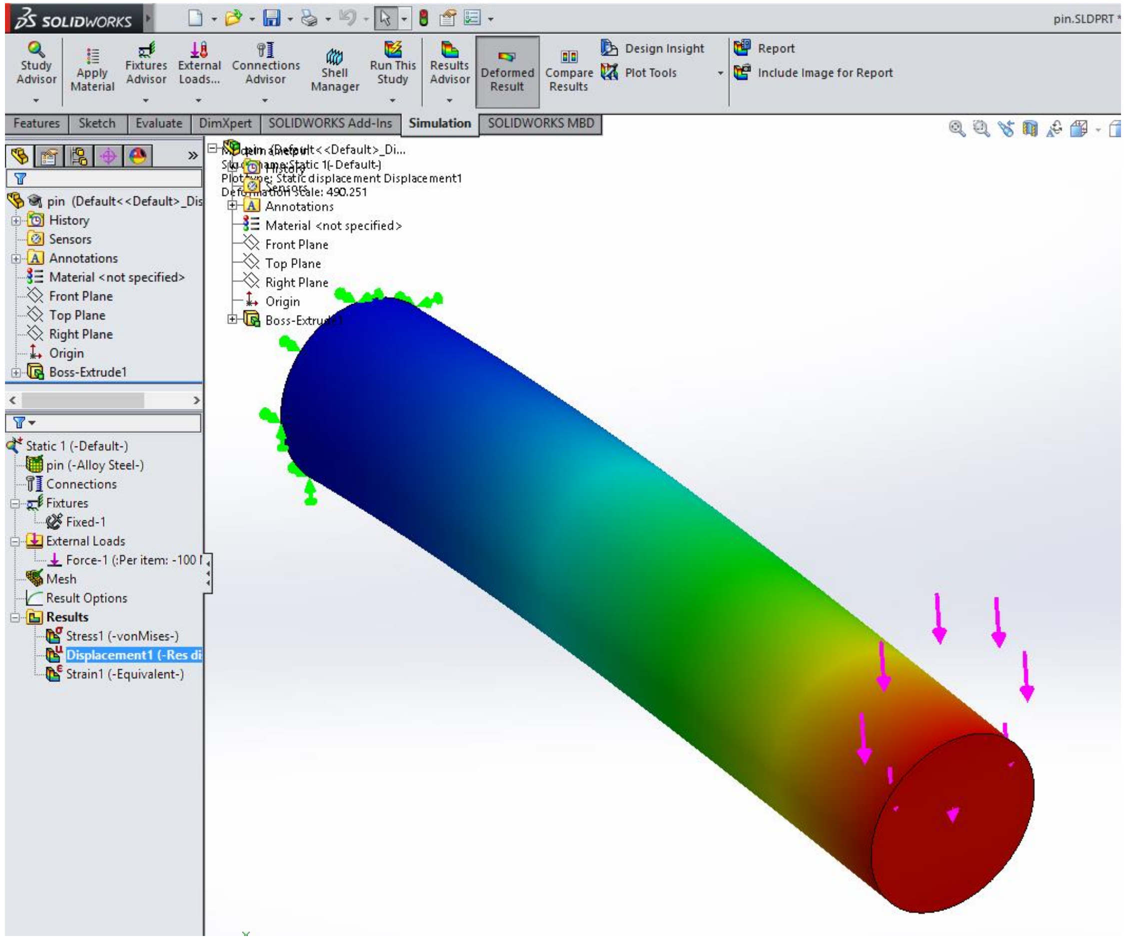
Solidworks Simulation example



Solidworks Simulation example



Stress distribution



Displacement plot

Theoretical background for modal analysis

- **Equation of motion (assume zero damping)**

$$[M]\{\ddot{u}\} + [K][u] = 0$$

- **Solving the equation**

$$\{u\} = \{\phi\}\sin\omega t$$

$$-\omega^2[M]\{\phi\}\sin\omega t + [K]\{\phi\}\sin\omega t = 0$$

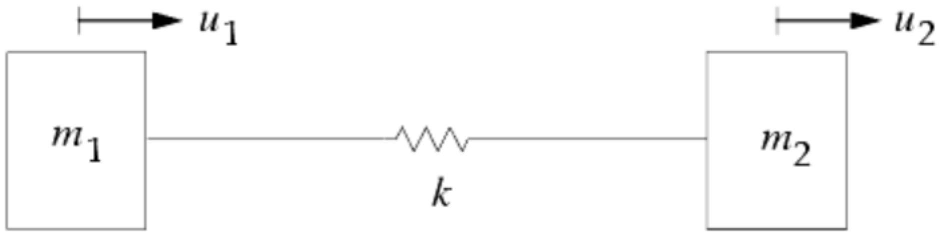
$$([K] - \omega^2[M])\{\phi\} = 0 \quad \longleftrightarrow \quad [A - \lambda I]x = 0$$

Eigen value problem

$$\boxed{\det([K] - \lambda[M]) = 0}$$

Simple analytic model

❖ Two mass block connected with a spring



$$[K] = \begin{pmatrix} k & -k \\ -k & k \end{pmatrix} \qquad [M] = \begin{pmatrix} m_1 & 0 \\ 0 & m_2 \end{pmatrix}$$

$$[K] - \varpi^2 [M] = \begin{pmatrix} k - \lambda m_1 & -k \\ -k & k - \lambda m_2 \end{pmatrix}$$

$$\det([K] - \lambda [M]) = \lambda^2 m_1 m_2 - \lambda k m_2 - \lambda k m_1 \quad \text{where} \quad \lambda = \varpi^2$$

Eigen values

$$\left\{\cancel{0}, \frac{k m_1 + k m_2}{m_1 m_2}\right\} \quad \text{if } m_1 \neq 0 \wedge m_2 \neq 0$$

Resonant frequency

Eigen functions

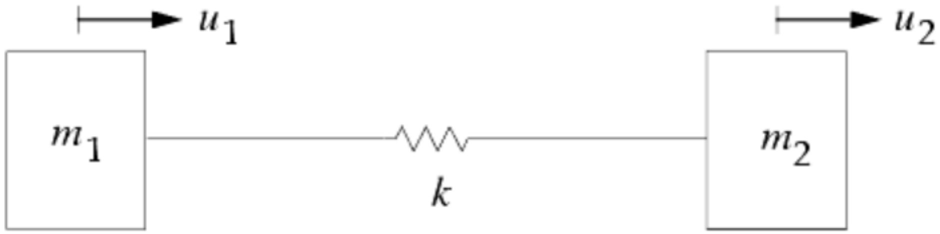
$$\left[\cancel{\begin{pmatrix} 1 \\ 1 \end{pmatrix}}\right] \quad \text{and} \quad \left[\begin{pmatrix} -\frac{m_2}{m_1} \\ 1 \end{pmatrix}\right]$$

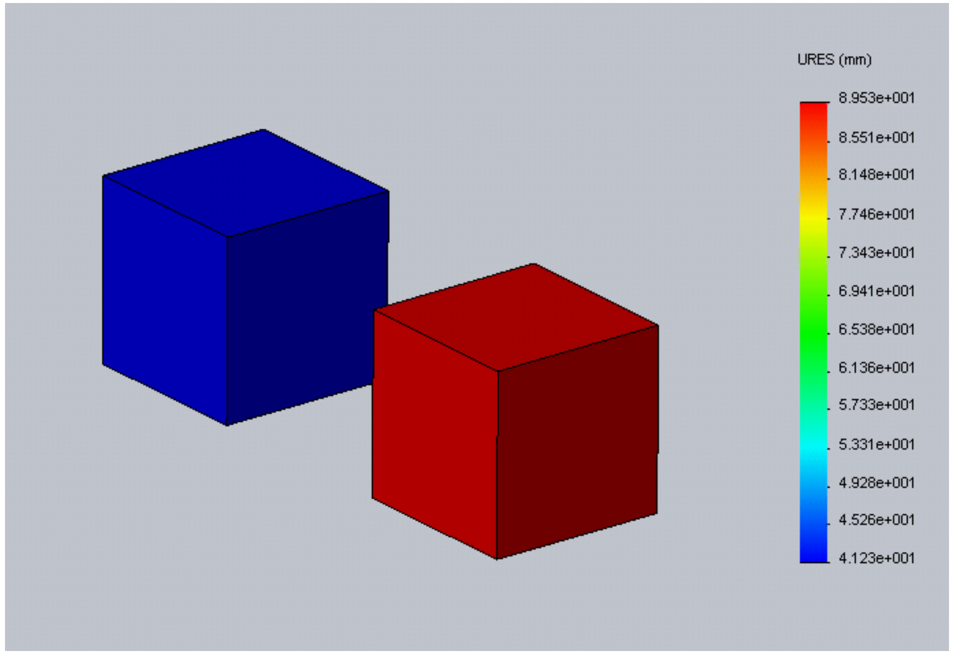
Rigid body motion

Mode shape

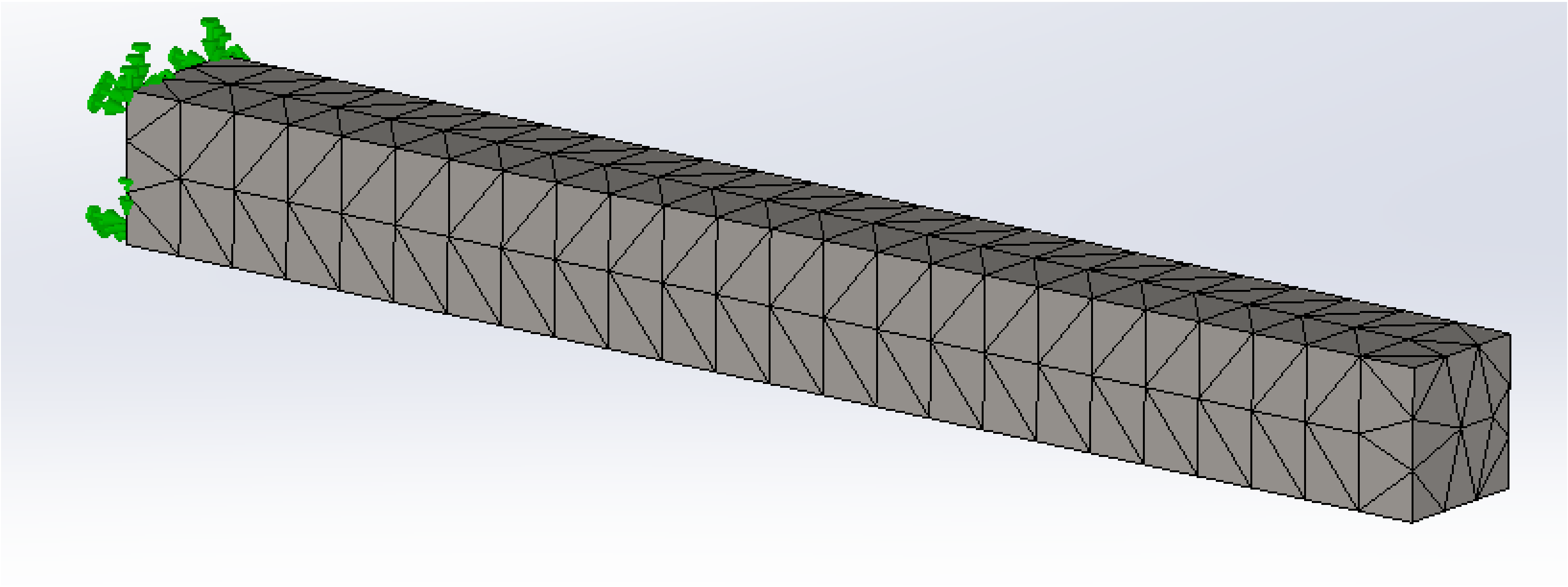
Simple analytic model

- **Two mass block connected with a spring**

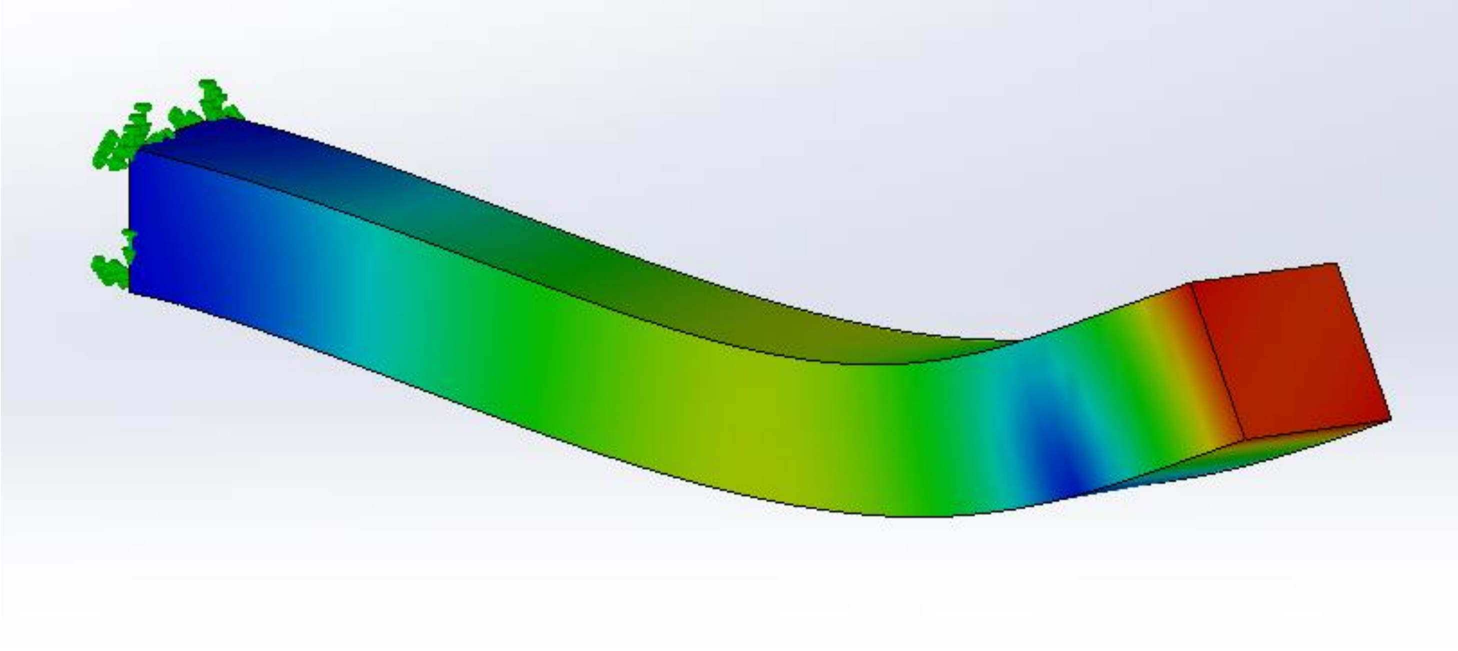




Target model – Modal frequency analysis



Target model – Modal frequency analysis



4th order mode shape for example

List Modes

Study name: Frequency 1

Mode No.	Frequency(Rad/sec)	Frequency(Hertz)	Period(Seconds)
1	2080.9	331.19	0.0030194
2	2081	331.2	0.0030194
3	12469	1984.5	0.00050391
4	12472	1985	0.00050378
5	17866	2843.5	0.00035168

Close Save Help

Modeling Tips

- **Tip #1 Understand the physics of the problem and always start with a sketch. Before you model have a plan and know:**
  - What you are going to model and what results do you need?
  - How you are going to develop your model?
  - How you are going to support the model and apply loads?
- **Tip #2 Start simple and increase complexity as required.**
  - When modeling systems – start with a single optical element.
- **Tip #3 Build simple test models for understanding.**
  - Check for load and boundary condition accuracy
    - Above depends on the type of analysis you are running
  - Check for mesh accuracy – do convergence studies
- **Tip #4 Always request reaction forces in the output.**
  - For models with both structural and gravity loads, turn off gravity and check your reaction forces. Do they match the applied load?
  - Turn on gravity. Is the increase in reaction force consistent with the gravity load? Is the direction correct?

Modeling Tips

- **Tip #5 Understand your constraints and use care not to over constrain the model.**
  - Are all six (6) rigid body translations and rotations accounted for?
  - A model with too few constraints causes a singular stiffness matrix.
  - An over constrained model creates alternate load paths.
  - When in doubt, release constraints and add soft springs.
- **Tip #6 Study the deformed shape. Does it look correct?**
  - Properly modeled, symmetric loads and constraints will produce symmetric results.
  - Always generate a symmetric mesh for optical surfaces. Automatic mesh generators rarely produce a symmetric mesh.
- **Tip #7 As a starting point, there must be enough elements to accurately predict the deformed shape.**
  - Use a minimum of 4 elements through the height or thickness or sections subjected to bending.
  - Do simple convergence studies to determine an acceptable mesh density.
- **Tip #8 An accurate stress analysis requires more elements than an accurate displacement analysis.**
  - Increase the mesh density for accurate stress analysis.
  - Check nodal stress in surrounding elements sharing a common node.

Modeling Tips

- **Tip #9 Check, check and recheck your model and results.**
  - Do hand calculations and back of the envelope calculations to verify results.
  - Assume the results are wrong until proven correct.
- **Tip #10 When in trouble – get help.**
  - Consult a senior analyst for help and tips