

((CSWA - SM))

((Section 1))

- * testing a product for safety and reliability is essential for sound product development cycle.
- * Simulation tools gives you the power to test your product virtually in no time before attempting to even prototype it.

↳ ensures better designs at a lower cost.

↳ creating a physical prototype for the purpose of testing can be costly and time consuming.

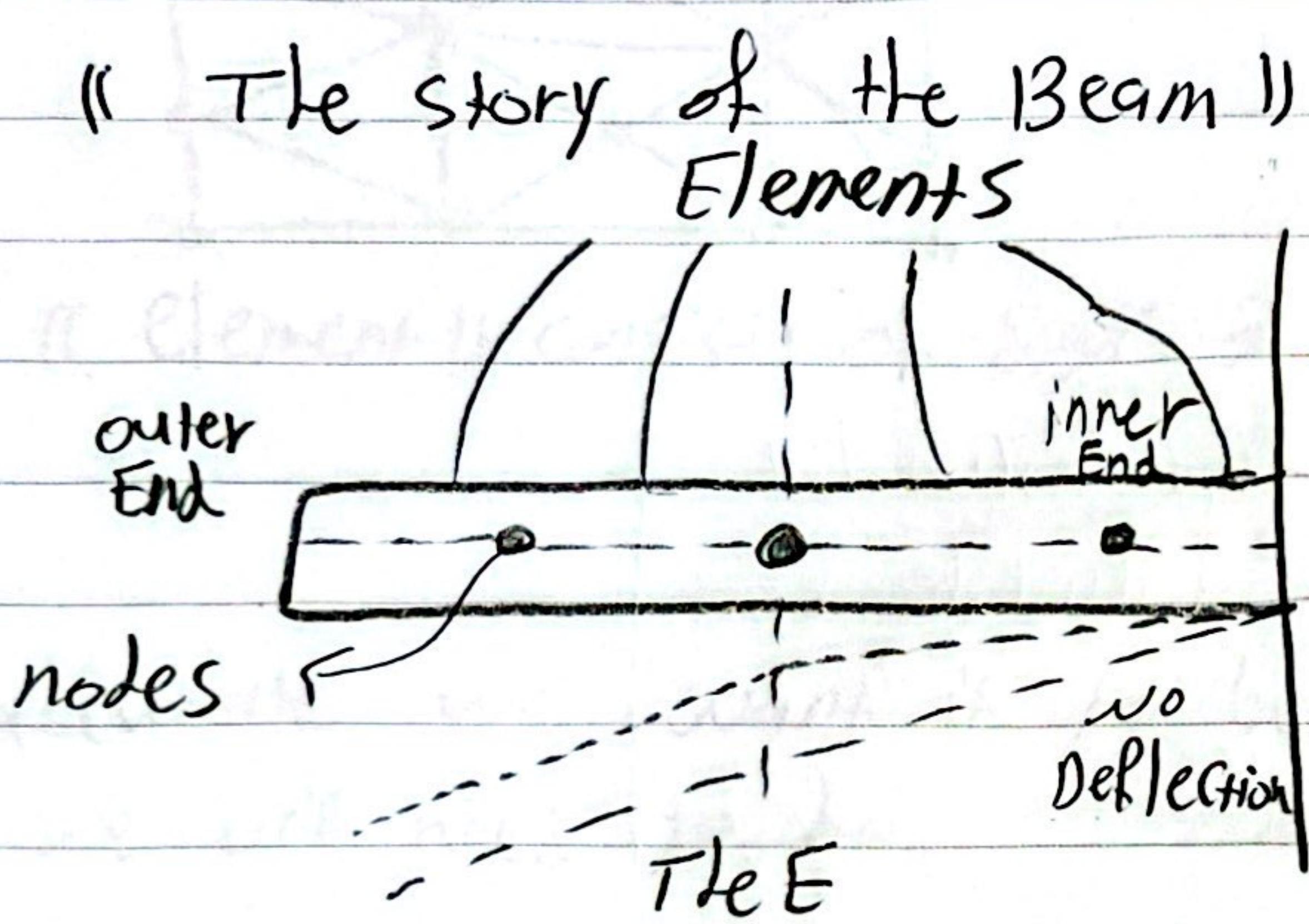
elaborate (adj) → very complicated and detailed, carefully prepared and organized.

ex :- elaborate designs.

advantages of simulation:

1. validate your design for safety and reliability before making physical prototype.
2. speed up product development cycle ((By testing many variations of your design)).
3. Design better and more optimized products.

- * This course will take you from knowing nothing to somewhere between The Associate and professional level.
- * it will cover only using tools for static analysis.
- * solidworks simulation tools use the FEA [finite element analysis model for its computation]



what if we want to know The deflection at the middle of the Beam? split The Beam in half and recalculate

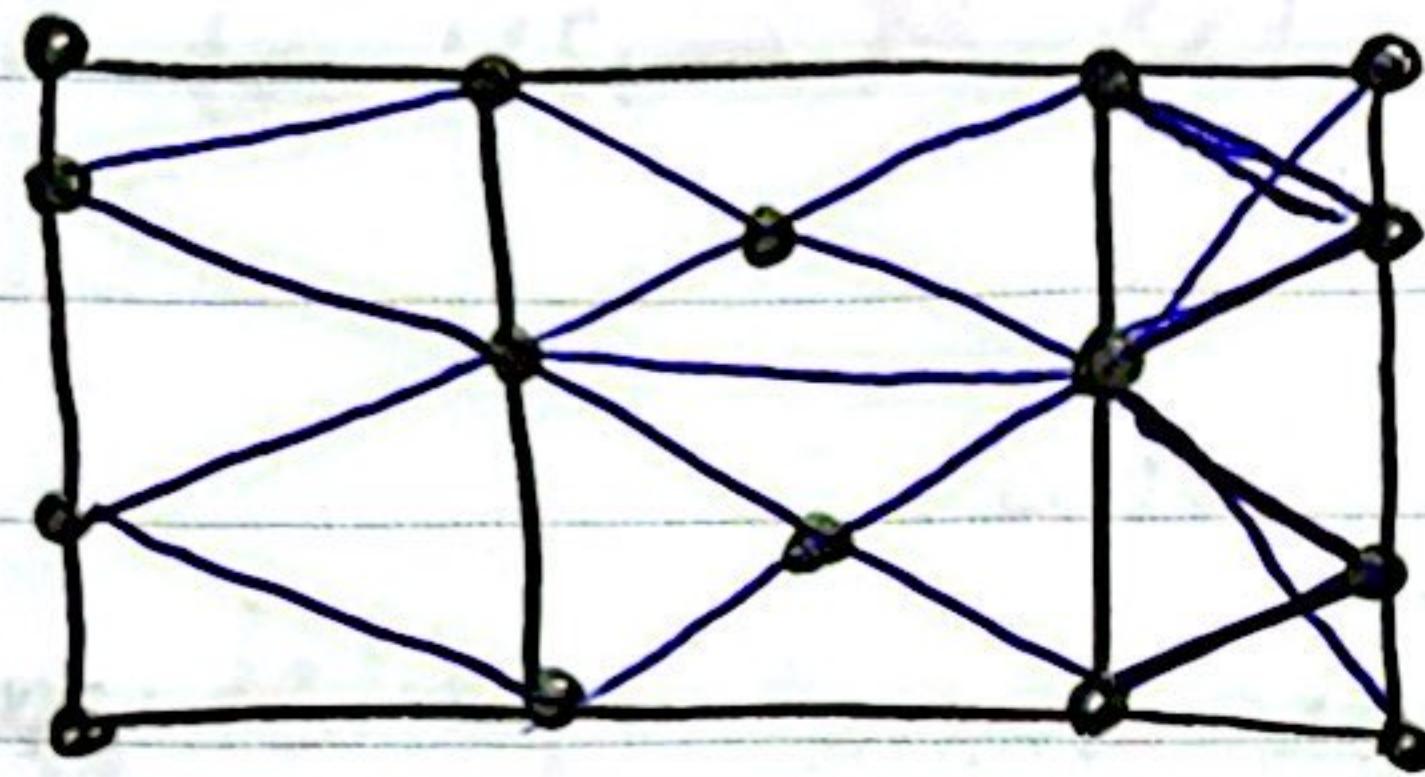
* we can split the beams as many times as required.

So what is FEA!? it's dividing a structure into smaller finite elements, which we can analyze one at a time putting all those elements again together would give us an understanding of the whole structure.

What about the two dimensions ((shells , sheets)) !?

- * anything that is relatively thin of uniform thickness can be simplified into a shell like a piece of paper.

- * we will split the whole sheet into many nodes and lines
we will switch the elements from — to Δ

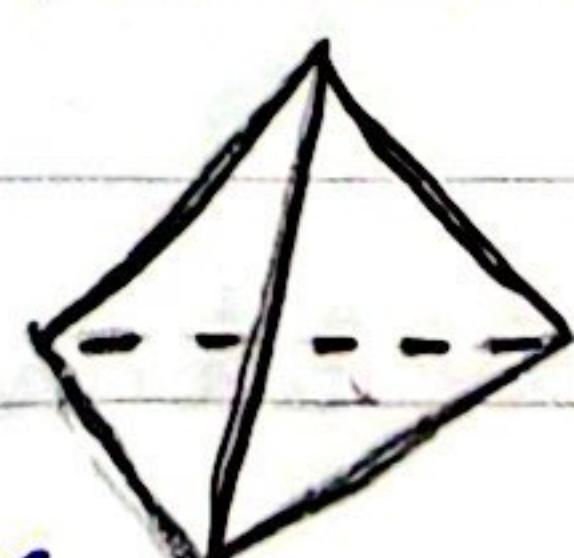


- * each triangle ((element)) consists of ~~fewer~~ 3 lines and 3 nodes.

- * the more accurate we want it to be the more calculation we will need to do.

What about 3 dimensional objects such as turbines, gears etc---? same as 2D but our element shape will have to be three dimensional such as tetrahedral..

Notes- with one location of the design known to us we can use the equation to solve what is happening



- * all the previous explanation also applies to stress distribution , factor of safety distribution etc---

↳ 12

|| Section 2 ||

* to use solidwork - simulation we will have to add the add-in first.

tools → Add-Ins → add solidwork simulation → apply

two new tabs will appear :- 1- simulation which we are going to use to set and run our simulation studies

2- analysis preparation

contains a set of modeling commands that are often used to prepare a model for simulation study.

|| The simulation process ||

steps for static studies:-

- 1- Have our 3D model prepared for simulations.
- 2- Assign the structural material (different materials have different mechanical properties)
- 3- Start the simulation study
- 4- Apply the contact conditions (if we have more than one part or more than one body)
- 5- apply fixtures (it's important that the model have certain restrictions on its degree of freedom)
- 6- apply external loads (force, pressure, Torque)
- 7- Generate The mesh (nodes The software will use as a base to calculate The stresses movements and other calculations)
- 8- run the simulation. 9- View and interpret the results

Type of stress

$$\uparrow \rightarrow \sigma = F/A$$

* Von Mises (Stress) $\rightarrow N/m^2$

↳ Show the maximum stress we expect the design to be subjected to.

* at the bottom you will see an indication of yield strength.

if yield strength > max stress

There is a good chance our object won't fail.

displacement

\uparrow Resultant

2- Displacement - URES:-

* it will be as default in mm

* you will notice that it's exaggerating as you will see Deformation scale (ex: 113) which means this displacement is scaled up by a factor of 113.

3- Factor of safety :

* you will need to focus on minimum factor of safety.

(The three types of models)

1 - Beams 2. Shells 3- methods/solids

↳ These types are different based on how the models are structured and how elements are formed.

(Section 3)

(Factor of safety)

* Factor of safety is The ratio between (The maximum load possible for a structure, The maximum intended load for the same structure)

if $FoS < 1 \rightarrow$ good likelihood That our design will fail

$FoS = 1 \rightarrow$ we ate at the edge, The design will likely fail in a short time.

$FoS > 1 \rightarrow$ The design will not fail, at least when all The conditions are perfectly meet.

Few notes to keep in mind :

- 1- The software gives approximation not exact real values , leave some space for margins of error for example by increasing FoS by a little bit.
- 2- The FoS you will get on software will not count for the age of The used materials , which can naturally make The material weaker over time.
- 3- Whenever we do something on the software we often deal with perfectly controlled situation , which may not be The cas in real life

((FOS in design))

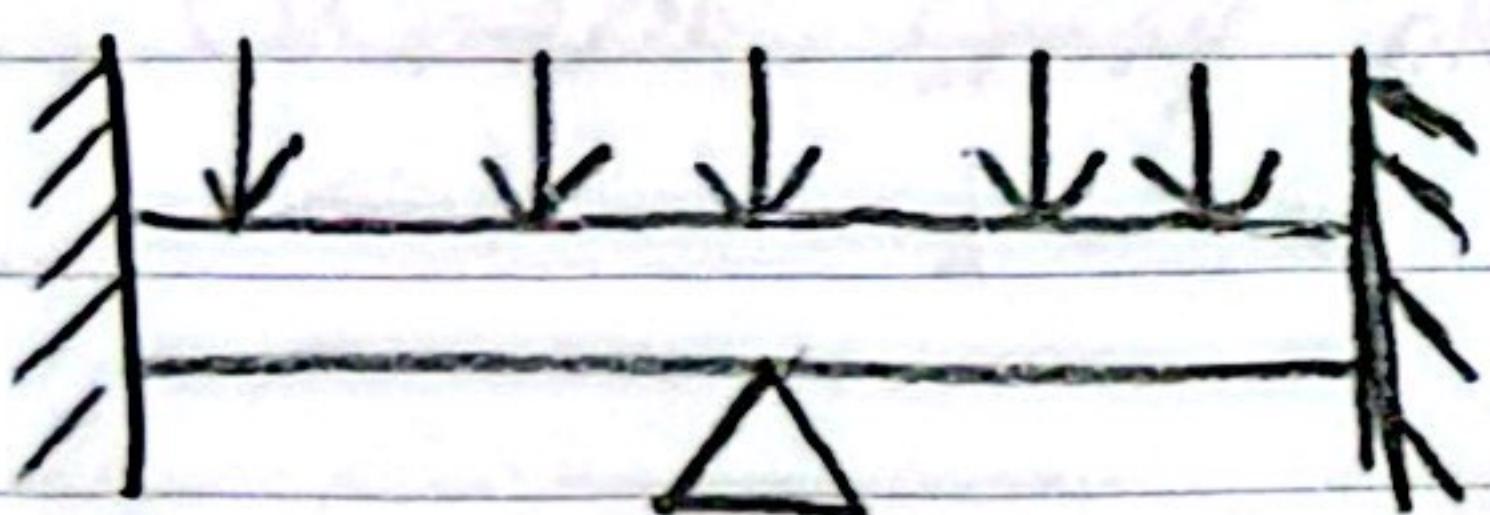
- select The aimed FOS ex: $FOS = 2$
- start Designing based on this Fos, you can add thickness or make sure you are not using too much materials for no reason

By The way How Can you determine FOS!?

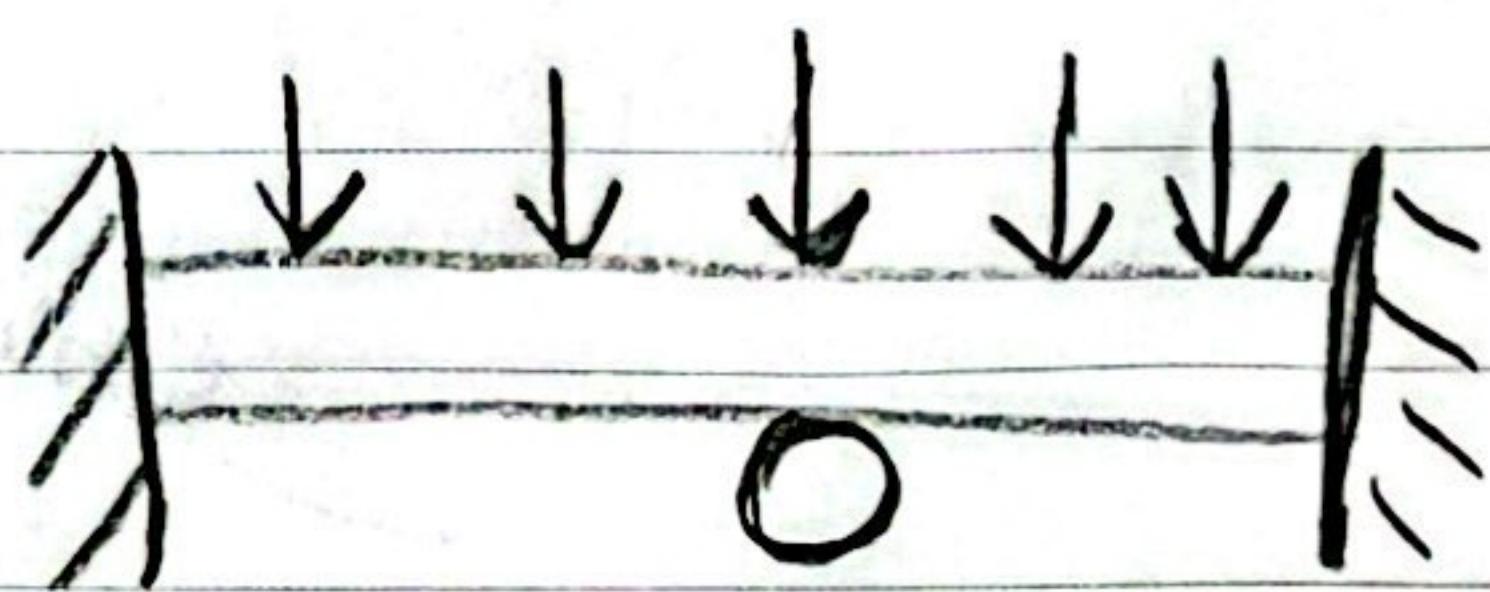
There is no magical number that can work for everything, if you did not find standards in your company, it will be an open ended standard that you would have to establish.

((Fixed VS immovable))

Fixed



Immovable



both of them will give us a fixed geometry

* all translational rotational movement are zero | * rotational movement is not zero, we will have a slope for that area.

* Apply to beams, solids, shells | * Apply to beams and shells only

(i) Default Unit System)

- * depends on the country or company you are working in
- * based on your preferences you can change it from simulation → options → default options

(ii) Default Mesh Setting)

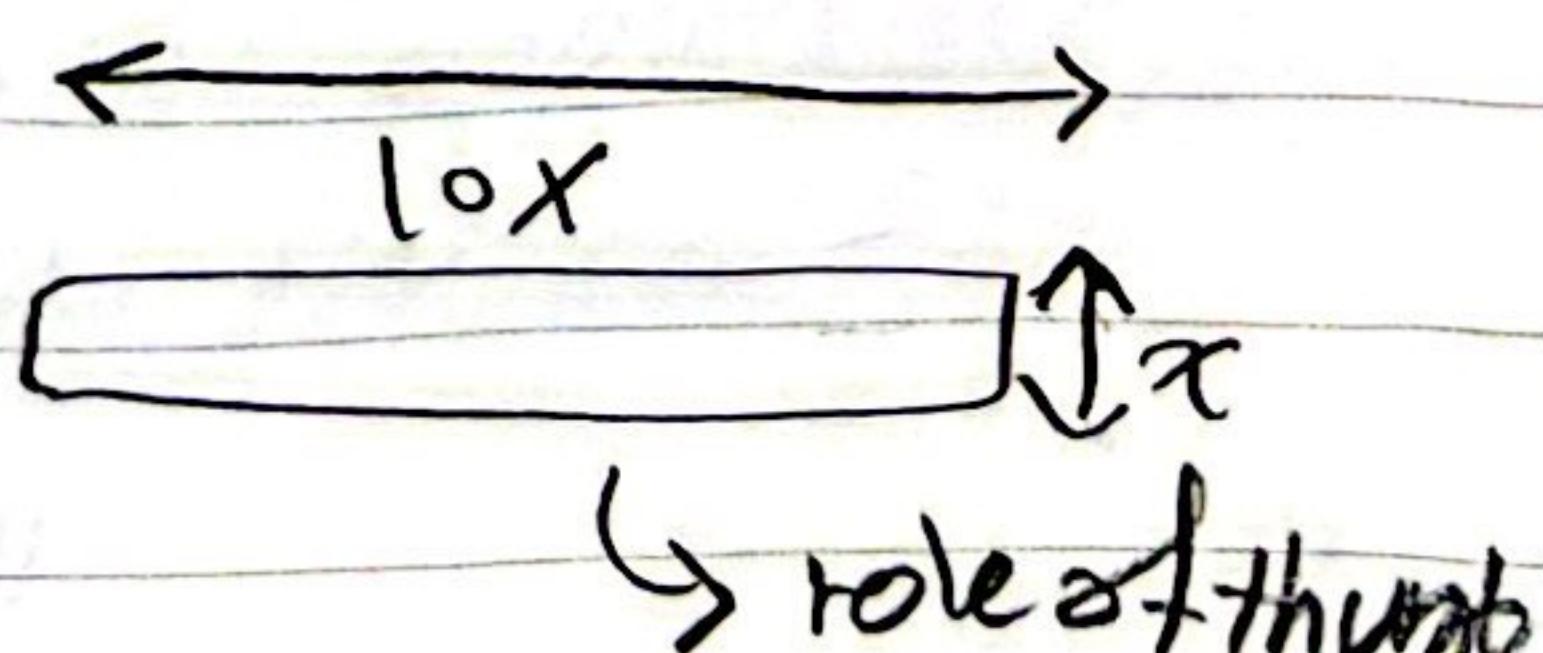
- * different mesh setting will produce different results.
- * to change mesh setting (simulation Tab → options → default options → mesh---) or from (mesh → right click → create mesh → optimize mesh parameters) → override the default meshing options for the current study only.

(iii) Section (i))

What is a beam? → a beam is looked at and interpreted as a single one dimensional element, it's generally long and has a uniform cross section.

When to treat a structure as a Beam?

1. Models made with Solid Works Workbench tools
2. Model that resemble a beam



Beam Case 1

1. length: 2 m
2. material: steel AISI 1020
3. fixtures: fixed on both sides
4. load: 2000 N

You will answer:

- * How will the beam react? (stress, displacement)
- * produce graphics for the stress distribution and displacement?
- * produce an animation showing how the beam reacts to the applied force.

Stages to solve the case:

1. Setup our study and run it.
2. Save our results as pictures.
3. Animate the simulation.

((Setting up and solving))

* apply steps for static studies

- * note: under cut list ((name - material and beam symbol))
- * note ● are beam joints at which we are going to apply fixtures.
- * Note: when choosing force you choosed Beams, direction is front plane and as a result the force normal to it.

note:- When Generating The mesh The shape of The beam might look different, This is just solidworks trying to simplify The model to save on some processing power and because it's 1-D beam.

note:- you can revert back to the original beam shape by (right click on mesh → Create quality plot → render beam shape)

note:- when you see The result you will see The beam in its simplified form you can change That by ((double click on the plot → render beam profile)) or from ((Edit definition) and it's only available at The stress plot).

points about The Beam's rendered View!?

- 1- The mesh, elements and nodes are in a line form
↳ The best way to look at values is by looking at one cross-sectional segment at a time,
 - 2- Beams have a uniform section by nature
↳ if our design is flexible you better treat it as solid bodies
- note:- Because you will find exaggerated results in the displacement result ((may not be appropriate for the purpose of showing How The Beam will react to the applied forces)) you can change it ((right click on the displacement → edit definitions → True scale))

((Saving images))

- * right click \rightarrow save as \rightarrow image type
- * the image includes ((part, legend, information box))
- * it's like taking a screen shot of the canvas

((Animation))

right click on stress result \rightarrow animate \rightarrow save as an AVI
 The angle as the canvas angle \leftarrow You have to run the animation again \leftarrow
 There are 3 variables that we can control during animation :

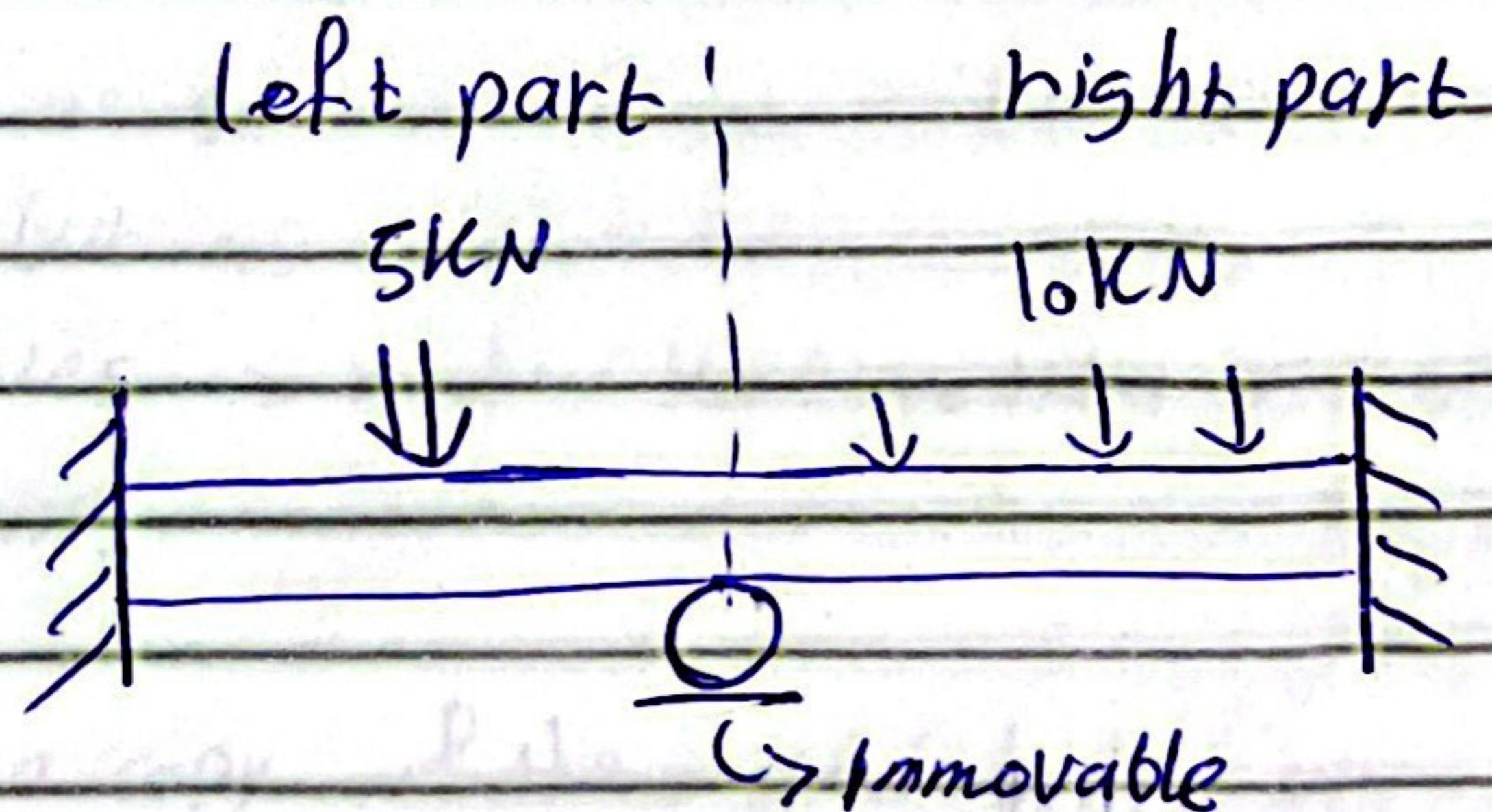
- 1- frames per second \rightarrow smoothness of animation is typical.
- 2- speed \rightarrow Reciprocate.
- 3- animation type \rightarrow
 - Loop.
 - forward only.

Reciprocates \rightarrow move backward and forward in a straight line.

((CASE 2))

conditions :

1. Length : 2000 mm
2. Fixed at end points
3. Immovable center
4. Loads as in Figure
5. Material : AISI 1020



requirements:-

1. stress distribution
2. min and max stress values and their locations

Solution Phases:

1. Adjust the 3D model for simulation
2. setup simulation study and run it
3. Find out & display the max and min stress values.

(model prep)

goal → prepare selectable areas or locations in the model that we can then use to locate a fixture or a load.

Changes to apply in this CASE :-

1. split the model draw a line in the middle → from analysis preparation or command window select split. → you will see The two bodies in the cut list.

2. add a reference point.

reference geometry → select face → center of face

* * it will specify where we want the force to be.

Note:- you will not see joint points as solidworks will interpret The splitted beam as 2 beams but as solid bodies

To Change that:- Select both bodies → right click → treat selected bodies as beams

Note:- it's a good idea to save a copy of the model before working on simulation you may be ((splitting , changing reference geometry)) just for the simulation.

(Setting and getting results)

1. Apply Contacts :- as we have more than one body we can now setup connections ((by default solidwork will apply a global contact condition))

global contact:- where two bodies are in contact they are basically completely glide , welded or sealed to each other almost being one part.

note :- for the middle joint you will choose immovable not fixed fixtures.

note : when applying loads on the left side you will choose point , front plane for direction, normal to plane option .

(min and max values)

right-click on stress-plot → edit definition → Chart options
→ Show max annotation -
Show min annotation

You can display the factor of safety from Results Advisor

(Section 5))

(Shells)

Shells: two dimensional structure with uniform thicknesses.
we can find them in sheet metal structures.

(What is a shell?)

shell → 2D i.e. (paper)

→ small thickness

→ uniform thickness

When to treat a structure as a shell?

1. Made with Solidworks sheet metal tools.
2. Non thickened surface model.
3. Models that resemble a shell.

(CASE 1))

1. Create a selectable area for the force (Drawline - analysis prep split line)
2. Define reference axis for rotation (analysis prep - select the canonical faces)

(CASE 1 - simulation setup)

* note at the left the icon indicating that it's a sheet metal.

⇒ Start new static study → advanced fixtures (reference geometry)

→ Select the two cylindrical faces + axis

note:- we have an example window to show us the type of applicable movements.

⇒ if the icons in the Translational and Rotational sections are gray
This means the model is free to move in all those directions.

|| CASE 1 - reaction Forces ||

Newton's 3rd law of motion; any applied force will experience an equal and opposite reaction force.

To find the reaction force:-

right click on the Results folder → list result force → select face → update

Note:- if you selected all the faces where fixtures are applied then
The resultant force = applied force.

|| CASE 2 ||

Note:- The model does not have any thickness because it is made using solid work surfaces, and so the model is not a solid body of any form

|| CASE 2 - surface thickness ||

How to assign thickness,- Right click on the file name surface → edit definition
→ Thin → offset (Bottom surfaces)

Note:- if there is more than one surface use shell manager.

Note:- the model will still show as a sheet with no thickness, but it is going to be treated as a shell.

• CASE 2 - surface simulations

* We will use the selected direction option when applying forces to be able to accurately define the force direction.

• Section 6 •

(understanding our models)

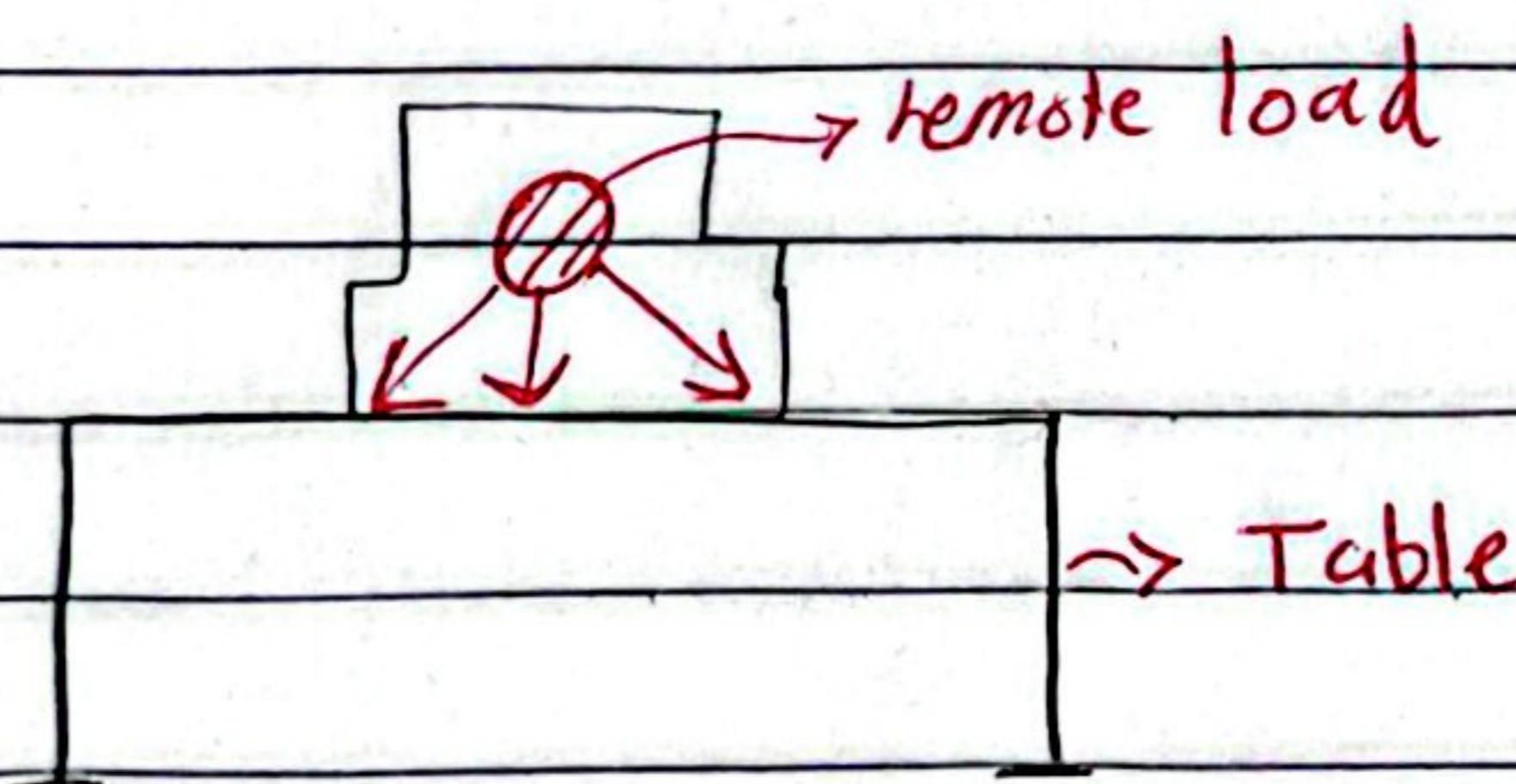
* Sometimes we might get the loads and fixtures and we only translate it to simulation study, However That is not always the case.

We need to know how the model is going to be used in real life so the simulation study is useful to us.

* Identifying the critical cases is very important.

(Remote loads)

Remote loads:- Remote loads allow us to apply forces, displacement and masses to account for a component that is not modeled in the simulation.



* The machine might be too complex to replicate in CAD, or we don't want to invest the resources to include it.

• Adaptive meshing •

* It is one way of auto mesh refinement that we can ask Solidworks to do for us.

* The mesh can impact study results.

* Meshing can be adjusted with manual custom controls.

adaptive meshing

→ h-adaptive mesh \Rightarrow rerun simulation \rightarrow refine \Rightarrow better result

↳ p-adaptive mesh

(Section F - solid bodies)

((what is a solid?))

- * any shape that we cannot classify as a beam and can not classify as a shell can be classified as a solid body.
- * solid bodies can have irregular shapes and non-uniform dimensions.
- * any beam or shell can be treated as solids, but not all solids can be treated as beams or shells.

((CASE 1))

((model simplification))

- * many times we include elements in our design to be more of aesthetic branding or for user experience, in many cases those might not have much notable impact on the strength of our model. However they might be troublesome for the simulation package i.e ((remove grooving and fillets))
- * There is no right or wrong answer on how to simplify a model

((model preparation))

- * The load will be distributed around a circle (50mm).
 - ↳ we need to make that area selectable so we will use split line feature.

((simulations))

((FOS plot))

- * note The plot is all in red which is not communicative and does not indicate what are the actual parts that will cause the failure.

right click on sof plot → **Chart options** → manually enter the max FOS value based on your opinion → **settings** → **Finge options** → **discrete**

* we need to specify where in the part $F_{os} > 4$ to identify the areas that we should not worry about

right click on sof → chart options → specify colors for values above max.

(XY plots)

1. communicate design elements
2. Easy to print, share and interpret.

right click on stress → probe → on selected entities → choose The edge A → update → Report options (plot)

* The x-axis (represent parametric distance not actual distance)

* make sure to select one edge at a time

* if we want to generate a plot for a non-existent edge we can make an edge using split line command.

(Virtual walls)

* For the case where the hanger is on a wall, we need to have a virtual wall.

Step 1:-

Model → analysis prep → plane at a distance from the back

Step 2

back to simulation → right click on connections → local interactions

→ virtual wall → select the back face, plane

* you will notice the deflection of the model does not go beyond this virtual wall

(CASE 2)

- * The model is made with sheet metal functions, but we want to treat it as a solid body because it is thick and the two cassettes do not conform to a typical design of a shell (uniform thickness)
 - * also we are asked to use h-adaptive mesh which only applies to solid bodies
- right click → treat as solid.

(i) Remote loads

right click on external loads → remote load → select face → specify location ⇒ enter force values

(adaptive mesh)

1. create Mesh
2. right click on simulation title ("static1") ⇒ properties ⇒ adaptive

Target accuracy; - acceptable variance of strain between two consecutive loops

accuracy bias; where accuracy is more important

1. local; - care more about areas with stress concentrations

2. global; - care about the entire model equally

Max loops; - maximum number of mesh modification loops

Mesh coarsening; only finer mesh or both ways

- * note if you right click on mesh ⇒ Show ⇒ you will notice it has variable density based on stress concentration.

|| Section 8 ||

(Connections)

* Connections allow us to define how different parts interact with each other.

There are two major categories of connection conditions:-

1. global interactions

2. non-global interactions

* we can first apply global interactions and then non-global interactions as an exception (or override the global default).

→ local interactions, we can specify the interactions between specific areas of our parts.

→ component interactions, specify interactions between whole components.

+ note:- local interactions will override component interactions.

(Connection types)

1. Bonded → parts completely glued or welded

2. Contact (no penetration) → can touch and push on each other without penetrating or getting into each other

3. Free (allow penetration) → areas or parts can interfere or get into one another while or before the simulation is running.

4. Shrink fit → simulate interference fit between surfaces.

5. Virtual wall → simulate something like a wall or floor.

(large vs small displacement)

right click on top of The simulation tree → properties → large displacement

by default solidworks account for small displacements.

the large and small displacement options control how the ^{software} will look at stiffness.

stiffness is a geometric property, design-dependent (Same amount of material can have different stiffnesses because This material was designed and formed differently).

→ small displacement , Constant stiffness

→ large displacement , Variable stiffness

→ The software will apply loads in steps and apply updates on stiffness while This is happening

→ it is a safe option (Hardware demands)

note:- The software has a promoted warning for large displacements, The choice depends on your expectations but if The deformation is not visible to the naked eye use small displacement option ; if it is visible use large displacement option.

|| SECTION 9 ||

« assemblies and multibody parts »

- * in static analysis both assemblies and multibody parts are treated the same
- * it is just as the previous simulation studies but with one more step « connections »
- * They can contain more than one body type at a time.

|| CASE 1 ||

1. we need to Fix the assembly at the shown condition.
2. Start simulation study → right click on the handle → exclude from bonded Global interaction will already exist analysis.
3. if you created a Z-displacement plot you will notice the results are directional +ve, -ve