SSG 511 Part Simplification & Simulation

Instructor: OA Fakinlede
oafak@unilag.edu.ng, oafak.com
Department of Systems Engineering
University of Lagos

Steps in Simulation

- We have been looking at full assemblies of machine elements. At this point we shall go through a step-by-step simulation of a machine element.
- Each part of the assembly can be made of its own material, or it may in fact have several material constituents.
- Forces will be applied to it through contacts with other members, and/or, through kinematical constraints. Some kinematical constraints come from symmetries

KISS: Keep it Simple …

Whenever a choice exists, select the simpler approach:

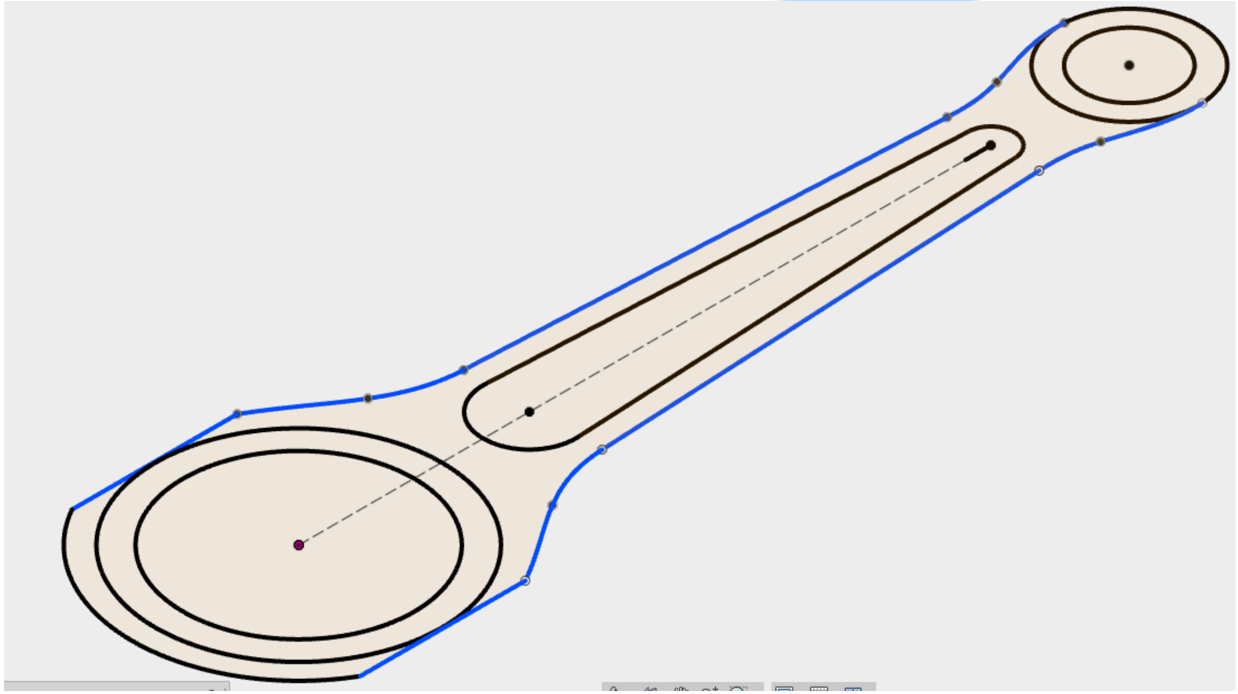
* The simulations you will do is likely going to task your computer to its limits. Do not seek avoidable complexities or complications
* Capitalize on kinematical symmetries.

Simplification Example

Consider the simple connecting rod profile that we are already used to:

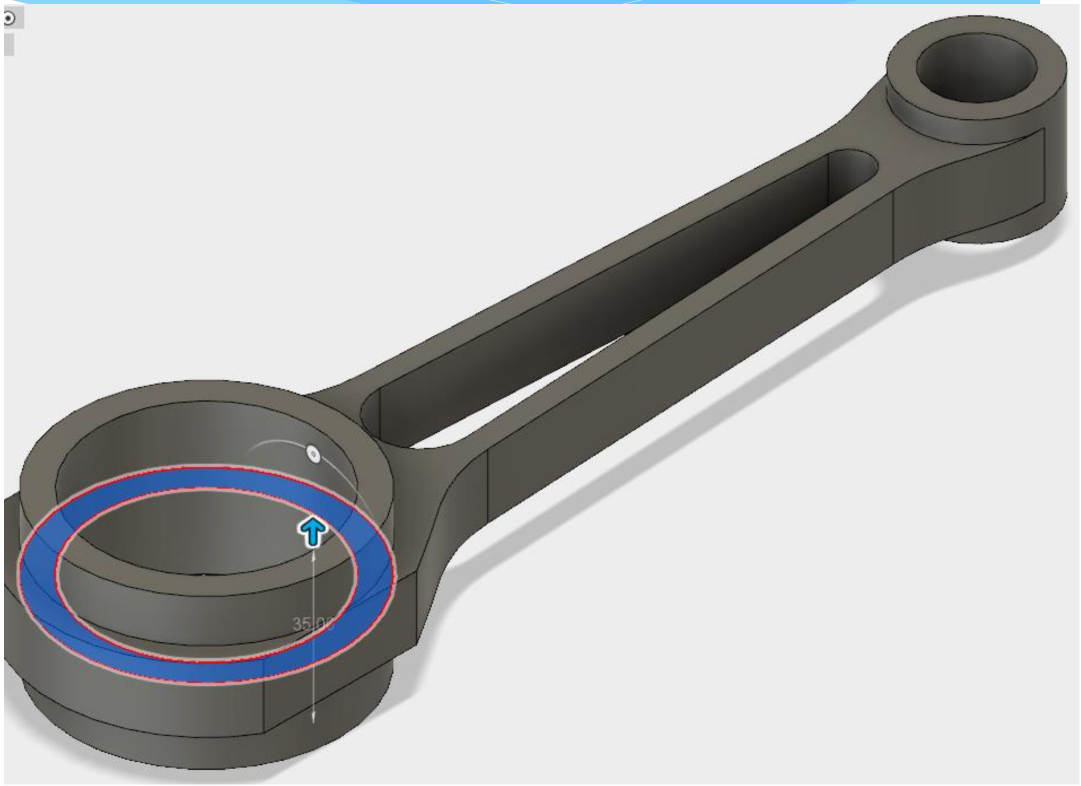
The full rod requires the extrusion of the profile shown here.

Next page is the complete connecting rod



Connecting Rod

* Here, we have just extruded the big pin of the rod to a dimension of 35.
* Of course, NASTRAN can analyze this connecting rod by placing loads on it, meshing and sending the model to solve,
* There is a better way:

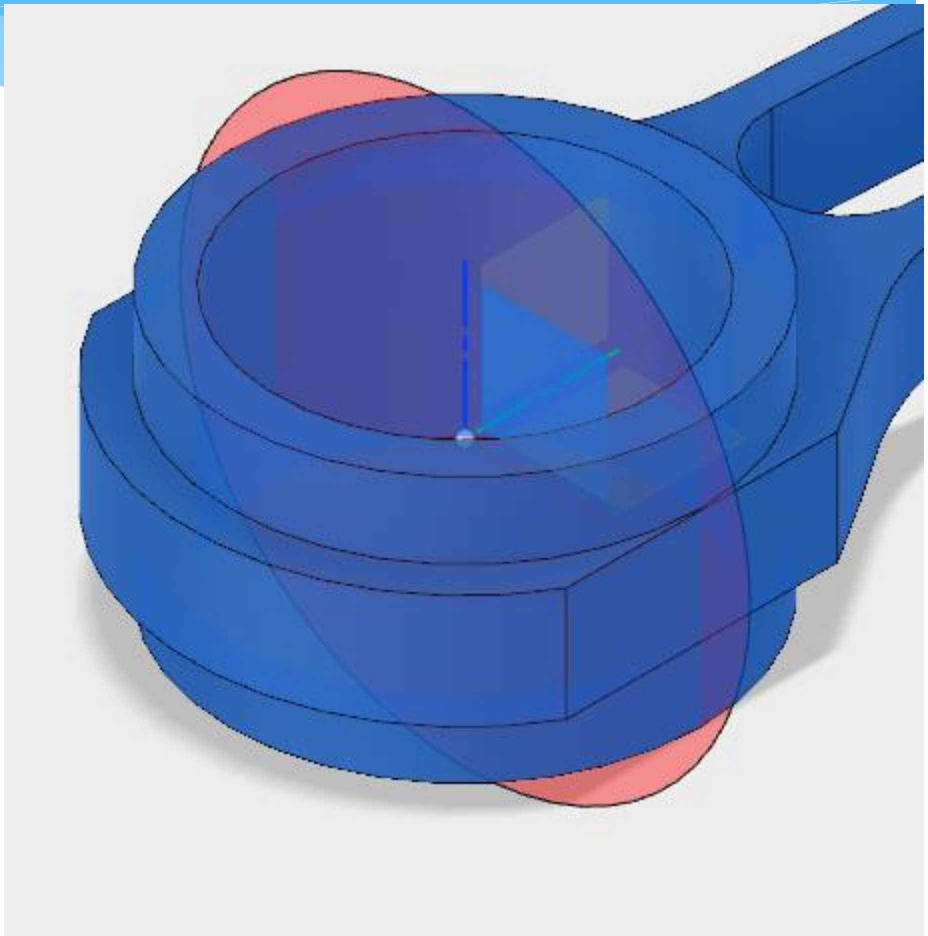


Simplification

* First recall that in reality, we have a split end of the big pin to allow for assembly. The real service rod needs a splitting here.
* Under "Modify" we can request for a split body in the drop down list of options.
* After selecting the body to be split, we need to provide a "splitting tool". The required tool here is the Z-X plane which will act as a cutting plane in this case:

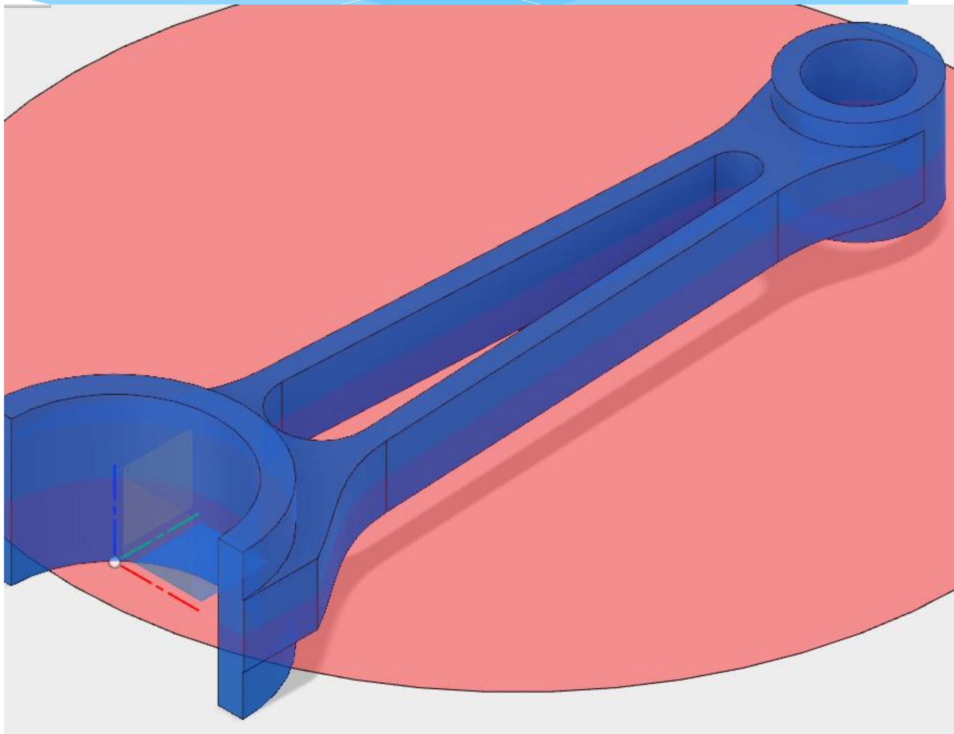
Big-End Split

- The big end split after choosing the splitting tool looks like this:
- Observe the cutting tool presents just enough cutter to enclose the body at the section to be split.
- You can accept this and the body is split here.



Kinematical Split

* The splitting done above represents an actual split in the body. It was added to make the connecting rod more realistic. Our next split here is different. It is of a more technical nature. It is a kinematical split. Our split is for analytical convenience rather than reality.
* It is along the x-y plane as shown



Simple Mesh

* A simple mesh of the simplified component can be seen in:

Constraints & Loading

* For this particular simulation, let us keep the big pin fixed kinematically and all the exterior surfaces free of stress.
* But the split surface is NOT an exterior surface. How do we know what load to apply?
* The answer is that "we don't know"! But we know for sure that no point on that surface can move in the z-direction. For that to happen, the symmetry of motion will be violated. We can add this as a constraint.

Load the small pin

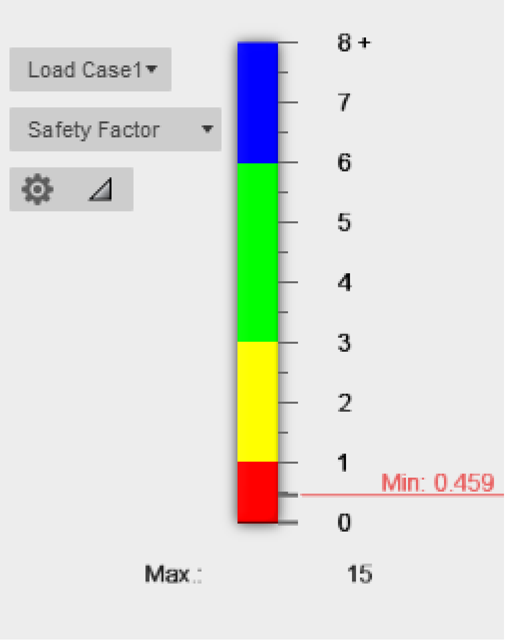
* WE will do two simulations:
* 1. Add 1 kilo Newton in the y-direction to the inner part of the small pin
* 2. Additional x-directional kilonewton in the small pin.

Running the model

- You can run this simple simulation locally or in the cloud
- You do not need to have a lot of bandwidth to run the model in the cloud. The model only loads to the cloud and the actual execution and heavy-lifting takes place there and your results are sent back.
- You can try both to see how it works out.

Results

* You may, at this point, go to the results view under display.
* You will see the following legend choices usually at the bottom right corner of your screen:
* Behind the second drop down menu, you may select such things as displacement, strain, stress, reaction forces, etc.



Displacement

* You already know that for each point in the body, there are three displacements. Fusion 360 adds a fourth: the total displacement – a vector sum of the three displacements.
* Usually, we are concerned with only one of these in practical situations. Why is that so?

Stress

* You have the possibility of observing up to 9 different stresses. What are these?
  1. The six components of the stress tensor
  2. The von-Mises equivalent stress
  3. Two other stress invariants
* Recall that engineers design with equivalent stresses or the von Mises stress.

Strain

- The number of elements displayed follow the same line as the stresses. Why is that so?
- Recall that strain is also a tensor. A symmetrical tensor so that it has six components. In addition, just like any other second-order tensor, it also has invariants – on of which is the equivalent strain.
- All these are available in the results

Reaction forces

* How many reaction forces will you expect Fusion 360 to display for you at any point? Why?
* Where will you expect these to occur?