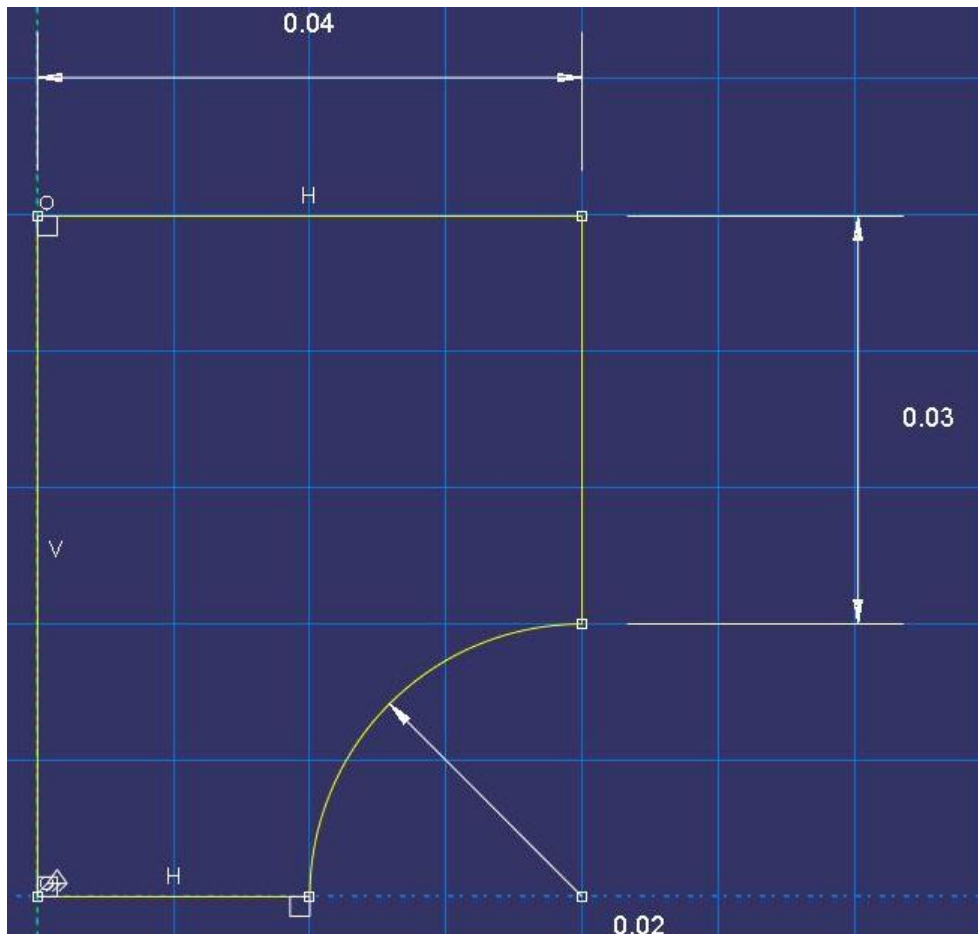


Remeshing Example

- Lunch **Abaqus** software
- In Part module: **part**→**create**; name=*Rod*, modelling space=*axisymmetric*, type=*deformable*, base feature=*shell*, aproximate size=*0.5* and continue
- Sketch your model as you can see below



- In Property module: **Material**→**create**; name=*steel*, **Mechanical**→**Elasticity**→**Elastic**; young modulus= $200e9$, poisson ratio= 0.3
- In Property module: **Section**→**create**; name=*Section-steel*, category=*solid*, type=*homogeneous*, continue and Ok
- In Property module: **Assign**→**section**; select the model, click Done and Ok
- In Assembly module: **Instance**→**create**; Instance type=*independence* and OK

- In Step module: **Step**→**create**; name=*static*, procedure type=*static, general*, Continue and Ok
- In Load module: **BC**→**create**; name=*axi-bc*, step=*Initial*, category=*mechanical*, types of selected step=*symmetry/antisymmetry/encastre*, Continue and pick the left line from model, choose *xsymm* and OK
- In Load module: **BC**→**create**; name=*sym-bc*, step=*Initial*, category=*mechanical*, types of selected step=*displacement/rotation*, Continue and pick the bottom line from model, toggle on *U2* and Ok
- In Load module: **load**→**create**; name=*pressure-load*, step=*static*, category=*mechanical*, types of selected step=*Pressure*, Continue and pick the top line from model, magnitude=*1000*
- In Mesh module: **Seed**→**Edge Biased**; choose bottom line from model and Done, Bias ratio=*7* and Enter, Number of element along the edge=*7* and Enter, again choose right line from model and Done, Bias ratio=*7* and Enter, Number of element along the edge=*7* and Enter
- In Mesh module: **Mesh**→**Element type**; select the model and Done, Element library=*standard*, Family=*axisymmetric stress*, Geometric order=*Linear* and OK
- In Mesh module: **Mesh**→**Controls**; select the model and Done, Element shape=*Quad-dominated*, Technique=*free* and OK
- In Mesh module: **Mesh**→**Instance**; select the model and Done
- In Mesh module: **Adaptivity**→**Remeshing rule**→**create**; select the model and Done, name=*RemeshingRule*, error variables indicator=*mises equilanet stress*, in Constraint tab you can define your maximum and minimum size of your elements and OK

1- Manually remshing

- In Job module: **Job**→**create**; name=*remesh-job*, Continue, job type=*full analysis* and OK
- In Job module: **Job**→**manager**; submit the job
- After the solution, in Mesh module: **Adaptivity**→**manual Adaptive remesh**; ODB=(*your previous job Odb file address*), click on Display results to see the error indicator results and the element count, since you are not satisfied with your error value, click on remesh model, then submit the job again and repeat this step till you will be satisfied

2- Automatic remeshing

- In Job module: **Adaptivity** → **create**; name=automatic-remeshing, automated controls=3 and OK
 - In Job module: **Adaptivity** → **manager**; submit the adaptivity process
 - Your number of iterations for automated controls declares the maximum number of operations to reach the user defined Constraints which were adjusted in Mesh module: **Adaptivity** → **Remeshing rule**
-
- In Mesh module: **Mesh** → **verify**; select the model and Done, here you can verify your remesh model due to the size and aspect ratio of elements

Tips:

- 1- You can define a partition to not remesh the whole model and only remesh the location where you want
- 2- You can use the Manually remshing and Automatic remeshing together