PLANE STRESS ANALYSIS USING THREE POINT BENDING TEST

Spring Semester 2021

CE42006 : Behaviour of RC Structures

Assignment 2

Abaqus Modelling

Deepak Choudhary

17CE31001

Department of Civil Engineering Indian Institute of Technology Kharagpur 721302

Part 1

Introduction to model

Software used is **Abaqus 6.14**.

This project involves studying the failure patterns in a concrete cylinder of diameter 150 mm and variable height (300 mm, 600 mm, 1000 mm) loaded in unconfined compression by stiff as well as flexible plattens.

An axisymmetric finite element model is created for cylinder in abaqus. Concrete material is modelled using Concrete Damaged Plasticity model.

Properties given for concrete are –

- Compression strength = 27 MPa
- Ultimate strain = 0.003.
- Density = 2400 Kg/m3
- Young's modulus = 29 GPa
- Poisson's ratio = 0.2

Plastic properties for concrete are –

- Dilatation angle = 36 degree.
- Flow field eccentricity, ε = .1
- ratio of biaxial compressive yield stress to uniaxial compressive yield stress = 1.16
- Kc, ratio of stress invariants on tensile and compressive meridian = .67

Three Point Bending test.

Procedure –

Part –

- Modelling-2D Model,
- Type- Deformable,
- Base feature- Shell.

Partition - Due to notch Partition is used to create uniformly aligned mesh.

Property – Concrete Property same as assignment 1 using Popovic,s paper (Compression damage) and Hordijk,s paper (Tensile damage)

Step – Dynamic Explicit step is created, with Smooth step amplitude

Mesh -

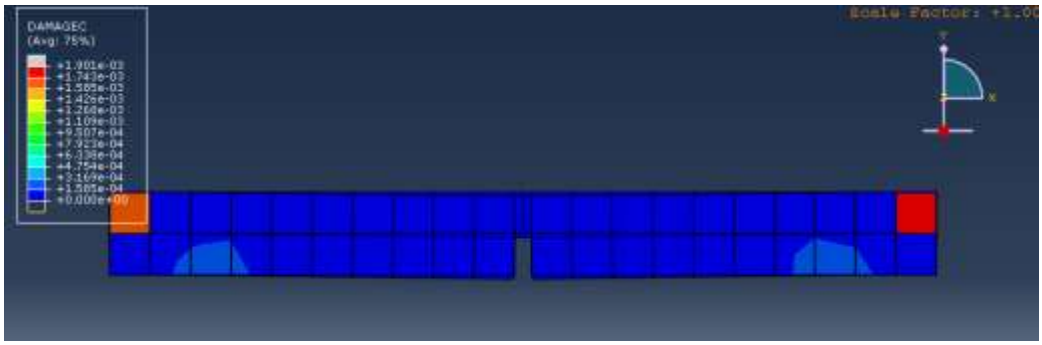
- Element type – Explicit
- Control – Quad, Structural
- Set- 3 sets are created
  (2 for reaction forces, 1 for Displacement)
- Mesh size : 100mm is used.

Load – Boundary condition are created. For 2 support in both lower sides, pinned case is used. For applying load displacement is given of amplitude 5mm in 1 sec. (speed= 0.05m/s)

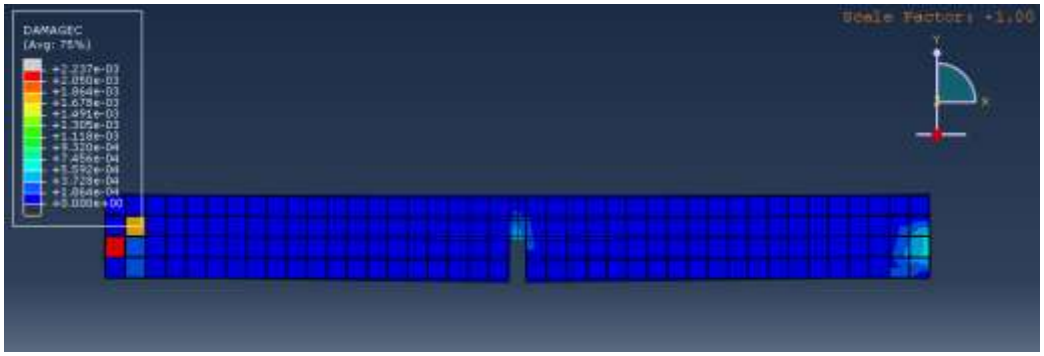
Part 2

Convergence and Mesh Dependence

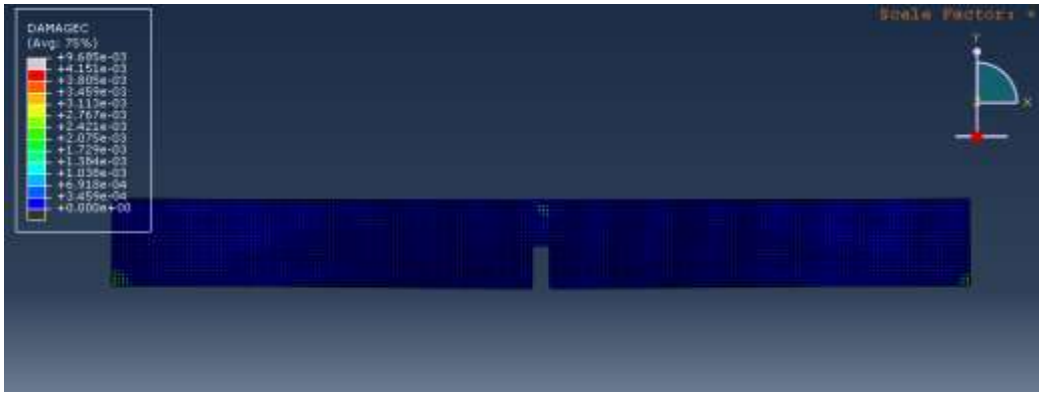
Mesh :100size Element 84



Mesh :50 element: 328



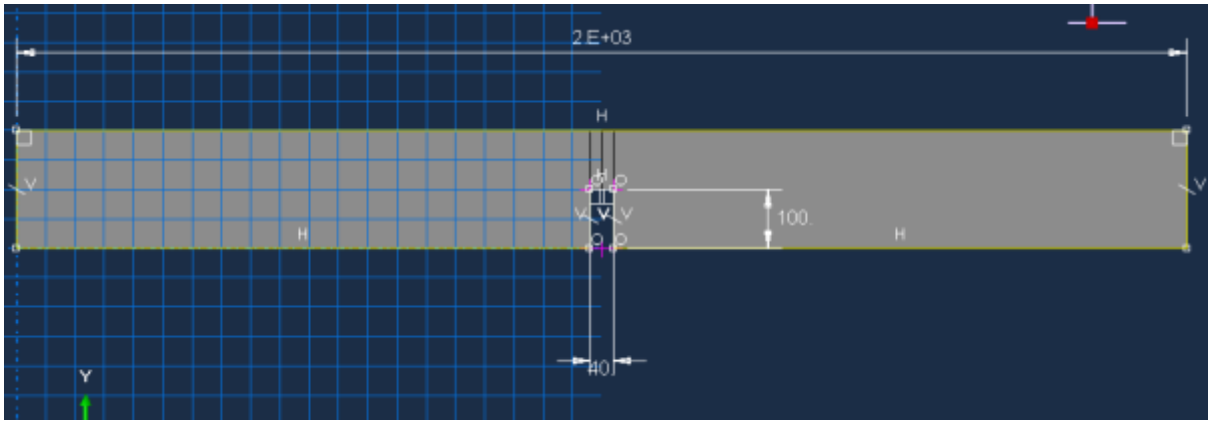
Mesh :10 element: 7920



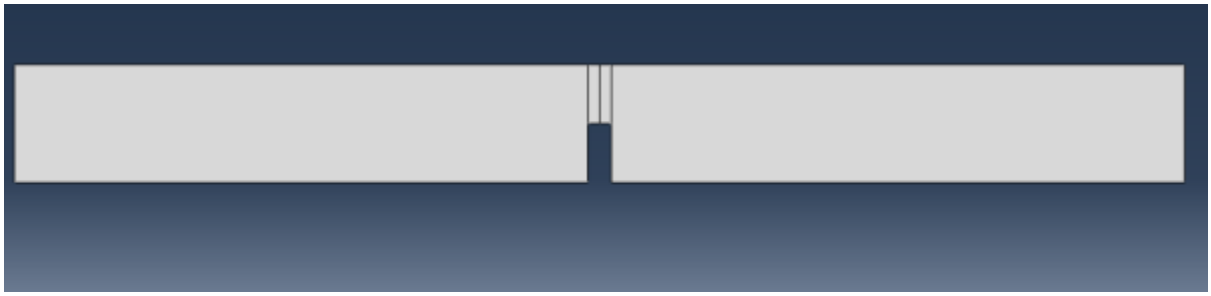
Part 3

Model Size 1:1

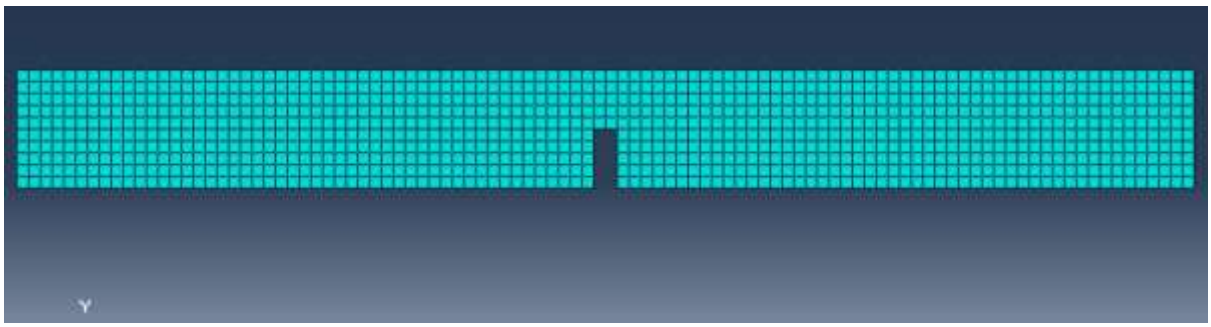
Model Dimension –



Model Partition-

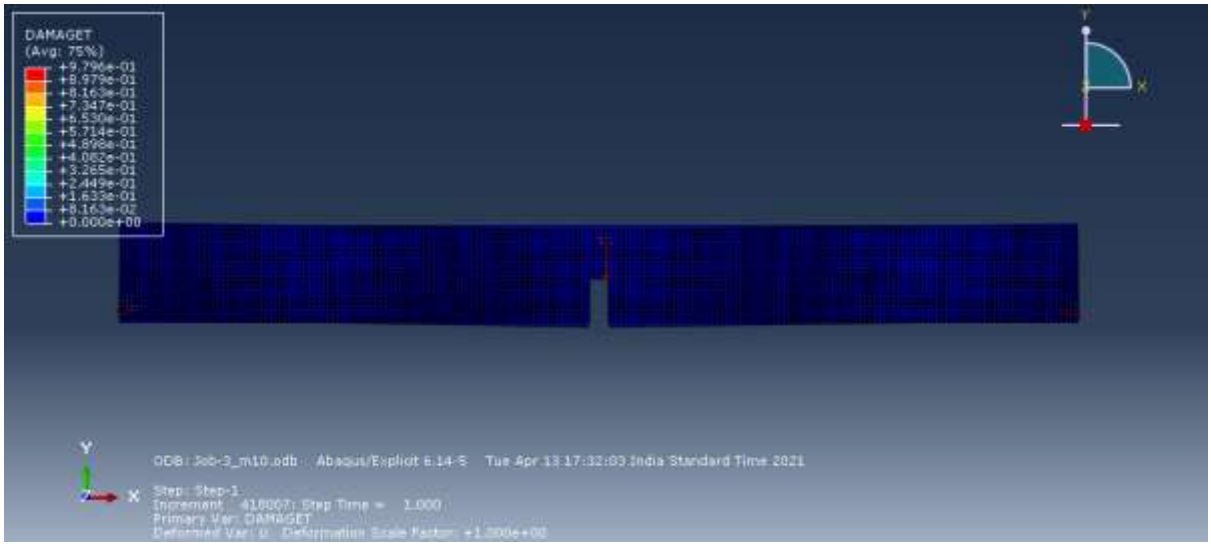


Mesh-

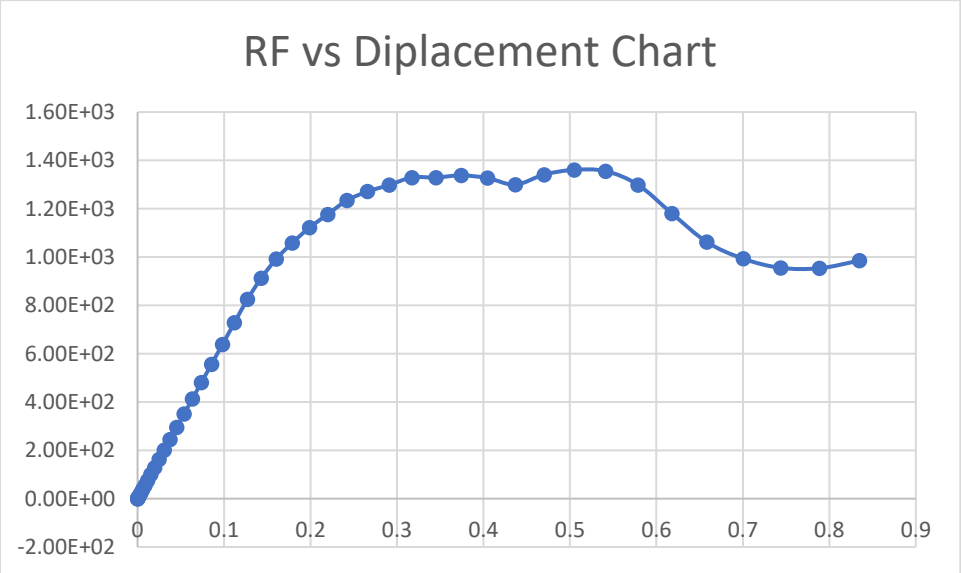


a) Full Fracture Energy

Tensile damage

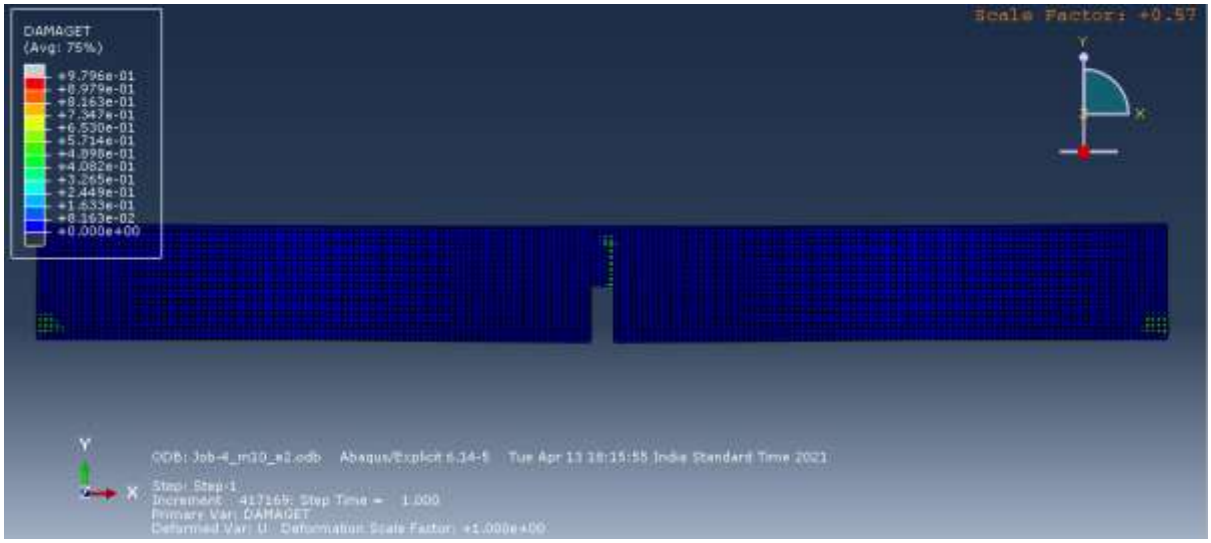


Force displacement curves

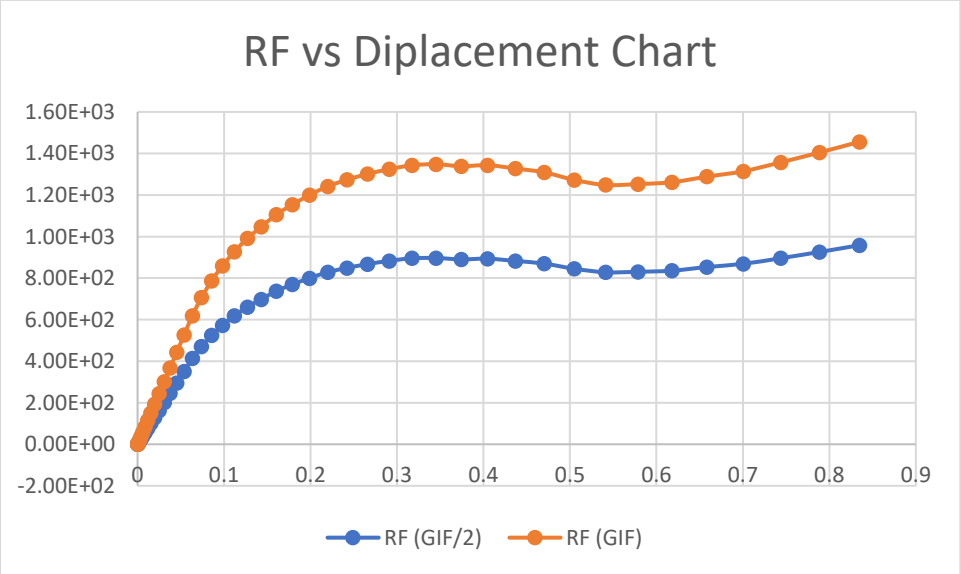


b) Half Fracture energy

Tensile damage



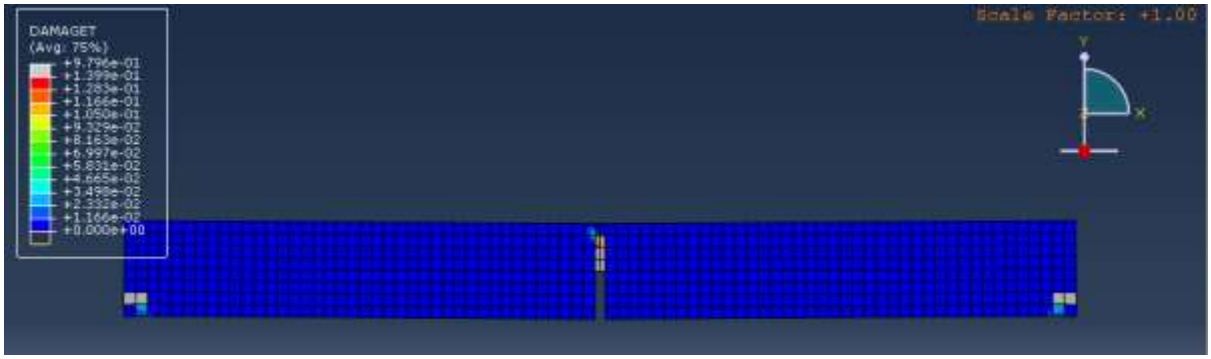
Force displacement curves



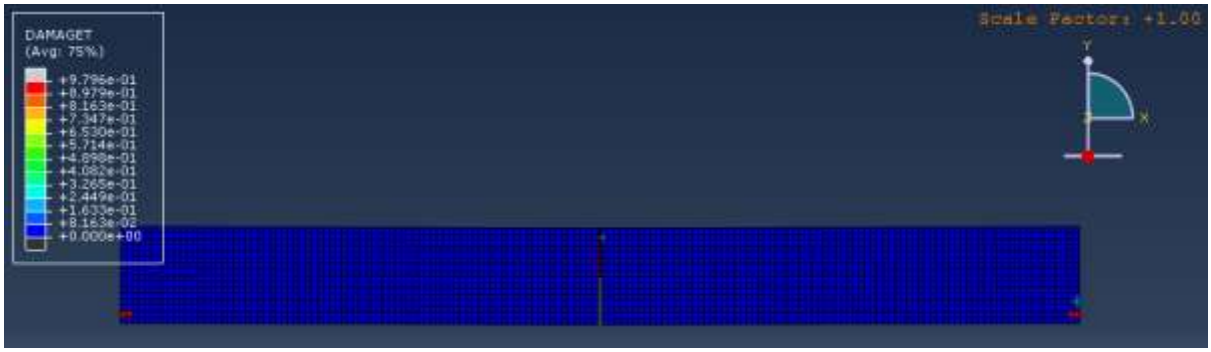
Part 4

Self Similar Beams

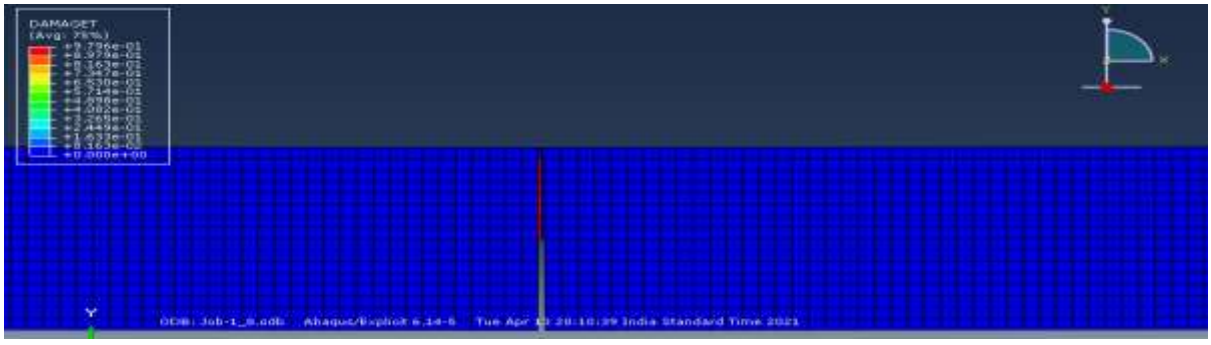
1:2 Tenslile Damage



1:4 Tensile Damage

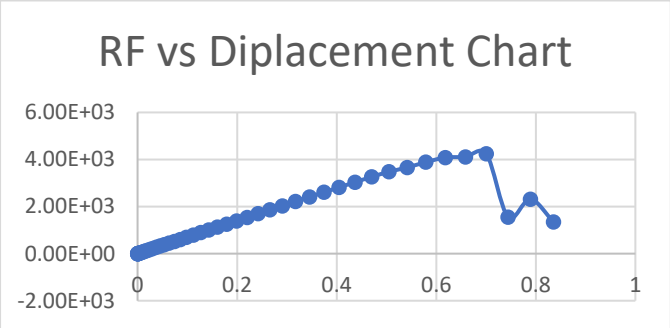


1:8 Tensile Damage

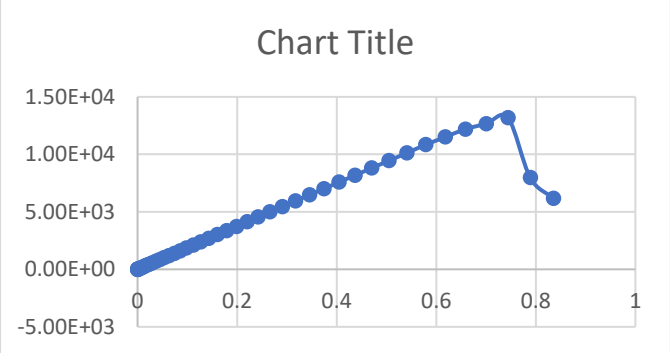


Force displacement curves

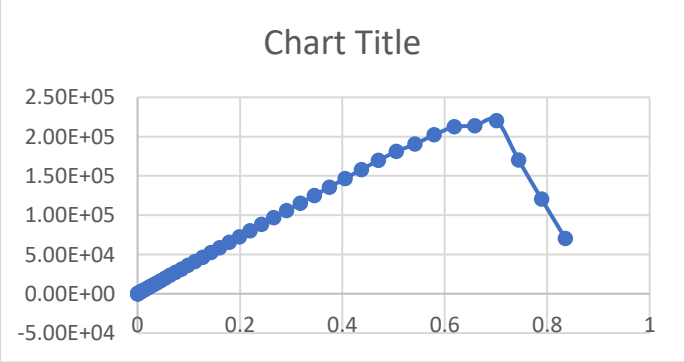
For size 1:2



For size 1:4



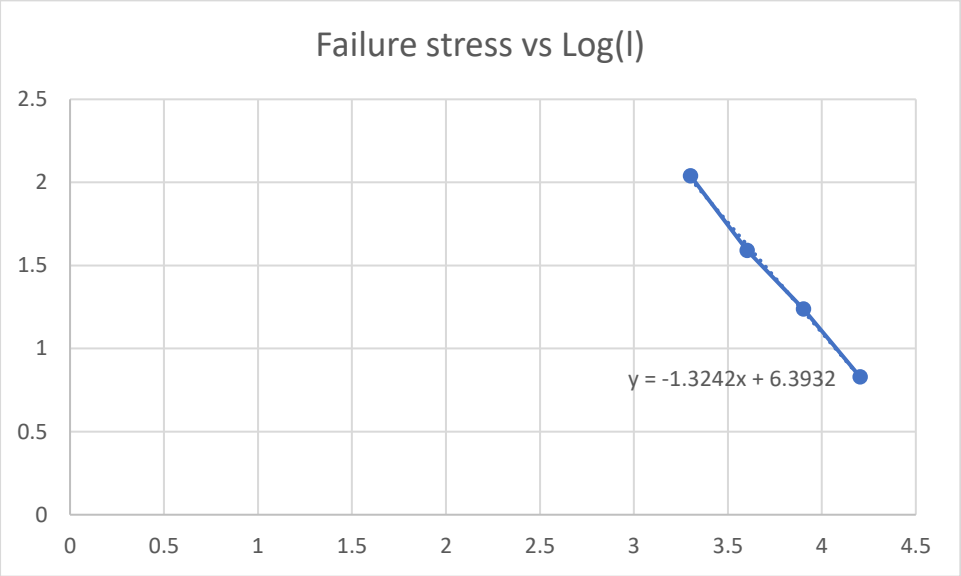
For size 1:8



Relation Between Failure stress versus log (*l*)

Model	L (mm)	log(L)	b (mm)	D (mm)	RF	F
size 1:1	2000	3.30103	50	200	1360	2.04
size 1:2	4000	3.60206	100	400	4240	1.59
size 1:4	8000	3.90309	200	800	13200	1.2375
size 1:8	16000	4.20412	400	1600	35360	0.82875

Graph



Empirical Equation

$$\sigma = \text{-}1.3242\log(\text{l}) + 6.3932$$

Part 5

Reference

- ABAQUS Analysis User's Manual, v6.9, Simulia Corp., Providence, Rhode Island.

- "A Numerical Approach to the Complete Stress Strain Curve of Concrete", Popovics S., *Cement and Concrete Research*, 3, 1973.

- "Local approach to Fatigue", Hordijk, P. A, Doctoral thesis, Delft University of Technology, 1991.

- Concrete damaged plasticity - https://abaqus-docs.mit.edu/2017/English/SIMACAEMATRefMap/simamat-c-concretedamaged.htm#simamat-c-concretedamaged-compressivehardening