Final Project Report

CEE 328

Submitted By:
Anna Cooper, EIT
Robert Dieter II

Date Submitted: 12 June 2025

Abstract

In this project, we seek to model a real-world scenario from Northwestern's concrete lab in which an ultra high-performance concrete (UHPC) slab undergoes an applied load from a piston located overhead. Although UHPC is quasi-brittle, we simplify the material to be brittle for the purposes of our model. Accordingly, the Abaqus simulation results reflect this assumption: although the cracks appear in the same locations in real life and in the model, a lower applied force before fracture is achieved in the model compared with the true force applied in the lab. This behavior is visible on the model's force-displacement curve, which corresponds to a brittle fracture.

Introduction

Figure 1: Concrete 3D printer in action

In Prof. Gianluca Cusatis's lab at Northwestern University, research surrounding 3D printed concrete is currently underway. One area of study is comparing the properties of 3D printed concrete objects with traditional, cast concrete objects by performing fracture tests on concrete plates produced using both methods.

Figure 2: Cast and 3D printed concrete slabs (note crack lines from load tests)

This project seeks to investigate whether the fracture of the cast concrete slab, like that shown above, can be accurately modeled in Abaqus. To this end, we will seek to use the same geometry, material properties, and loading/support conditions in the model as in the lab's real experiments. After the Abaqus analysis, we will compare the simulation results with the results from real lab tests.

Background

In order to mirror the lab tests as closely as possible, we will use the same properties and conditions as those used in the lab, with only minor assumptions and simplifications as specified here.

Material Properties

The material used in the lab is ultra high-performance concrete (UHPC), the defining characteristic of which is the steel fibers (approximately 1cm in length) that are mixed into the concrete. Because the steel fibers are randomly distributed throughout the slab, the material can be treated as isotropic.

Figure 3: Steel fibers visible along the internal crack edge and outer surface of the slab

In conversations with Raul, we learned that UHPC behaves as a quasi-brittle material. Displaying characteristics of both brittle and ductile materials, UHPC can undergo some minor deformation before fracturing, but then fail suddenly. Furthermore, in quasi-brittle materials, fracture occurs in a *region* where multiple crack initiations collide. (Unlike in brittle materials, where fracture occurs as a *point*, and in ductile materials, where the failure is plastic).

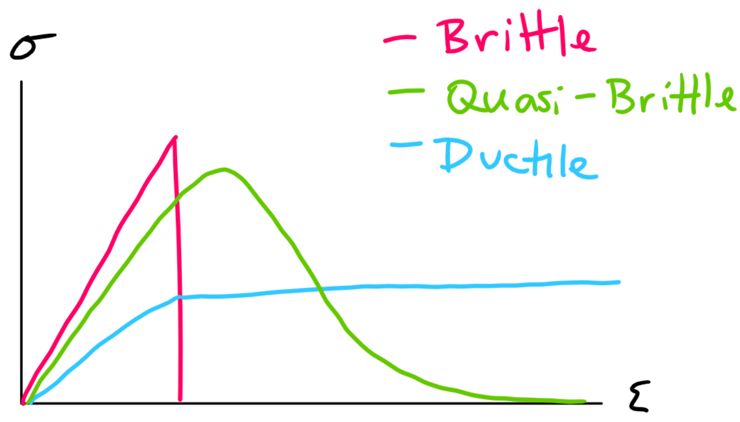

Figure 4: Stress-strain curves for different types of materials

For the purposes of this project, Raul advised that we could simplify the analysis by treating UHPC as a brittle material, as that behavior aligns more closely with the subject of our test. This simplification will be revisited in the following sections.

The UHPC used in the lab has the following material properties:

Table 1: UHPC Material Properties

σ_c	120 MPa
σ_t	$10\% \cdot \sigma_c = 12$ MPa
E	50 GPa
υ	0.20
Specific weight γ	0.084 lb/in^3

Using the brittle material assumption, the concrete slab will experience full failure when σ_t is achieved.

Geometry

The concrete slab we model will use the same geometry as the real slabs used in the lab:

Table 2: Concrete slab geometry

Length	32”
Width	32”
Thickness	2.75”

With this geometry and the specific weight from above, the slab weighs approximately $(0.084 lb/in^3)(32in)^2(2.75in)$ = **236.5 *lbs***. Raul verified this, commenting that lifting a single concrete slab typically requires four people.

Loading and Boundary Conditions

The final considerations for mirroring the real-life tests are the loads and boundary conditions. In the lab, the slab is simply supported on all sides. A load is applied at the center of the slab by a 4”-diameter steel piston arm under displacement control. To maintain a quasi-static system (i.e. neglect dynamic effects), the piston head moves on a geological time scale, approximately 0.01mm/s.

In Abaqus, the simple supports will be applied as a zero displacement boundary condition in the z-direction (U3=0) along the edges of the slab, with the x- and y-directions unconstrained. That x- and y-direction displacements remain free is necessary for the model to produce realistic damage results; in real lab tests, the slab splits apart after cracks form, and therefore the slab must be free to split apart in Abaqus as well.

Figure 5: Testing apparatus

Methods

The Abaqus model was dynamic, explicit with consideration for nonlinear geometry due to significant plate bending and damage. The plate was modeled as a 3D, deformable, planar, homogeneous shell with STRI3 element type, with element deletion enabled. It was supported by rollers on all edges and placed in contact with a rigid piston arm that was displacement controlled. The imposed displacement control was set to be 0.5”. This was a somewhat arbitrary selection, but it is proportional to the size of the model, and it resulted in full crack formation (most similar to experimental results). The piston arm was modeled as a 3D, deformable, homogeneous solid, extrusion with a depth of 12” and a radius of 2”.

Results

Although Raul was unable to provide exact plots showing the lab’s experimental results, he suggested two ways of verifying our model. Firstly, Raul noted that the maximum reaction force achieved before fracture is typically about 1500lbs. Secondly, visual observations of the fractured concrete slabs indicate that cracks appear along the vertical and horizontal directions through the center of the plate (i.e. along the axis of symmetry), and also radially originating from the piston head.

Reaction Force

After running the Abaqus simulation, the force-displacement dataset was extracted from the Visualization module and plotted below. The maximum force applied by the piston arm at the center of the concrete plate reached a magnitude of 468.5 lbs. Figure 6 below shows the force-displacement curve resulting from the finite element simulation:

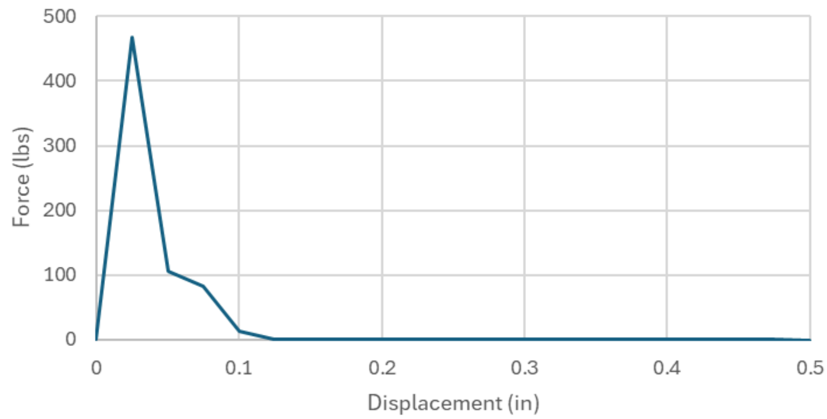

Figure 6: Abaqus force vs. displacement plot

Observing the force-displacement plot, we are reminded of the assumption made earlier to model UHPC as a brittle material instead of a quasi-brittle material. Indeed, this curve shows a very small displacement as the force increases until fracture, at which time rapid fracture occurs and the force drops off – just as we would expect for a brittle material.

However, we note that the maximum force value (468.5 lbs) is about 70% less than the experimental value of 1500 lbs. More than likely, this phenomenon can again be explained by the same modeling assumption. Because the simulated slab fails due to brittle fracture, a lower load at failure is expected than would be achieved if some ductile behavior were permitted. In that case, both the deflection and the reaction force would likely have been greater.

Crack Appearance

Recall from Figure *2* above (results from the lab test) that the primary cracks appear along the slab's axis of symmetry (where the bending moments are greatest), with additional cracks emanating radially from the center. This pattern is also achieved using concrete damage plasticity in the Abaqus model.

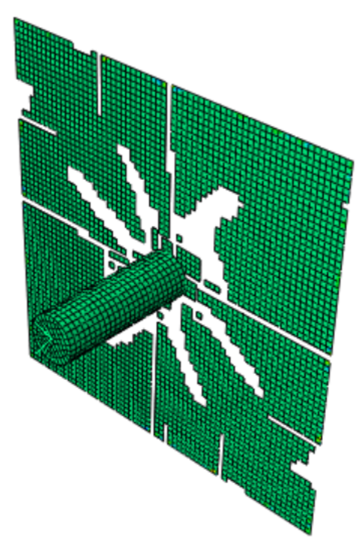

Figure 7: Abaqus model showing damage

Although the mesh resolution is somewhat coarse (refining the element size beyond 0.5” created jobs that took > 15 minutes to run), this same pattern can be seen: cracks spanning the full length and width of the slab, in addition to radial cracks starting near the head of the piston.

Conclusion

Several aspects of these results suggest that a UHPC slab under these loading conditions can be successfully modeled in Abaqus. The force/displacement curve displays brittle behavior, which is what it was idealized to be. Since the concrete plate, in reality, is a quasi-brittle material, an ultimate failure load lower than the experimental value (1500 lbs) was expected. Finally, the crack pattern on the deformed shape mirrors the experimentally observed failure behavior as well. The primary cracks can be seen extending along the x- and y-axes of symmetry in both cases. The radial cracks extending from the center are visually similar as well.

Further studies might seek to produce more accurate results by treating UHPC as a quasi-static material in Abaqus, rather than using the brittle material modeling assumption we made here.

THANK YOU Prof. Fleming for a great quarter! We both really enjoyed this class, especially hearing all the stories from your work. Thanks for letting us choose a final project we were passionate about - it allowed us to explore some new areas of Abaqus, and of Northwestern!
Cheers!

- Robert and Anna

Appendix A: Abaqus Modeling Procedure

Part

- Create Part (Part-1; Plate)
    - 3D, Deformable, Planar, Shell
    - Rectangle
        - First pt = (-16, -16) [in]
        - Second pt = (16, 16) [in]
- Create Part (Part-2; Piston Arm)
    - 3D, Deformable, Solid, Extrusion
    - Circle
        - Center pt = (0, 0) [in]
        - Perimeter pt = (2, 0) [in]
        - Depth = 12 [in]

Property

- Create Material (Concrete)
    - General, Density
    - Mechanical, Elasticity, Elastic
        - E = 7.252e6 [psi]
        - $\nu = 0.2$
    - Mechanical, Plasticity, Concrete Damaged Plasticity
        - Compression Behavior (enter experimental yield stress and inelastic strain data)
            - Suboptions, Compression Damage (enter experimental damage parameter and inelastic strain data)
        - Tension Behavior (enter experimental yield stress and cracking strain data)
            - Suboptions, Tension Damage (enter experimental damage parameter and inelastic strain data)
- Create Material (Steel)
    - General, Density
        - 1e9 [$lb \cdot s^2/in^4$]
    - Mechanical, Elasticity, Elastic
        - E = 30e6 [psi]
        - $\nu = 0.3$
- Create Section (Part-1; Plate)
    - Shell, Homogeneous
    - Material: Fiber Reinforced Concrete
    - Shell Thickness = 2.75 [in]
- Create Section (Part-2; Piston Arm)
    - Solid, Homogeneous
    - Material: Steel
- Assign Section (Part-1; Plate)
    - Section: Plate
    - Shell Offset = Middle Surface

- Assign Section (Part-2; Piston Arm)
  - Section: Piston Arm

Assembly

- Create Instance
  - Select both parts
  - Instance Type = Dependent
- Tools, Reference Point
  - Place a RP in the center of the piston arm's circular surface away from the origin

Step

- Create Step
  - Dynamic, Explicit
    - Note: Nlgeom = ON (due to large rotations in this problem)
  - Mass Scaling
    - Use scaling definitions below
    - Create
      - Type: uncheck "Scale by factor"; check "Scale to target time increment of:", Increment = 1e-5

Interaction

- Create Interaction
  - General Contact (Explicit)
  - Create Interaction Property
    - Contact
    - Mechanical, Tangential Behavior
    - Friction Formulation: "Penalty"
    - Friction Coefficient = 0.1
  - Global Property Assignment: "IntProp-1"
- Create Constraint
  - Rigid body
  - Region type, select "Body (elements)"
  - Edit Selection
    - Select piston arm
  - Reference Point, Edit
    - Select RP on piston arm

Load

- Create Boundary Condition
  - Displacement/Rotation
  - Select plate edges
  - Select U3
- Create Boundary Condition

- Displacement/Rotation
- Select RP-1
- Check "U3", set equal to -1; check all others and leave equal to 0
- Create Amplitude
  - Tabular [(0,0), (1,1)]
- Amplitude = Amp-1

Mesh

- Element Type (Part-1; Plate)
  - STRI3
  - Enable element deletion
- Edit model database according to this video
- Seed Part (Part-1; Plate)
  - Select plate
  - Approximate global size = 0.5
- Mesh Part (Part-1; Plate)
  - Select plate
- Seed Part (Part-2; Piston Arm)
  - Select piston arm
  - Approximate global size = 0.5
- Mesh Part (Part-2; Piston Arm)
  - Select piston arm

Job

- Create Job
  - Submit

Visualization

- Tools, XYData, Create
  - ODB field output
    - Variables
      - Position: Unique Nodal
      - Select U3 and RF3 from list
    - Elements/Nodes
      - Node sets
      - REFERENCE_POINT_2
    - Plot
- Tools, XYData, Create
  - Operate on XY data
    - Operators
      - combine(X,X)
        - XYData
          - First X: select U3

            - Second X: select RF3
        - Save As
- Tools, XYData, Manager
    - Select XYData-1
    - Edit
    - Copy data and paste in Excel