ACIC 2019

2D FINITE ELEMENT ANALYSIS OF GFRP REINFORCED CONCRETE CONTINUOUS DEEP BEAMS WITH BOND MODELLING

Othman Hameed Shalookh
Professor Ashraf Ashour
Dr. Therese Sheehan

CONTENTS

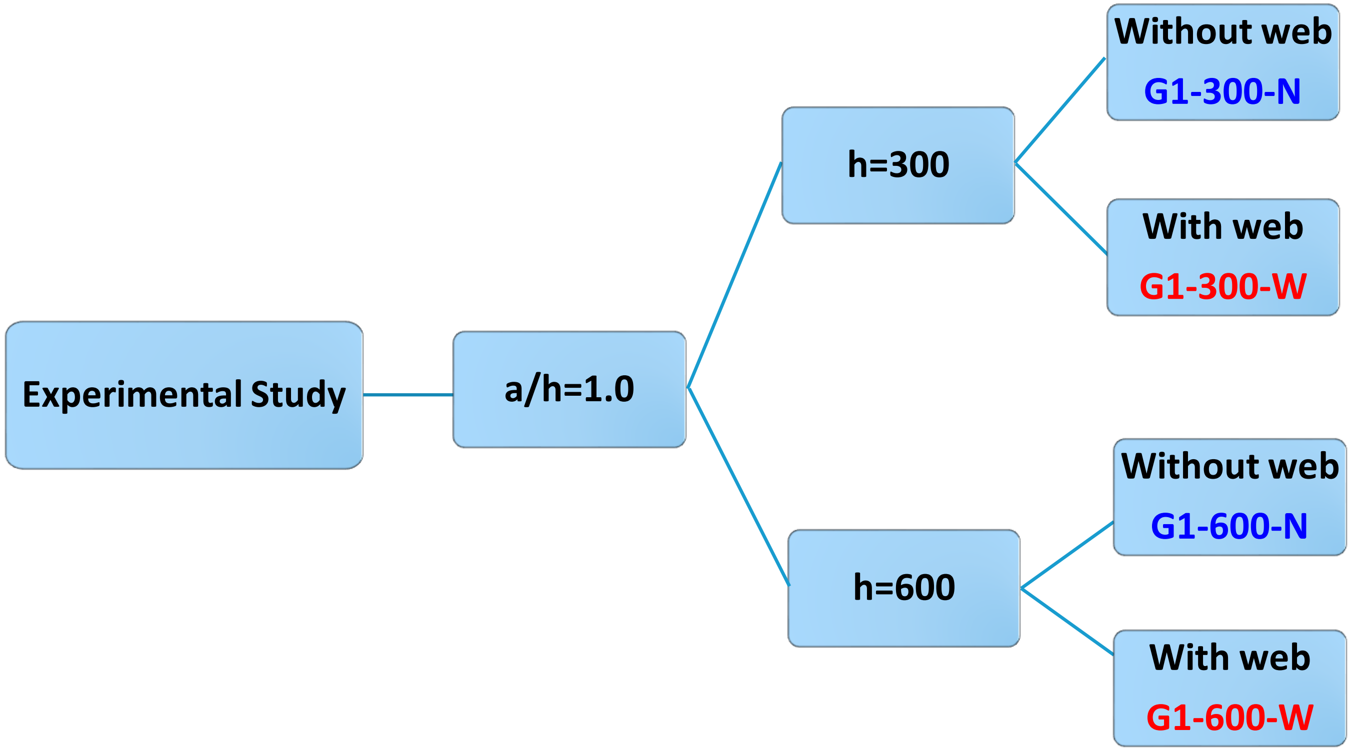
- Research significance
- Experimental programme
- Details of the finite element model
    - Constitutive models of materials
    - Element types
    - Boundary conditions
    - Mesh size
    - Interaction between FRP bars and concrete
- Finite element results
- Conclusions

RESEARCH SIGNIFICANCE

- The current paper presents the first numerical investigation for FRP RC continuous deep beams.
- The finite element model introduced in this paper took into account the bond effect between the longitudinal reinforcement and the concrete using the results of pullout tests conducted for this purpose. Additionally, a perfect bond between the longitudinal reinforcement and surrounding concrete was also modelled to evaluate the validity of this assumption introduced by many previous FE studies.
- The validated FE model can be used then to implement a parametric study for a wide range of the critical parameters that govern the behaviour of such beams.

EXPERIMENTAL PROGRAMME

➢ Experimental Plan



Experimental Study

a/h=1.0

h=300

Without web

G1-300-N

With web

G1-300-W

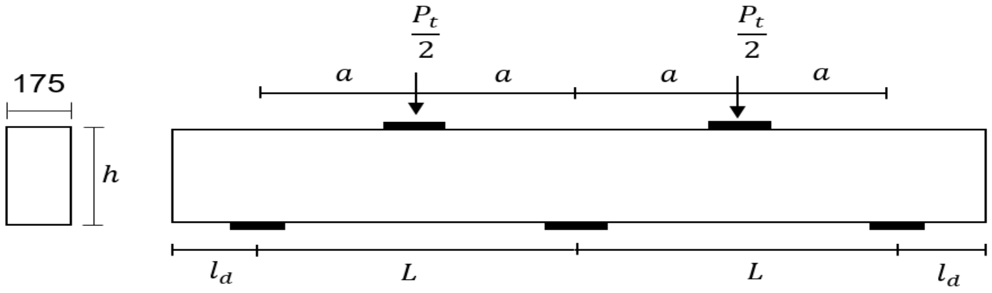
h=600

Without web

G1-600-N

With web

G1-600-W



Constant values

a/h=1.0

b=175 mm

Longitudinal reinforcement=1.2%

Development length=400 mm

Compressive strength≈55 MPa

Web reinforcement

ρ_v and ρ_h=0.4%

EXPERIMENTAL PROGRAMME

➢ Specimens’ preparation

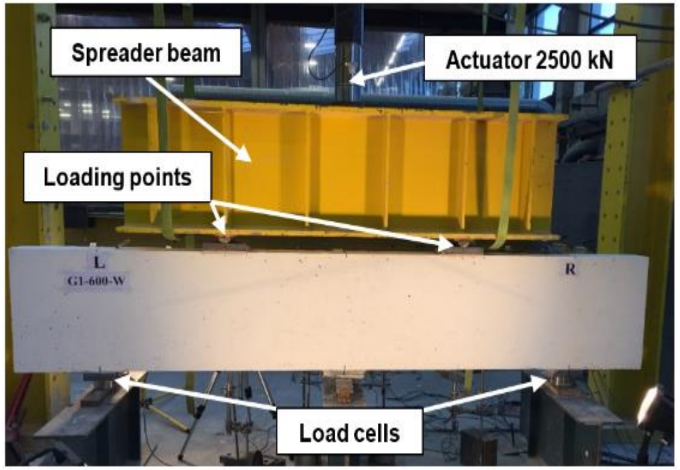
Formwork

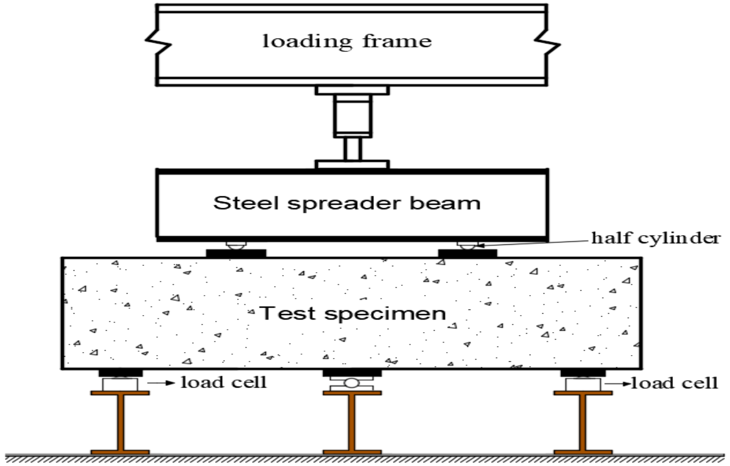
Ready mix concrete

Pullout samples

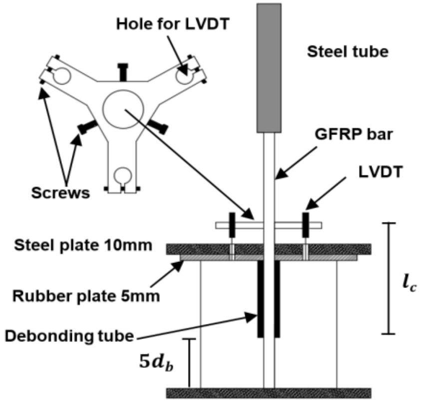
EXPERIMENTAL PROGRAMME

➢ Test Setup





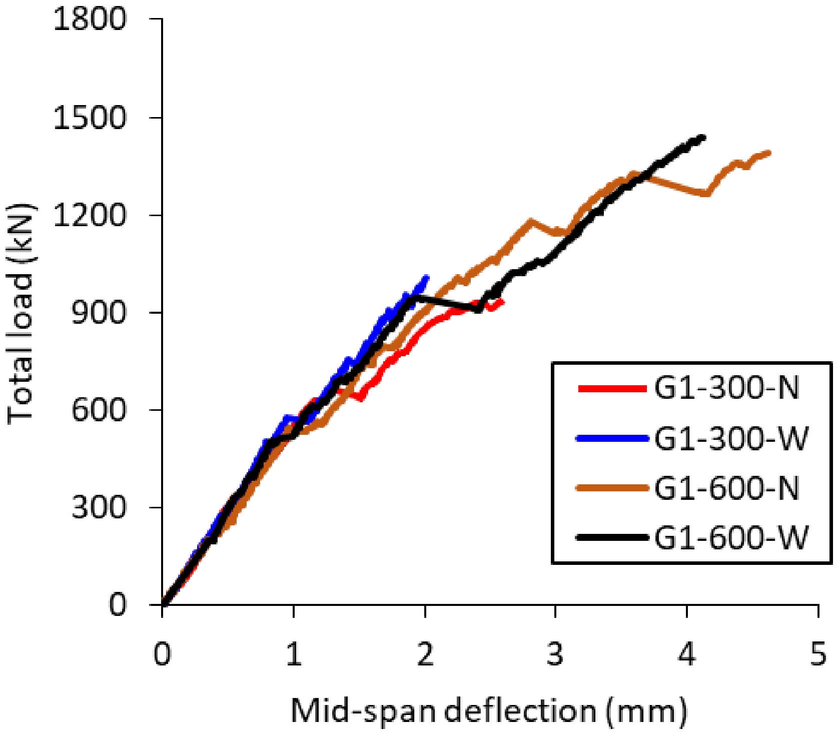
Test setup of deep beams

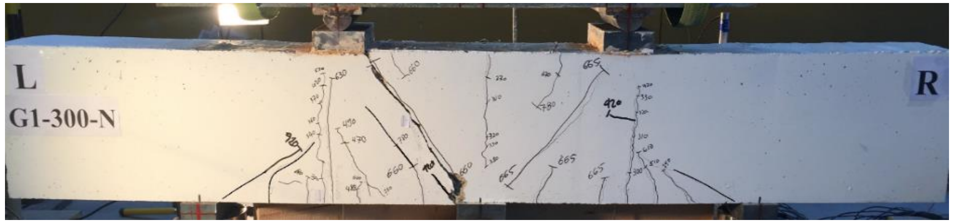


Test setup of pullout tests

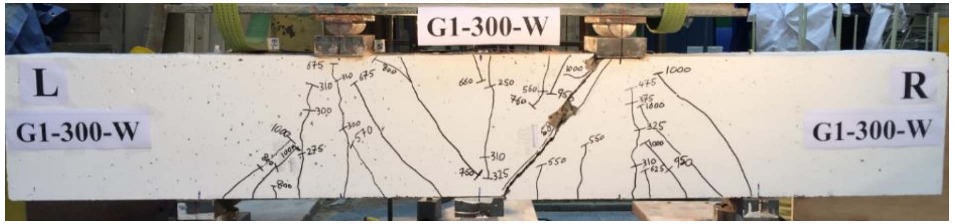
EXPERIMENTAL RESULTS

- **Load-deflection and the failure mode**





G1-300-N



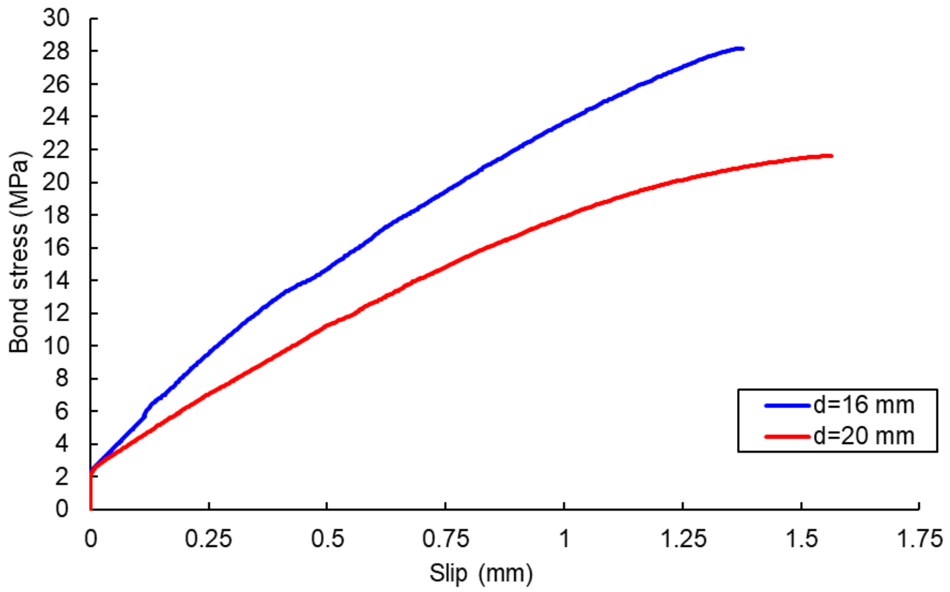
G1-300-W

G1-600-N

G1-600-W

EXPERIMENTAL RESULTS

- **Bond results**



Bond stress-slip

Splitting failure

FINITE ELEMENT ANALYSIS

Why Finite Element analysis is used?

- Experimental works are time consuming and expensive
- FE gives a comprehensive analysis at any point in the structural member rather than relying on a fixed number of points as that in the experimental study.

Limitations of Finite Element analysis

- A 2D finite element analysis using the ABAQUS/Explicit approach was carried out to develop a model that can predict the behaviour of GFRP reinforced continuous concrete deep beams.
- In order to apply the explicit analysis to the static problem, a quasi-static analysis is achieved by applying the load in steps with an amplitude function to keep the ratio of the kinetic energy to the internal energy (ALLKE/ALLIE) less than 5%.

FINITE ELEMENT ANALYSIS

Constitutive models for materials

- Concrete damage plasticity model (CDPM) was used to model concrete
- The elastic modulus and Poisson’s ratio were used to define the elastic behaviour of concrete and GFRP bars
- Hognestad formula was used to define concrete in compression

$$\sigma_c = f_c' \left[2\left(\frac{\varepsilon_c}{\varepsilon_o}\right) - \left(\frac{\varepsilon_c}{\varepsilon_o}\right)^2 \right]$$

$$\varepsilon_o = 2\, f_c' / E_c$$

$$\varepsilon_c^{in} = \varepsilon_c - \frac{\sigma_c}{E_c}$$

- A linear tensile stress-strain relationship was adopted to characterise the tensile behaviour of concrete
- Due to the brittle behaviour and the absence of the yielding response of FRP bars, the plastic part was defined by specifying negligible values of inelastic strain beyond the rupture

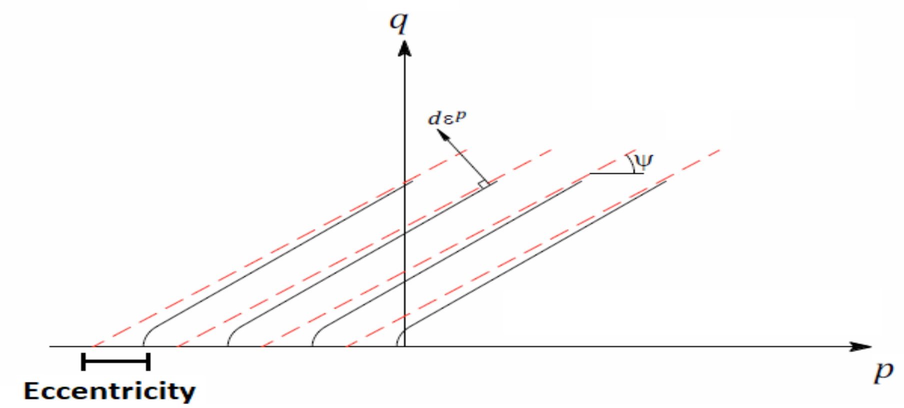
FINITE ELEMENT ANALYSIS

➢ Plasticity parameters and mesh size

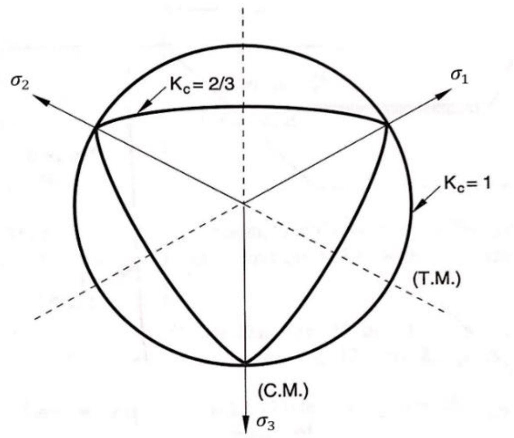
- A sensitivity analysis was performed to select the proper mesh size and the dilation angle required to achieve accurate results compared with the experimental ones

Plasticity parameters used for concrete damaged plasticity model

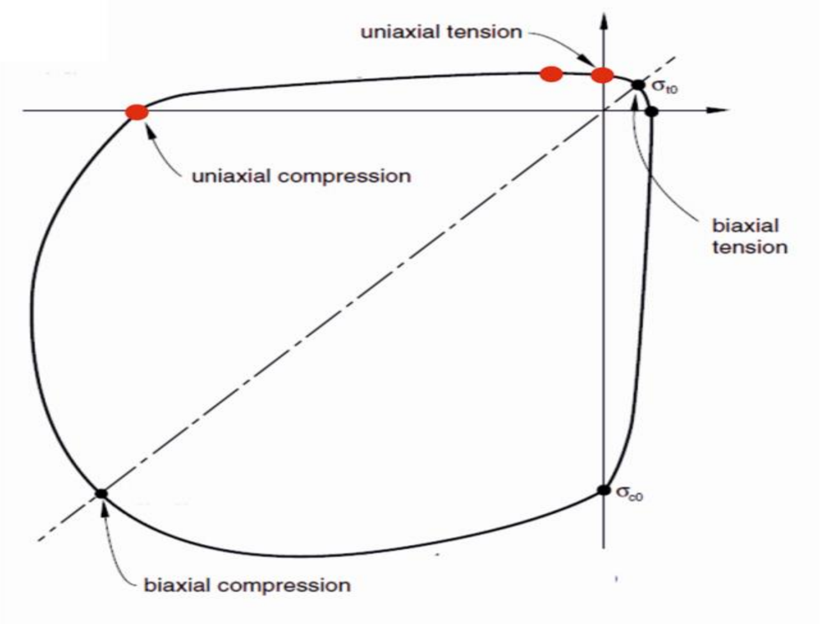
dilation angle (ψ)	eccentricity (ϵ)	σ_{bo}/σ_{co}	K_c	viscosity parameter (μ)
50 (calibrated)	0.1 (default)	1.16 (default)	2/3 (default)	0 (default for explicit analysis)



Dilation angle and eccentricity in the meridian plane



Yield surfaces in the deviatoric plane



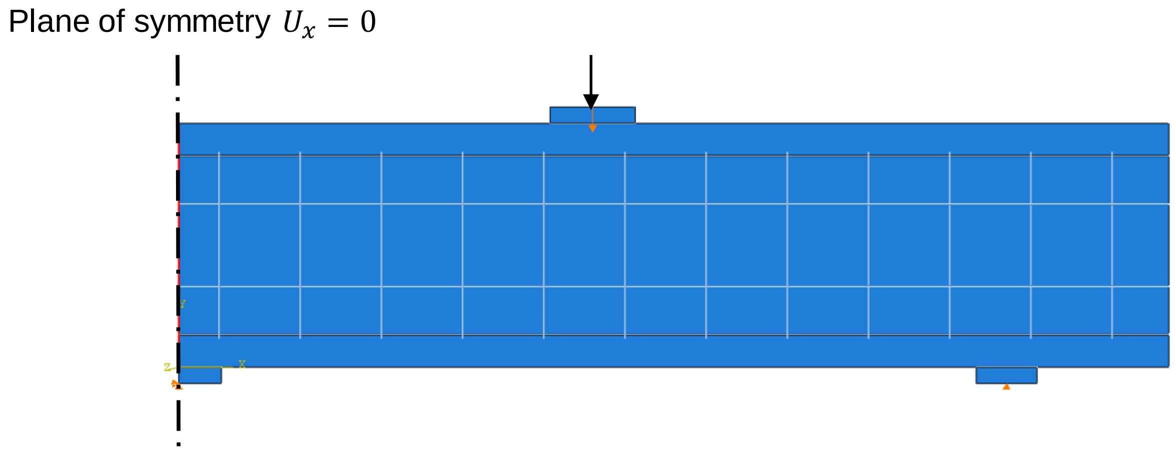
Yield function in the stress plane

FINITE ELEMENT ANALYSIS

➢ Model geometry and boundary conditions

- The symmetry about the centre line of the mid-support for concrete geometry, boundary conditions, reinforcement and the loading arrangement was utilised to model half of the system.

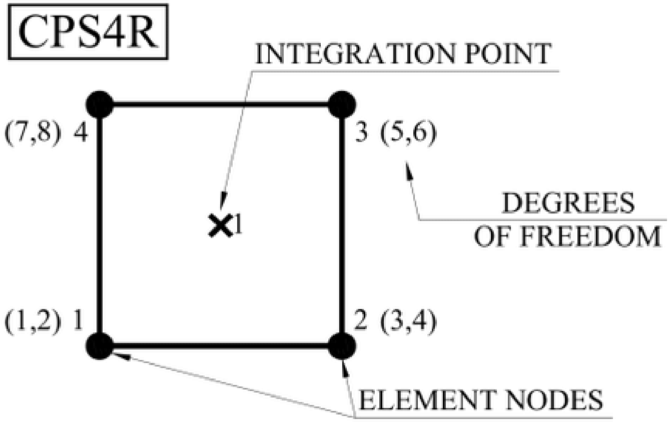
- The middle support was assigned as a hinge to allow rotation only by restricting the horizontal and vertical movements, while the end-support was modelled as a roller by restricting the vertical movement and allowing rotation and horizontal movements.



FINITE ELEMENT ANALYSIS

Element types

- The concrete and bearing plates in the proposed finite element model were modelled as linear plane stress elements (CPS4R) with a reduced integration element, having a quadrilateral shape.

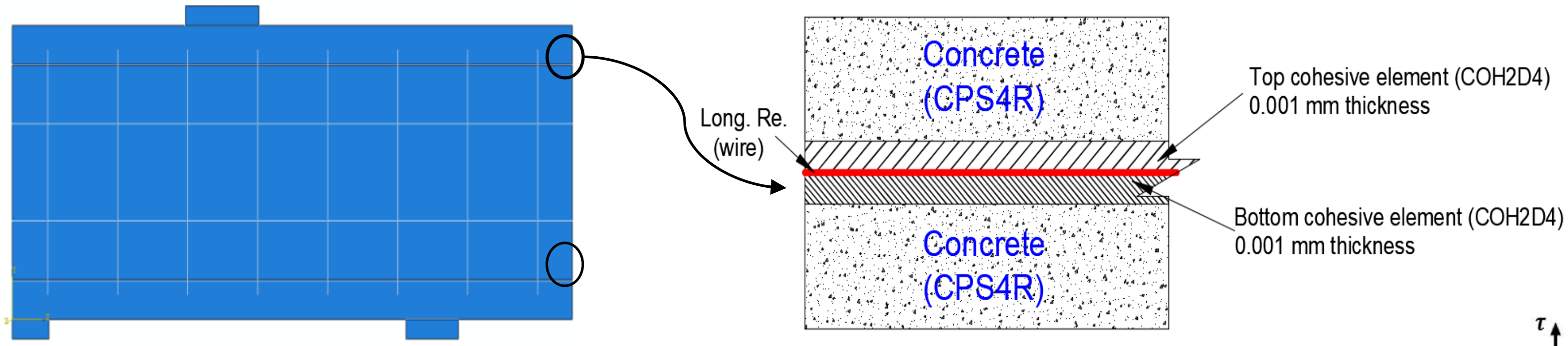


- Longitudinal and web reinforcements were modelled using a 2-node truss element (T2D2).

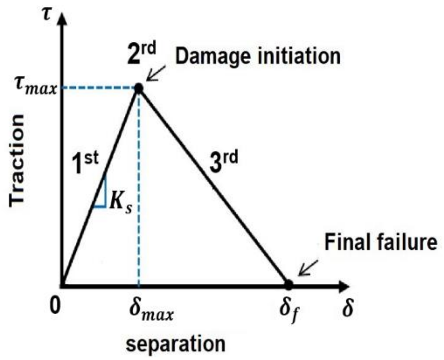
FINITE ELEMENT ANALYSIS

➢ Interaction between Concrete and FRP bars

Using pullout results, the bond between the longitudinal reinforcement and concrete surface was modelled using a cohesive element (COH2D4) tool available in ABAQUS. Furthermore, a perfect bond between the longitudinal reinforcement and surrounding concrete was also modelled to evaluate the validity of this assumption introduced by many previous FE studies.



Cohesive element simulation in ABAQUS



Bilinear traction-separation law used for the cohesive element model

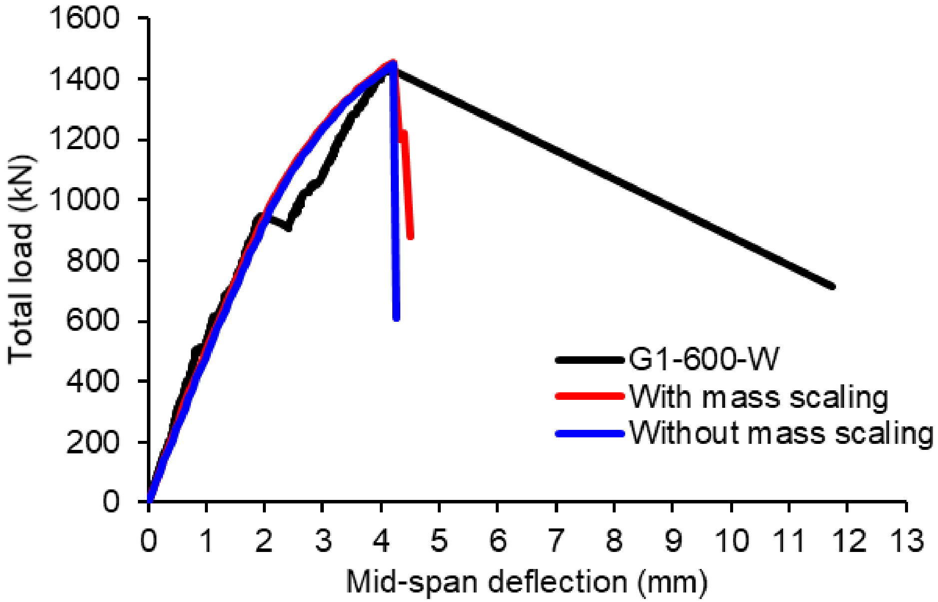
Bond values used for bilinear traction-separation law

Bar diameter (mm)	K_s (N/mm^3)	Damage initiation (bond strength) τ_{max} (MPa)	Damage evolution δ_f (mm)
16	20.4	28.18	5.5
20	13.8	21.61	5.8

FINITE ELEMENT ANALYSIS

➢ Mass scale

For computational efficiency and to speed up the running time without affecting the accuracy of the analysis, a mass scaling was used with a fixed time increment equal to 5e-5 in all regions of the model.



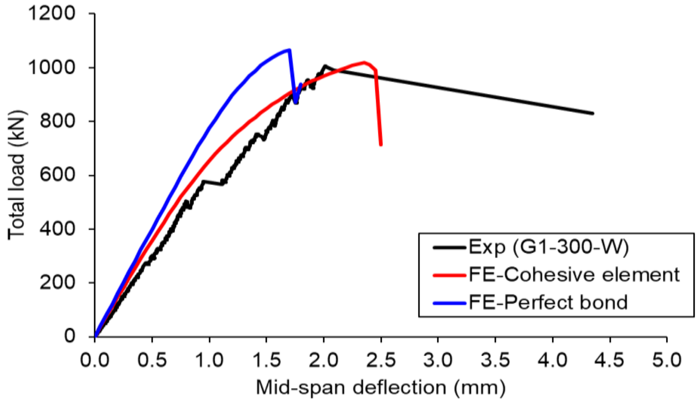
$$\Delta_t = \frac{l_{cr}}{s_{sw}}$$

$$s_{sw} = \sqrt{\frac{E}{\rho_e}}$$

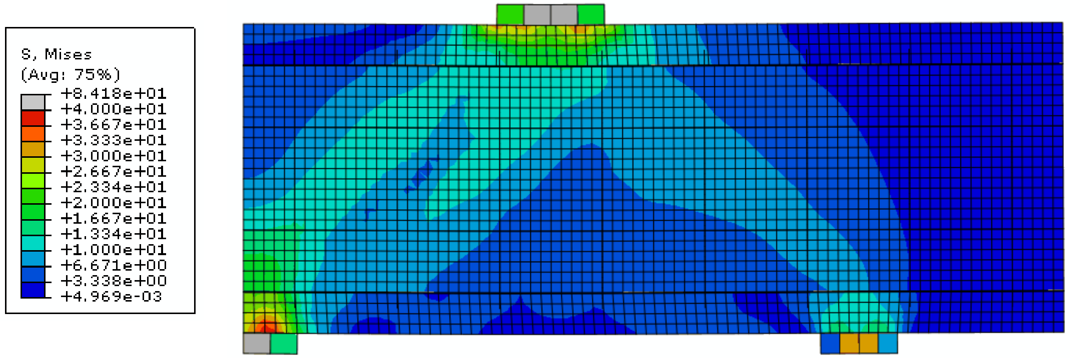
- Δ_t is the stable time step
- l_{cr} is the characteristic element dimension
- s_{sw} is the speed of the stress wave
- E is the modulus of elasticity of the material
- ρ_e is the density of the material

FINITE ELEMENT RESULTS

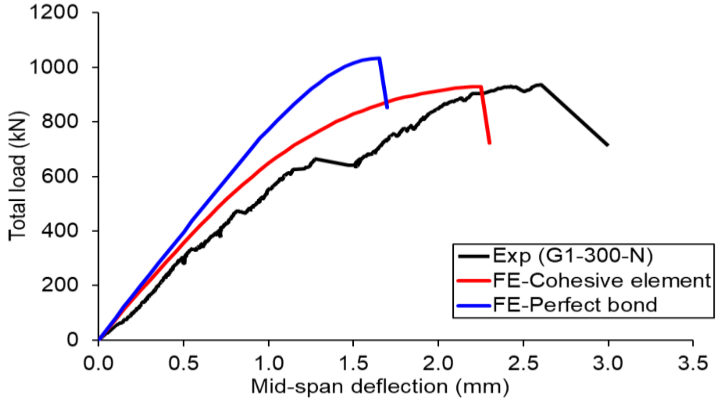
- **Mode of failure and the load-deflection behaviour**



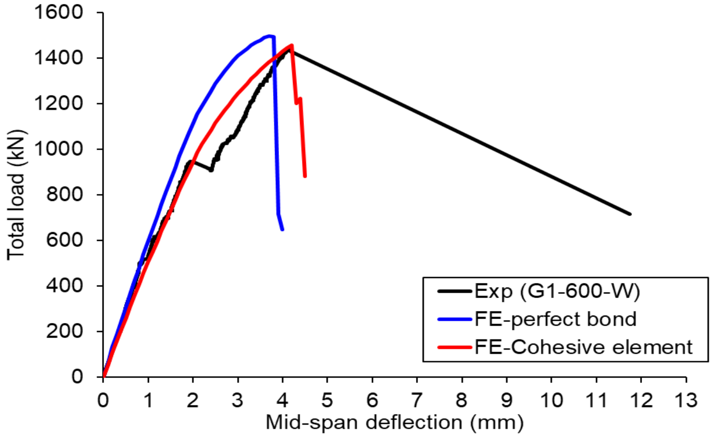
G1-300-W



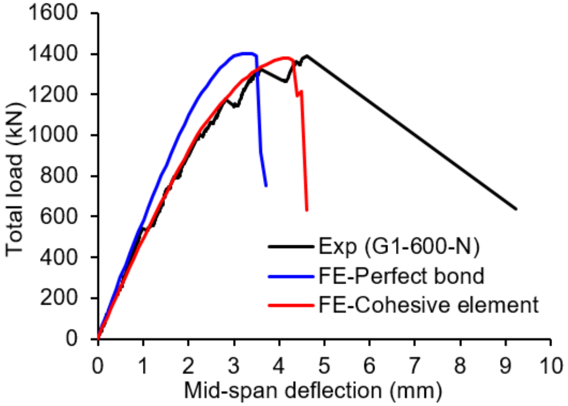
Stress distribution at an ultimate load of beam G1-600-W



G1-300-N



G1-600-W

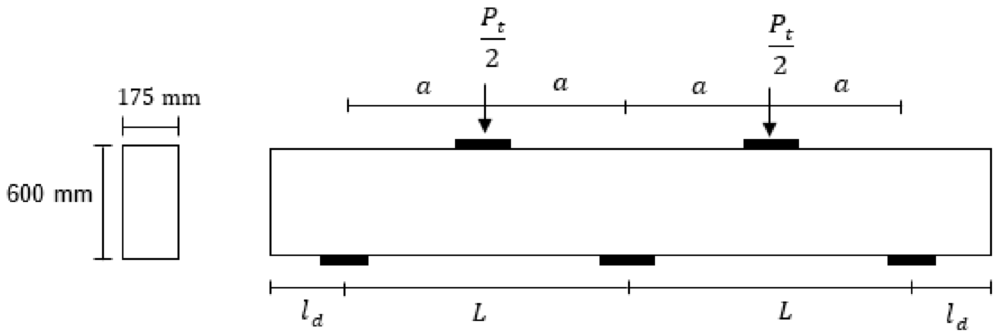


G1-600-N

FINITE ELEMENT RESULTS

➢ Parametric study

The validated model of cohesive element was then employed to implement a parametric study for the critical parameters that govern the behaviour of beams tested. The investigated parameters included a shear span-to-overall depth ratio, concrete compressive strength, web reinforcement, FRP reinforcement type, and longitudinal reinforcement.



Parameters investigated in the parametric study

Investigated parameters	values
a/h ratio	0.5, 1.0, 1.5, 1.7 and 2.0
Compressive strength	30, 40, 50, 60 and 70 MPa
Orthogonal web reinforcement	0, 0.4% and 0.8% (in each direction)
FRP type	GFRP and CFRP
Longitudinal reinforcement ratio	0.6%, 1.2% and 1.8%

Geometrical dimensions used in the current parametric study

a/h	a (mm)	L (mm)	h (mm)	b (mm)	l_d (mm)
0.5	300	600	600	175	400
1.0	600	1200			
1.5	900	1800			
1.7	1020	2040			
2.0	1200	2400			

MAIN CONCLUSIONS

- In terms of the load-deflection behaviour and mode of failure the developed FE model achieved a satisfactory agreement with the test results of GFRP reinforced continuous concrete deep beams.
- Modelling the interface between FRP bars and surrounding concrete is not essential for load capacity predictions, however, its important for other investigations, such as deflection
- The parametric study showed that the higher the a/h ratio the more pronounced the effect of web and the longitudinal reinforcements and the lower the effect of concrete compressive strength; and vice versa when a/h ratio reduces
- The parametric study showed that the higher the compressive strength the lower the contribution of web reinforcement to enhance the load capacity.

Thank you for listening