Kafrelsheikh University
Faculty Of Engineering
Department Of Civil Engineering
Forth Year, Computerized Structural Analysis (CES4227)

Date 10/06/2019
Time Allowed: 3 Hours
Full Mark: 60 (30+10+10+10)
2 pages

ANSWER AS MUCH AS YOU CAN

يسمح للطلاب بالدخول بالمصفوفات إلى اللجنة

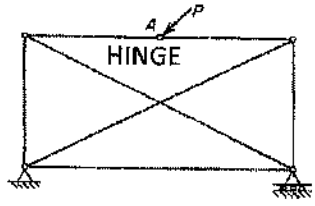
(ILOS a-1, a-3, b-1, b-2, c-2, c-6)

برجاء الحل بترتيب الأسئلة وتنظيم الإجابة

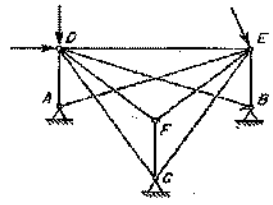
الإمتحان من خمسة ورقات ويسلم الطالب الورقة الأخيرة من الامتحان مع كراسة الإجابة فهى مكملة لها

For all questions (E =20000000 t/m^2, G= 8000000 t/m^2, $\alpha = 1*10^{-5}$ / °c).

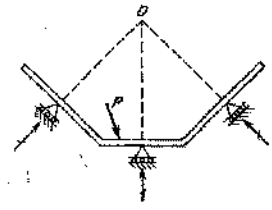
Q1) a- Classify each of the structures shown as stable or unstable. If the structure is stable, then determine the degree of static and kinematic indeterminacy and dimensions of matrices K11, K12, K21, and K22. *إجبارى على الطالب عمل جدول بعد حل هذا الجزء ليلخص فيه الإجابه*



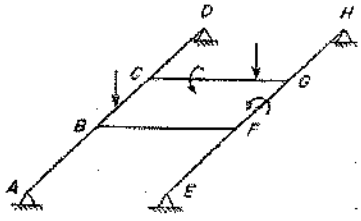
1. PLANE TRUSS



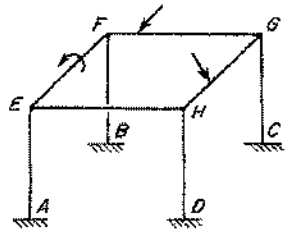
2. SPACE TRUSS



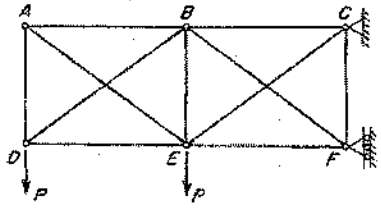
3. PLANE FRAME



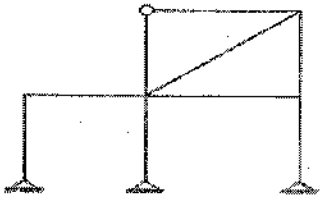
4. GRID



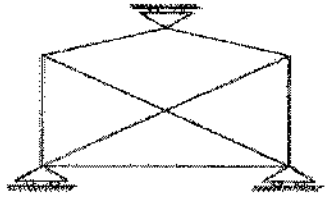
5. SPACE FRAME



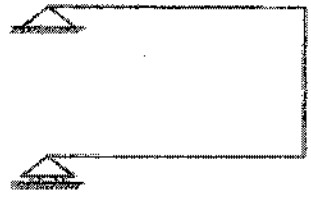
6. PLANE TRUSS



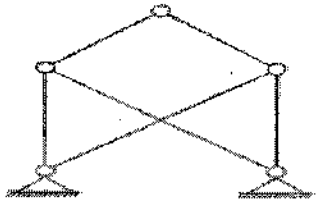
7. PLANE FRAME



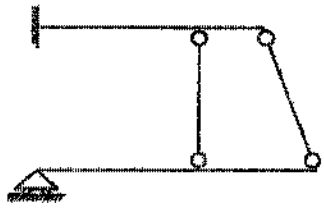
8. PLANE TRUSS



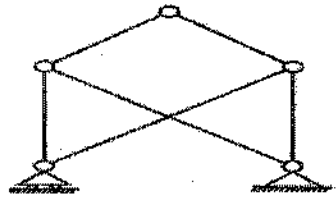
9. PLANE FRAME



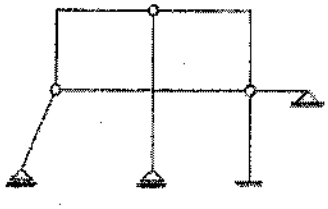
10. PLANE TRUSS



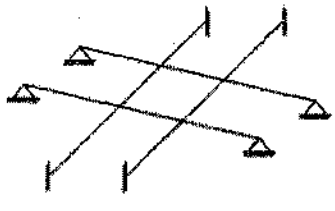
11. PLANE FRAME



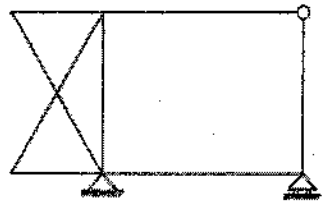
12. PLANE TRUSS



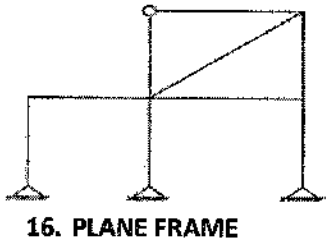
13. PLANE FRAME



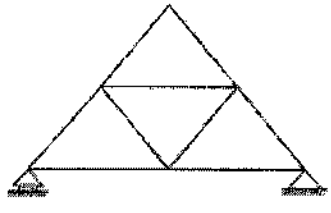
14. GRID



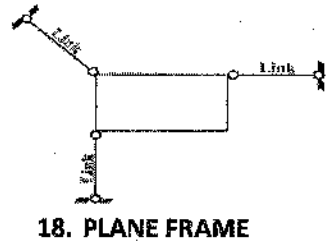
15. PLANE FRAME



16. PLANE FRAME



17. PLANE TRUSS



18. PLANE FRAME

b- In program SAP2000, Choose the correct answer

1. To define the units, you can select (..... → New Model).

a – File	b - Edit	c - Define	d - Assign

2. When you select (Define → Materials →.....), Two materials automatically were defined, their names are:

a – 4000Psi, A992Fy50	b - f_c 5000Psi, ASTM A992Fy50	c – EN C12/15 , ASTM A36	d - Otherwise

3. To define material EN C12/15 which exist in SAP2000 library you can select (Define → Materials → .....).

a – Add new material quick	b – Add new material	c – Add copy of material	d - Otherwise

4. To define 0.3m*0.6m beam section you can select (Define → Section Properties →.....)

a – Frame Sections	b – Tendon Sections	c – Cable Sections	d – Area Sections

5. To create grid system you can select (... → Coordinate System/Grids → Modify the global system).

a – File	b - Edit	C - Define	d - Assign

6. The grids may displayed as.....

a – Ordinates and spacing	b – Relative and absolute	c – a and b	d - Otherwise

7. To change the self-weight multiplier of DEAD LOAD you can select (DEFINE → ..... and modify it).

a – Load Patterns	b – Load Cases	c – Load Combinations	d - Otherwise

8. To get a design case equal 1.4DL+1.6LL after definition of DL and LL you can select (DEFINE → Load Combinations →.....).

a – Linear Add	b – Envelope	c – Absolute Add	d - SRSS

9. To make an intermediate hinge at the end of frame element you can select (Select the frame element → Assign → Frame →.....).

a - Frame Sections	b – Property Modifiers	c – Releases	d – Insertion Point

10. To make a support you can select (Select the support joint → Assign → Joint → .....).

a - Restrains	b – Constrains	c – Springs	d – Masses

11. To make an inclination to support (Select the support joint → ..... → Joint → local axes)

a – File	b - Edit	c - Select	d - Assign

12. To make a spring at joint (select the joint → .....→ JOINT → Springs).

a – File	b - Edit	c - Select	d - Assign

13. To reduce degrees of freedom of framed structures (Analyze → ..... → Check on XZ plane button).

a – Set analysis option	b – Create Analysis Model	c – Run Analysis	d – Model Alive

14. To run the model. Press (..... on keyboard).

a – F1	b – F3	c – F5	d – F7

15. To show the elastic line of beam (.....→ Show Deformed Shape).

a – View	b - Draw	c - Analyze	d - Display

16. To show reactions of structures (→ Show Forces/Stresses → Joints).

a – View	b - Draw	c - Analyze	d - Display

17. To show B.M.D due to in plane loading for framed structures (DISPLAY → SHOW FORCES/STRESSES → FRAMES → Check MOMENT .....).

a – 1-1	b – 2-2	c – 3-3	d – 2-3

18. To show S.F.D due to in plane loading for framed structures (DISPLAY → SHOW FORCES/STRESSES → FRAMES → Check SHEAR.....).

a – 1-1	b – 2-2	c – 3-3	d – 2-3

19. To create grid system you can press right click, then (.....)

a – Edit Grid Data	b – Plane Fine Grid Spacing	c – Edit Dimensions	d – Otherwise

20. SAP2000 uses ..... method in analysis.

a – Finite Element	b – Stiffness Matrix	c – Virtual Work	d – Slope Deflection

c- In program LINPRO, Choose the correct answer

21. LINPRO cannot model ..... structures.

a – Space Truss	b – Space Frame	c – Grid	d – All Previous

22. To define grids you can select

a – Button in the left vertical tool bar	b – Structure	c – Draw	d – View

23. To cancel step you can.....

a – Select pointer in the left vertical tool bar	b – Right click and select pointer	c – a and b	d – Otherwise

24. To define a new cross section you can select (Structure → .....)

a – Member	b – Cross Section	c – Coordinates	d – Section Properties

25. To define a spring you can select (STRUCTURE → .....)

a – Member	b – Cross Section	c – Coordinates	d – Section Properties

26. To define the units, you can select (.....).

a – Button in the bottom horizontal tool bar	b – Structure	c – Draw	d – View

27. LINEPRO uses ..... method in analysis.

a – Finite Element	b – Stiffness Matrix	c – Virtual Work	d – Slope Deflection

28. The temperature units is .....

a – Kelvin	b – Celsius	c – Fahrenheit	d – Otherwise

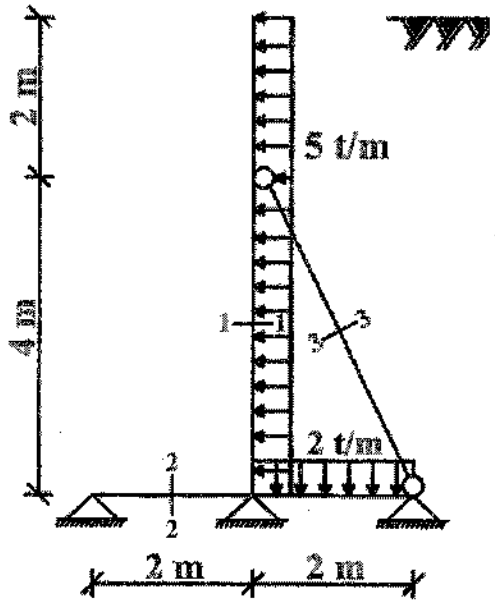
29. To create an intermediate hinge, you can select.....

a – File	b – Structure	c – Draw	d – View

<u>d- Through your study of Stiffness Matrix elements.</u> Shade <u>TRUE</u> if the statement is correct, if you believe otherwise, shade <u>FALSE</u>.

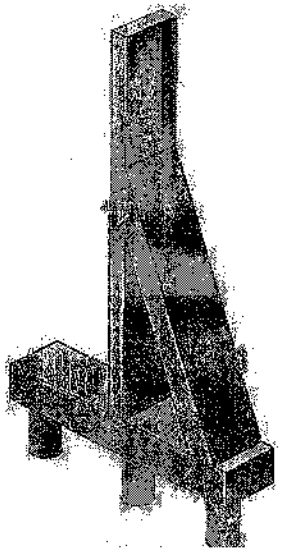
30. The beam element stiffness matrix take the effect of horizontal loads.
31. The straining action of beam element are N.F, S.F and B.M.
32. The straining action of grid element is N.F., S.F.D and T.M.D only.
33. The displacements of beam element are vertical displacement and rotation.
34. The bar element resists any types of moments.
35. The three hinged frame is unstable structure.
36. Stiffness matrix method uses to solve 2d structures only.
37. The paneled beam can be considered as grid element.
38. The member stiffness matrix of bar is [3x3].
39. The member stiffness matrix of 2d truss and beam is [4x4].
40. The member stiffness matrix of 3d frame and 3d truss are [12x12] and [6x6] respectively.
41. The member stiffness matrix of rectangular membrane area element is [9x9].
42. The member stiffness matrix of triangular plate area element is [12x12].

Q2) For counterfort wall shown in Figure 1, using stiffness matrix method it is required to make a complete analysis then draw N.F.D, S.F.D and B.M.D.



Trussed-Frame Modeling

Figure 1



E = 2000 t/cm²	
Sec.1	1.0 * 0.3
Sec.2	1.0 * 0.6
Sec.3	1.0 * 0.5

Q3) For the shown grid system if support C settled 0.5 cm and member BC suffered from variation in temperature as shown. Calculate S.F.D, T.M.D and B.M.D (Take $I=2*10^{-4}$ m^4, $J= 5*10^{-5}$ m^4). sellemnt

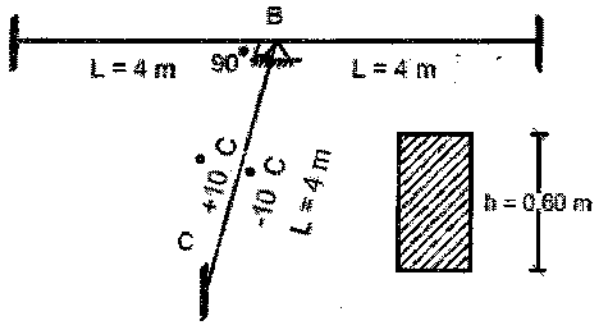


Figure 2

Q4) For truss shown in Figure 3, if A1 = 50 cm^2, A2 = 100 cm^2, K_s =1000 t/m. Find the nodal displacement only. (Due to the given external loads + Temperature + Settlement)

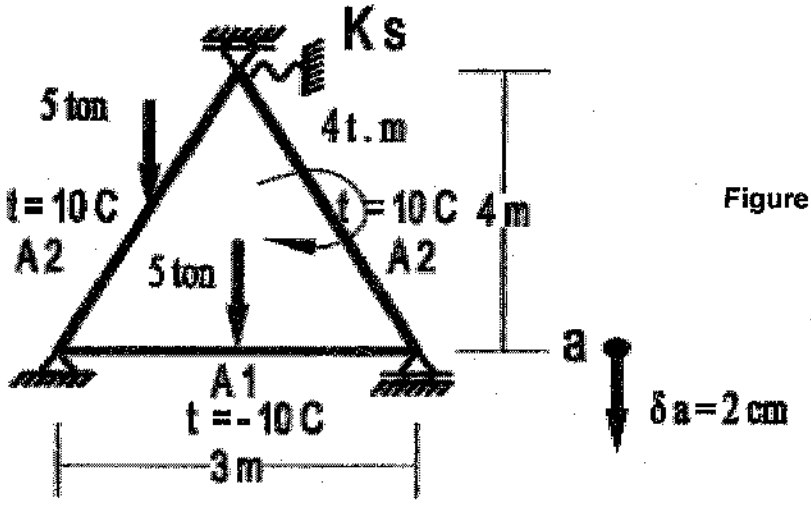


Figure 3

Kafrelsheikh University
Faculty Of Engineering
Department Of Civil Engineering

Forth Year, Computerized Structural Analysis (CES4227)
Date 10/06/2019
Time Allowed: 3 Hours
كنترول الفرقة الرابعة

إسم الطالب/	الرقم السري / خاص بالكنترول
رقم الجلوس/	

إقطع من هنا.....

1	a	b	c	d	31	T	F
2	a	b	c	d	32	T	F
3	a	b	c	d	33	T	F
4	a	b	c	d	34	T	F
5	a	b	c	d	35	T	F
6	a	b	c	d	36	T	F
7	a	b	c	d	37	T	F
8	a	b	c	d	38	T	F
9	a	b	c	d	39	T	F
10	a	b	c	d	40	T	F
11	a	b	c	d	41	T	F
12	a	b	c	d	42	T	F
13	a	b	c	d			
14	a	b	c	d			
15	a	b	c	d			
16	a	b	c	d			
17	a	b	c	d			
18	a	b	c	d			
19	a	b	c	d			
20	a	b	c	d			
21	a	b	c	d			
22	a	b	c	d			
23	a	b	c	d			
24	a	b	c	d			
25	a	b	c	d			
26	a	b	c	d			
27	a	b	c	d			
28	a	b	c	d			
29	a	b	c	d			
30	T			F			