ACES ~ Bridge modelling made easy

1. Create the structure geometry

- Run ACES6 and click the *Start a new project* icon on the main options panel.
- Enter some job identification or simply click *Next >>* to continue.
- Select *Using parametric data entry (templates)* in the dialog box labeled *Create a new model by*
- Click the *Grillages* button in the dialog box labeled *Select a structure category:*
- In the *Select a grillage type* dialog box select *"2. Uniform skew"*
- In the grill dialog box enter values for any of the parameters <u>except</u> *Z-Offset* and *Girder width*
- Check the effect of changing parameter values by clicking the *Verify Geometry* button.
- When satisfied, click *OK* to lock the parameters in and generate the model geometry.

2. Modify the structure geometry

- The large crosses on the structural model indicate support points. If you included top or bottom cantilevers in your model you will need to remove these supports from any edge beams.
- To do this click the *Supports* button in the left-hand menu and select *Delete a support* from the options in the dialog box.
- With the left mouse button click on each support (large cross) that you wish to delete.

3. Assign properties to predefined member types

- Click the *DispMemb* button in the left hand menu. The model will be redrawn with all members highlighted in the colours corresponding to the four member types predefined by ACES.
- Now click the *Mem Prop* button. A dialog box with all currently available member property types will be shown. The first member property type, *[1] Main Girders*, will be highlighted. Click *Edit*.
- Enter property values into the appropriate fields or click *Select section from database* to do so.
- Click the small arrow icon (**>**) to the right of the *Colour* button to select the next member property type and enter its values. Continue this operation until all member types have been completed.

4. Create & assign a new member property type

- To create a new member property type from the properties dialog box click the button labeled *New Type*. Enter properties as before then click *OK* to exit from the member properties dialog box. (If the member properties dialog box is not already open click the *Mem Prop* button and select *New*).
- To assign this property type to a line of members in the model click *Assign Mp*. Click on the line of members you wish to change to the currently set property type.

5. Create a UDL and patch load case

- Click the icon labeled "**F**" on the icon bar. The left-hand menu will change. Click the '*LCase*' button and enter a load case title (e.g. *Dead Load*) in the dialog box that appears. Click *OK*.
- Select the *Memb Lds* button in the left-hand menu then click on any line segment defining a main (longitudinal) girder. A member loads dialog box will appear.
- Enter a *Load value* representing the self-weight of the girder then click *Add load as above*. (Note: The second load value only applies to trapezoidal loads and can be ignored or set to zero).
- Right-click on the remaining longitudinal girders in the model to apply the same loading to them or, if the loading differs, left-click to display the loading dialog box again.

- To add a **patch load,** create a new load case (click *Loads / New Load*) then select the *Patch Lds* button. Define the patch load parameters then click *Add*. Don't worry if the patch is off the model!

6. Create a vehicle load case

- Create a new load case (via *LCase*), click *Vehicles* in the left hand menu then select the required vehicle from the database and click *Open*. A vehicle path data dialog box will appear.
- Enter vehicle path data as required then click *OK* (do <u>not</u> click *Place in lane*). A vehicles dialog box will appear. To apply another vehicle to the same load case click *Add* or to finish click *Close*.
- To change vehicle path details click *Veh Path*, amend the data as required, then click *OK*.
- To view vehicle axle loads and spacings click '*Veh Data*'; to check how the vehicle will actually move along the structure click '*Veh Mvmt*'.
- Note that code loadings (*Code Lds*) and lane loads (*Lane Lds*) can only be added if lanes have first been defined (using the *Def Lanes* button in the left hand *Structures* menu).
- To create a totally new vehicle and add it to the database click '*New Veh*' then follow the prompts.

7. Analyse the model, define an envelope & view the results

- Click the red lightning bolt icon on the menu bar to perform the analysis.
- Select a range of members by either:- clicking the *Set Range* button and pointing to the members for which results are required; OR by using *Results / Select a range of members / Members of specified property type* and selecting *[1] Main girders* from the member types dialog box.
- Click the *Envelope* button in the left hand menu. Enter an envelope title then tick the load cases you wish to include in the envelope. If appropriate, indicate if they are a *Permanent Effect* then enter a *Load Factor* against each. Click *OK*.
- From the left-hand menu click the results you wish to display My, Vz (shear) or Tor (torsion).
- To display the moment diagram for the selected girder range click My. In the left menu bar click the *Options* button. Select *Do not display values* and *Draw on a baseline* then click *Draw*.
- The My diagrams for all members in the currently selected range will be displayed on a single base line and colour-coded to their respective members in the model for easy reference. Click the '+' button to increase the scale of the BM diagram or '-' to decrease the scale.
- Note the girder corresponding to the worst moment diagram. Click the *Clr Range* button in the left menu block followed by *Set Range* then click on the worst loaded girder. Repeat the steps from the 4th dot point above, but this time switch on the display of values in the *Options* dialog box.
- To print the current diagram select *Print / Current diagram* or click the *Print* icon.
- Repeat the above for other vectors. (Note: Dz represents vertical deflections and Fz reactions).
- To interrogate results for a specific vehicle loading case click the button labeled *LCase* (just below *Envelope*), select the loading you require then click *OK*.
- To see where the vehicle lies on the model click *App.Loads*. Use *Options* to change parameters.

8. View & print the results in tabular (report) form

- Click the button labelled *Reports* and indicate if you want results sorted in numeric member order. To save the report as a text file click the *Save* button and enter a file name.
- From the top menu line select *Reports / Moments and shears / Envelopes (Max+Min+Corresp)* then choose either an existing envelope or create a new envelope. Click *OK*. Select the vector for which the maximum value is required (e.g., *My*) and contributing load cases then click *OK*.
- The report will display the maximum and minimum values of the selected vector at the beginning and end of each member together with the corresponding values of all other vectors as well as all loading cases that contributed to that maximum (or minimum) value.