Paramarine Tutorial 5

In this tutorial we will learn how to perform a longitudinal strength analysis of our design. We will then define the structural elements that comprise the ship structure and check if the design satisfies the criteria for maximum stress. Finally, a midship section will be generated. A table with material properties, as well as materials catalogues are provided.

Longitudinal Strength Analysis

A longitudinal weight distribution for the vessel is required in order that a longitudinal strength analysis can be carried out.
The longl_weight_distribution_approx object allows an approximate longitudinal weight distribution to be calculated based on volumes. The method assumes that the main hull and first tier of superstructure are uniformly and homogenously filled with equipment. You specify a point weight defining the required total weight ('total_weight_point'), the bodies defining the main hull and superstructure in 'homogenous_bodies', and any large weight items for which distributions are known in 'discrete_weights'.

1. Insert a stability placeholder and name it long strength
2. Right click on long strength – insert – ship advanced stability - long_weight_distribution_approx
3. Right click on homogenous bodies – insert – body_pointer – name it hull
4. Select for object the hull solid.
5. Follow the same procedure for accommodation and process facility
6. Right click on discrete_weights – insert – weight_linear – name it turret
7. For the weight value set 10.000 tn
8. Define the points, min, max and centroid.
9. At the total_weight_point set the final value of weight to be 73.000 tn. This includes the weight of the homogenous bodies and the discrete weights.
10. Now that you have completed this step go back to the basic_ship – expand – right click on datum – operations – copy of – total_weight_distributed
11. Also link the centroid

In order to graphically visualise the longitudinal weight distribution use a weight_distribution_visualiser

1. Under the strength placeholder – insert – Miscellaneous - weight_distribution_visualiser
2. Expand the weight_distribution_visualiser – right click on weight – operations – copy of – select as object the total_weight_distributed from the longl_weight_distribution_approx
3. Set the internal_between_tabular_results, ie 10m
4. Double click on distribution_graph to see the result

1. Under the strength placeholder – insert – stability placeholder – name it sagging
2. Right click on sagging – insert – Basic Stability – wave
3. Expand the wave – select trochoidal
4. For the wavelength give a value close to the ship length
5. For the wave_crest give a value of 0
6. For the wave_height give the maximum 100 year wave height (22.8m)
7. Set the strips_per_wavelength to 20
8. Under the sagging placeholder – insert – Basic Stability – GZ
9. Link the stability settings to the default
10. Link the loading condition to full
11. Link the wave to the wave_sag
12. Set a range for heel range
13. Under the sagging placeholder – insert – Basic Stability – GZ_visualizer – link to GZ
14. Follow the same procedure for hogging

1. Under the strength placeholder – insert – Ship Advanced Stability - longl_bending_load_wave
2. Expand the longl_bending_load_wave
3. Link the still_water_definition to the full condition
4. Link the hogging_GZ_visualization to the GZ_visualizer under the hoggin
5. Do the same for sagging
6. Give values for wave period and days at sea
7. Double click on load, shear force and bending moment distribution graphs to see the results

Structural Definition

1. Insert a structure placeholder and name it structures
2. Under the structures – insert – struct_placeholder – name it materials
3. Under materials – insert – Structures Setup – material – name it steel Grade A
4. Expand steel Grade A and give values for basic properties like yield stress, Young's modulus, etc.
5. Under the structures – insert – struct_placeholder – name it tolerances
6. Under tolerances – insert – Structures Setup – plate_tolerance – name it – expand and give values
7. Do the same thing and add a stiffener_tolerance and a fabrication_tolerance
8. You can have the tolerances set to zero
9. Under the structures – insert – Structures Setup – stiffener_profile_holder
10. Right click on stiffener_profile_holder – insert - Stiffener_profile_built_up – name it for example: T250x100
11. Expand it to make it valid. Select a profile type, L or T. go to tolerance, double click and link it with the stiffener_tolerance defined earlier
12. Expand the design data and give values
13. Double click the section outline to see an image of the stiffener
14. Under the structures – insert – struct_placeholder – name it specifications

15. Under the specifications – insert – struct_placeholder – name it plate specifications
16. Under plate specifications – insert – Structures Setup – plate_specification
17. Expand the plate_specification and link the material and tolerance with the ones we have defined earlier
18. Give a design thickness
19. Follow the same procedure and add stiffener specifications placeholder under the specifications placeholder. Then add a stiffener_specification object. Link the profile with one of the stiffener profiles we have defined earlier and also link with the material.
20. You can add more specifications
21. Under the structures – insert – struct_placeholder – name it stiffener schemas
22. Under plate specifications – insert – Structures Setup - Stiffener_schema_frames – name it frames
23. Expand the frames – double click on the frame_generator – link it with the transverse bulkheads you have created earlier
24. Double click on the default_specification and link it with one of the stiffener specifications you have defined earlier
25. Expand the stiffeners folder to see the stiffeners
26. Under plate specifications – insert – Structures Setup - Stiffener_schema_planar – name it planar
27. Expand and select xz_stiffeners. For the alignment select keepi_in_plane.
28. Double click on the default_specification and link it with one of the stiffener specifications you have defined earlier
29. Right click on locations – insert – a-b for_next. Give the location of the stiffeners. Start at –beam to +beam, select increments
30. Expand the stiffeners folder to see the stiffeners
31. Under the structures – insert – struct_placeholder – name it scantlings
32. Under scantlings – insert – Structures Setup – scantling definition
33. Expand and link the fabrication_tolerances and the plate_specification with the ones defined earlier.
34. Right click on the stiffener_schemas folder – insert – name planar – in the pop-up window link with the planar xz_stiffeners
35. You should add more stiffener schemas, if you want to have large and small stiffeners on the same plate.
36. You can create more scantling definitions
37. Under the structures – insert – struct_placeholder – name it panels
38. Under panels – insert – Structures Setup – panel_generator_bodies
39. Right click on panel_generator_bodies – insert – name hull – choose the hull solid as an object in the pop-up window
40. Under panels – insert – Structures Setup – panel_generator
41. Expand and link the top_level_solids with the panel_generator_bodies
42. Expand the generated_panels folder
43. You will notice that the ship consists of 7 subfolders-panels.
44. Expand the port_hull subfolder and link the scantlings to the scantlings definition
45. You have to do that for all subfolders
46. Expand the stiffeners folder to see the results
47. Under the structures – insert – struct_placeholder – name it definitions

48. Under definitions – insert – Structures Setup – structural_definition
49. Expand – go to automatic_panles – linkn with the panel_genrator
50. Under definitions – insert – Structures Setup – enhanced_structural_definition
51. Expand and link to the structural_definition
52. Under the structures – insert – Structures Analysis – critical_section
53. Expand the critical_section and link with the enhanced_structural_definition
54. Double click on loadings and link it with the longl_bending_load_wave object
55. Give an x location for the section. Make sure you use the location of the maximum bending moment
56. Check the results
57. Double click on picture to see the midship section
58. For further reading go to the structural definition help file

To create a more detailed midship section you need to do the following. Go back to the geometry placeholder and under the hull placeholder you will find the solid which defines the hull. Remember this is a copy of the hull solid that we created with intellihull, so you don't want to change the original one.
Select the copy of the hull – click on operations – select subdivide – it will ask you for a division bound – choose one of the planes (decks, long or transverse bkhds) we had created before
As an example select the double bottom. Expand the hull solid and you will see that two new solids are created, one for the upper hull and one for the lower hull.
Do the same with the rest of the planes.
Now go to the structures folder – panels folder – panel_generator_bodies – right click – insert – point to the new sub-solids we just created.
Now if you go to the panel generator, you will see that the generated panels are more than the ones we had before.
Assign scantlings and check your midship section again. It should look like this: