FS2000 Demonstration Tutorials

The tutorial listed here are grouped in broad subject categories. They are contained in zip files which includes the tutorial text in pdf format and the associated FS2000 models. The zip files are in the user's FS2000\Tutorials folder. The pdfs can also be accessed using the links in this document.

BASIC OPERATION

T1_FS2000_GUI_INTRODUCTION. https://www.aes-uk.com/tutpdfs/T1_FS2000_GUI_Introduction.pdf

The object of this tutorial is to make new users aware of the more commonly used interactive controls used in the FS2000's GUI

T2_BASIC_OPERATION_SIMPLE_MODEL
https://www.aes-uk.com/tutpdfs/T2_BasicOperation_Simple_Model.pdf

The object of this tutorial is to introduce new users to how FS2000 is structured in terms of its basic operation. This is achieved by creating and running a very simple model.

T3_BATCH_OPERATION https://www.aes-uk.com/tutpdfs/T3_Batch_Operation.pdf

Once the model and load cases are created, they are solved and resulting output is created to suit the particular requirements of the job. The object of this tutorial is to introduce Batch Operation, in which simple script commands are used to control the running of the model and eliminate the requirement for repetitive user interaction.

T4_COLLATING_MODEL_DATA https://www.aes-uk.com/tutpdfs/T4_Collating_Model_Data.pdf

The object of this tutorial is to show how the formatted input and output data can be selected to form sequentially page numbered output reports that can be sent to a printer or 'print to' file.

T5_MULTIPLE_RESULTS OUTPUT https://www.aes-uk.com/tutpdfs/T5_Multiple_Results_Output.pdf

The object of this tutorial is to show how the formatted standard output data for different result cases can be contained in single multi-case output file.

GENERAL MODELING FEATURES

COUPLES_TUT https://www.aes-uk.com/tutpdfs/Couples.pdf

This tutorial demonstrates the use of couples by showing typical applications of FS2000's couple elements. In its simplest form a couple element is a spring connection between the freedoms of two nodes or between the freedoms of a single node and ground. Couple elements are very versatile, and their use is often invaluable when modelling many physical systems.

FE-SOLID_TUT https://www.aes-uk.com/tutpdfs/Solid-Finite Elements.pdf

The object of this tutorial is to provide guidance on the use FS2000's finite elements (non-beam elements) in common areas of application. It is hoped that if a user is not that familiar with FE, this will give some useful tips and highlight the important features of an FE model and its elements.

Which elements to be used, element shape and mesh quality are addressed.

It also addresses the use of coordinate systems and stress averaging used in stress output post-processing, a very important aspect for shell elements.

MESHING_TUT https://www.aes-uk.com/tutpdfs/Meshing.pdf

Illustrates how the solid element meshing commands can be used to create various types of model shapes using coordinate systems etc.

DYNAMIC_ INTERPRETATION https://www.aes-uk.com/tutpdfs/Dynamic_Interpretation.pdf

This tutorial demonstrates the use of FS2000's Dynamic Interpretation capability. Dynamic Interpretation provides the capability to modify a model when using Batch to run a model. This eliminates the requirement to use multiple models for reasons of physical geometry e.g. a moving crane boom.

SIMPLE LIFT ANALYIS PROCEDURE https://www.aes-uk.com/tutpdfs/A Spool Lift Analysis Procedure.pdf

This demonstrates an approach to undertaking lift analysis. A simple model of a pipe section is used to show how to create a stable static model for a single point lift.

STRUCTURAL BUCKLING ANALYSIS

FRAMEBUCKLING_TUT https://www.aes-uk.com/tutpdfs/Buckling_Analysis.pdf

This tutorial demonstrates how FS2000 can be used to undertake linear buckling analysis (Eigen buckling analysis) and non-linear buckling analysis of beam models. The subject model is a simple structure with easily understood buckling characteristics. The tutorial shows how buckling modes can be obtained and plotted and how an effective length factor used for code based design can be established. It also shows how initial displacements (imperfections) for use in non-linear buckling analysis can be obtained from linear Eigen buckling modes.

PLASTIC_ANALYSIS_TUT https://www.aes-uk.com/tutpdfs/Plastic_Analysis.pdf

This tutorial demonstrates elasto-plastic buckling and general plastic analysis.

NON-LINEAR ANALYSIS -DYNOFLEX

DYNOFLEX_TUT https://www.aes-uk.com/tutpdfs/DyNoFlex_Tutorial_1.pdf

FS-DyNoFlex is a general-purpose analysis option of FS2000 for the linear, nonlinear, static and dynamic analysis of structural systems. The objectives of this tutorial example are to show, using a simple model, the basic analysis procedure and typical settings used to obtain a time history solution from FS-DyNoFlex. It also shows how to extract meaningful results from the solution.

LARGEDISPLACEMENT_TUT https://www.aes-uk.com/tutpdfs/LargeDisplacemnt_Tutorial.pdf

This tutorial gives users awareness and an appreciation of the approaches that can be adopted when undertaking large displacement analysis. A large displacement solution is only required to be undertaken in cases where the deflections under load are high enough to change the stiffness of the structure and its resulting equilibrium condition. The tutorial uses example that demonstrate the effect of:

- P-Delta Effects
- Updated Geometry

The tutorial mainly uses DyNoFlex solutions but P-Delta effects using the Standard 3-D Solver is also addressed.

CATERNARY AND PIPELAY ANALYSIS https://www.aes-uk.com/tutpdfs/Catenary_Pipelay.pdf

The object of this tutorial is to demonstrate how FS2000 can be used to analyse structures in which catenary action is dominant. The analysis of catenary action in structures can sometimes be quite difficult and this difficulty is mostly due to establishing the initial strained condition. The examples demonstrate the use of both beam and spar elements in modelling pipelay type problems. Also demonstrated is a utility that is used to establish curvature from a line of string elements.

CFACT_COMMAND_TUT https://www.aes-uk.com/tutpdfs/CFACT_Command.pdf

This tutorial demonstrates the use of the CFACT command. This command enables the stiffness of a couple elements to vary in a time history solution. This command is very useful in simulating sequential loading where the effects of locked in stresses are required to be assessed. It can also be used when sections of models require to be connected or disconnected, e.g. lift analysis, flange make-up. An example of this command is also included in the Large Displacement Tutorial example.

CDISP_COMMAND_TUT https://www.aes-uk.com/tutpdfs/CDISP_Command.pdf

This tutorial demonstrates the use of the CDISP command. The command allows relative displacement between nodal degrees of freedom to be prescribed. The demonstration example evaluates the loading following the connection of misaligned pipeline flanges.

PLASTIC_ANALYSIS_TUT https://www.aes-uk.com/tutpdfs/Plastic_Analysis.pdf

This tutorial demonstrates elasto-plastic buckling solutions using DyNoFlex.

DYNAMIC RESPONSE (MODAL) ANALYSIS

MODAL_RESPONSE_TUT https://www.aes-uk.com/tutpdfs/Modal_Response.pdf

This tutorial demonstrates how FS2000 can be used to undertake linear frequency analysis (Eigen analysis) and how the resulting modes shapes can be used in modal response analysis. The subject model is a simple cantilever beam with easily understood dynamic characteristics. The tutorial shows how vibration modes can be obtained and plotted. It demonstrates how the dynamic response module can be used to obtain both frequency and time history solutions.

PLASTICITY

PLASTIC_ANALYSIS_TUT https://www.aes-uk.com/tutpdfs/Plastic_Analysis.pdf

The object of this tutorial is to introduce users to the features and methods available when FS2000 is used to undertake plastic analysis of beam type structures. It demonstrates how the following methods can be used undertake plastic solutions.

- Frame plasticity (ideal plasticity)
- Stress-Stain relationships (pipe elements only).
- Moment Curvature (M-C) relationships.

A varied range of examples demonstrate how the program can be employed and results obtained. Examples include collapse-ship impact, pipe reeling, post-reeled solutions, tubular denting and energy absorption.

NON-LINEAR PILE FOUNDATIONS

PILE_TUT https://www.aes-uk.com/tutpdfs/PileAnalysis.pdf

This tutorial FS2000 can be used for the analysis of 3-D piled structures using FS-Pile. The emphasis is to demonstrate, using a procedural example, how a non-linear piled foundation can be incorporated into an FS2000 3-D structural model.

WAVE LOADING

WAVELOADING_TUT https://www.aes-uk.com/tutpdfs/WaveLoad.pdf

This tutorial demonstrates how FS2000 can be used to generate hydrodynamic loading on framed structures. The tutorial uses a procedural example based on the loading of an offshore riser.

DYNOFLEX_WAVERAO_TUT https://www.aes-uk.com/tutpdfs/DynamicWave_RAOs.pdf

The object of this tutorial is to demonstrate the use of FS-DyNoFlex to undertake dynamic time history analysis of structural systems subjected to hydrodynamic loading due to ocean wave action. The tutorial covers three basic topics:

- The basic operation of FS-Wave and FS-DyNoFlex for undertaking time history analysis.
- Using RAO data to impart object motion.
- Simulating floating objects.

FATIGUE

FATIGUE_TUT https://www.aes-uk.com/tutpdfs/Fatigue_Tut.pdf

This tutorial demonstrates how FS2000 can be used for fatigue assessments using one or more the following approaches:

- SN (stress life) Endurance Curves.
- SE (strain life) Endurance Curves.
- Fracture Mechanics (BS 7910 methodology).

The objective is to show the stresses obtained from any or all stress points in a FS2000 result case can be processed to give estimates of fatigue damage. The examples used include:

- Deterministic and spectral fatigue of offshore structures.
- Dynamic solutions from DyNoFlex (Rainflow counting).
- Slugging flow in pipework.

PIPELINE ENGINEERING

PIPELINE_TUT https://www.aes-uk.com/tutpdfs/SpoolAnal.pdf

This tutorial demonstrates how FS2000 can be used for the analysis of pipelines. The objective is to show how to model the non-linear interaction between the pipeline and the soil foundation.

PIPELINE_PROPERTIES_UTILITY https://www.aes-uk.com/tutpdfs/Pipeline_Properties_Utility.pdf

The object of this tutorial is to introduce users to the PIPELINE PROPERTIES UTILITY. This very useful utility can be used to generate stiffness, loading properties and complete models for a variety of pipeline analysis scenarios.

UHB_TUT https://www.aes-uk.com/tutpdfs/Upheaval-Buckling.pdf

This tutorial demonstrates how FS2000 can be used to undertake upheaval buckling (UHB) analysis of buried pipelines. The demonstration uses the PIPELINE PROPERTIES UTILITY to set up a complex FS-DyNoFlex non-linear model using very little user input.

PIPELINE_CROSSING https://www.aes-uk.com/tutpdfs/Pipeline_Crossing.pdf

This tutorial demonstrates how FS2000 can be used to undertake the design assessment of pipeline crossings. The demonstration uses the PIPELINE PROPERTIES UTILITY to set up the complex a FS-DyNoFlex non-linear model of a typical crossing.

EXPANSION_WALKING https://www.aes-uk.com/tutpdfs/Expansion_Walking.pdf

This tutorial demonstrates how the PIPELINE PROPERTIES UTILITY can be used to create models to undertake pipeline expansion and pipe walking evaluations. This is achieved using a simple pipeline model.

MOVING_LOAD https://www.aes-uk.com/tutpdfs/Moving_Loads.pdf

The object of this tutorial is to describe and demonstrate the use of FS2000's Moving Load Generator. This enables the structural response of pipework to slugging flow and the assessment of subsequent fatigue damage. The generator uses a batch command line operation to create loading for DyNoFlex time history solutions.

CDISP_COMMAND_TUT https://www.aes-uk.com/tutpdfs/CDISP_Command.pdf

This tutorial demonstrates the use of the CDISP command. The command allows relative displacement between nodal degrees of freedom to be prescribed. The demonstration example evaluates the loading following the connection of misaligned pipeline flanges.

DYNOFLEX_WAVERAO_TUT https://www.aes-uk.com/tutpdfs/DynamicWave_RAOs.pdf

The object of this tutorial is to demonstrate the use of FS-DyNoFlex to undertake dynamic time history analysis of structural systems subjected to hydrodynamic loading due to ocean wave action. The tutorial covers three basic topics:

- The basic operation of FS-Wave and FS-DyNoFlex for undertaking time history analysis.
- Using RAO data to impart object motion.
- Simulating floating objects.

PLASTIC_ANALYSIS_TUT https://www.aes-uk.com/tutpdfs/Plastic_Analysis.pdf

Pipe plasticity analysis, Stress-Strain, Moment Curvature, Pipe reeling and Post-reeling solutions.

BOLTED CLAMP DESIGN

BOLTED_CLAMP_DESIGN https://www.aes-uk.com/tutpdfs/Bolted_Clamp_Design.pdf

The object of this tutorial is to introduce users to a design checker that can be used to assess the design performance of tubular bolted clamps (ISO 19902 & OTH 88 383). The type of clamps that can be checked are those of the type commonly used to form riser connections to offshore jackets and pipework support connections on subsea structures. The design utility will automatically check all clamps in all result cases.