VENECIA

SOFTWARE PACKAGE
FOR THERMOHYDRAULIC ANALYSIS OF
FORCED FLOW COOLED SUPERCONDUCTING MAGNETS
AND THEIR PRIMARY CRYOGENIC SUBSYSTEMS

DESCRIPTION

ALPHYSICA GmbH
Karlsruhe

CONTENTS

INTRODUCTION

From Vincenta to the advanced VENECIA package

The software package **VENECIA** is the next generation of the code Vincenta offering more flexible and general application and significant innovations given rise from 10 years experience of thermal hydraulic simulations for large magnet systems (ITER, KSTAR, JT-60 and validation of code by cryogenic tests of Central Solenoid Model Coil (CSMC) [1]).

Based on the original modelization approach first implemented in Vincenta, **VENECIA** enables numerical simulation of transient behavior of superconducting magnet systems as whole making allowance for:

- real geometry of magnet structure
- real nonlinear properties of materials
- real coolant properties
- real cryogenic accessories.

Efficient and adequate simulation is possible for both “short” transients (stability and quench of conductors etc.) and "long" transients (normal operation, warm up etc.).

01/1D models for the cryogenic circuit and 2D models for thermal diffusion in solids are coupled in a complex model to ensure comprehensive thermohydraulic analysis. The model detalization is variable over a wide range depending on the nature of problem and accuracy required.

VENECIA has benefits of an enhanced modular architecture and database approach. Each module implies a generalized algorithm for different calculation/modeling/control options. Particularly, a complex schematics is implemented for a primary cryogenic circuit that allows different modifications of control and adjustment of cryogenic parameters.
The software is now capable of simulation of up to 5 different fluids simultaneously within a single model. Databases of thermodynamic and kinetic properties of most conventional cryogenic fluids are available in the tabular format. The helium properties database is expanded to supercritical, superfluid and two-phase states. Additional databases can be synthesized from universally recognized data sources, such as CRYOSOFT HEPAK and GASPAK, and integrated into the package.

The analytical core has been substantially revised so as to provide much more efficient and stable calculations:

- the state equations now cover the superfluid He range;
- the two-phase state can be treated as a homogeneous mixture or as a gas/liquid separately, at user’s option;
- simulation of mass forces (gravity, centrifugal force) is enabled
- improved numerical stability and convergence are achieved, primarily in the events of strong non-linearity of the thermodynamic equations in vicinity of the normal zone boundary, where substantial variations of fluid properties are observed.
- Reduced runtime

Due to extended functionality, **VENECIA** offers efficient solution for some intricate aspects of thermohydraulic analysis, including:

- simulations of coolant transients in superconducting magnets and their primary forced flow cryogenic circuit that includes helium circulators and cold vapor compressors for decreasing the operating temperature down to 1.8K.

- stability and normal zone propagation studies, analysis of nominal cooling conditions, and emergency situations similar to quench resulted in realise of large quantity of coolant from the magnets.
- prediction of heat load variations on the cryoplant from magnets, active mitigation of these heat variations and analysis of flow transients at inlets of rotary machines for forced flow cooling of magnets (circulator) and cold compressors for decreasing of operating temperature.

To facilitate modelization and analysis, pre- and postprocessing tools are implemented that allows visualization model linkage, meshing, real-time result monitoring, selection and plotting of outputs. However, the input procedure is kept generally unchanged so that the Vincenta users could run the code in a familiar manner. Vincenta v.6.0 files are supported**.**

VENECIA functionality is steadily expanded and completed so that the code evolves constantly following the practical needs giving the user a powerful tool for solving most challenging thermohydraulic problems in demand.

MINIMUM REQUIREMENTS:

- Intel Pentium IV, 1Gb RAM
- Microsoft Windows XP or higher

APPLICATIONS

VENECIA is applicable for thermohydraulic studies for a wide range of devices including

- thermonuclear facilities,
- accelerators and transport systems,
- MRI-magnets,
- superconducting motors, generators and storage rings,
- experimental and diagnostic devices for scientific research,
- generators,
- superconducting cables and joints.
- Components of magnet primary cryogenic circuits (rotary machine for forced flow cooling, cold vapor compressors, force flow heat exchangers, bath with boiling coolant, long coolant transfer lines and control/relief valves).

VALIDATION

VENECIA results have been compared with existing Vincenta simulations and huge experimental database collected during CSMC tests under pulsed high power heat loads.

Excellent match between numerical and physical results in terms of temperature-pressure-flow at more than 50 principal points of the CSMC cooling circuit has proved that **VENECIA** is suitable for thermohydraulic analysis of complex magnet systems.

VENECIA IN USE FOR ITER

Several global **VENECIA** models were developed for the latest ITER studies on transient thermohydraulic behaviour of the Toroidal Filed (TF) magnet system, Poloidal Field (PF) magnet system, Central Solenoid (CS) magnet system and Correction Coils magnet system with appropriate elements of their primary cryogenic circuits. The models were thoroughly treated for different plasma scenarios and operational modes.

1. ITER Toroidal Field Coil

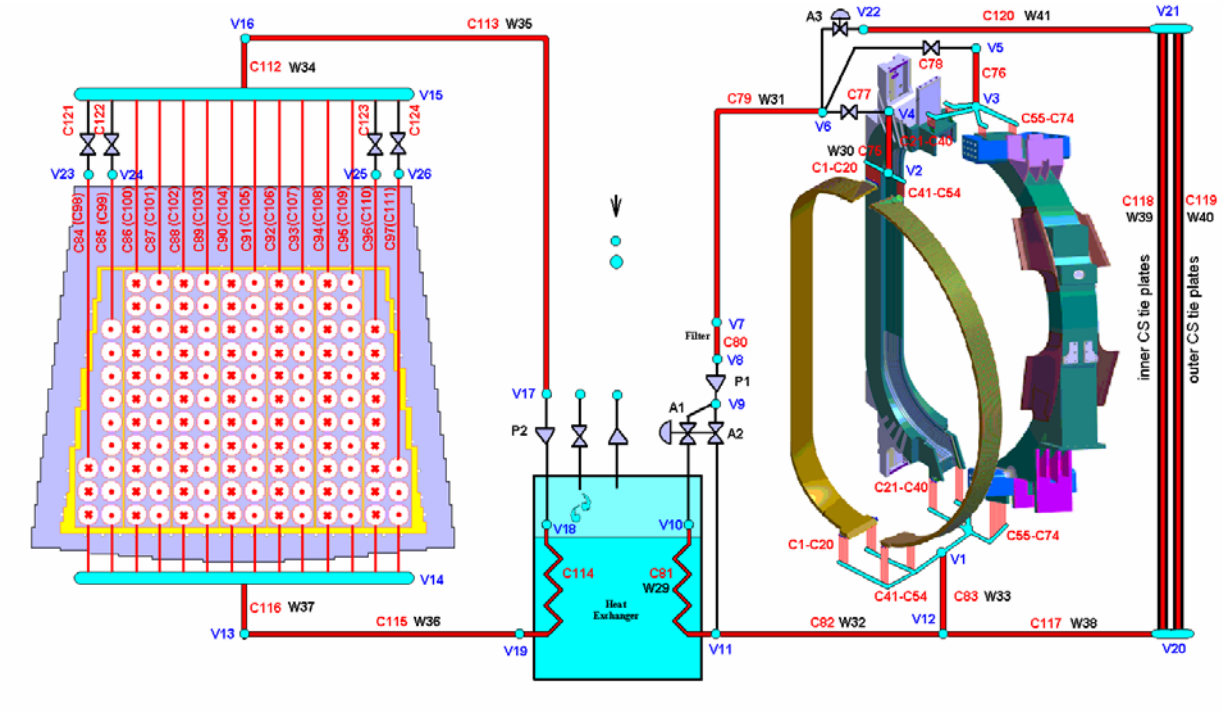
A. Normal operation

The thermohydraulic model of a TF coil allows detailed thermohydraulic analysis of the TFC winding pack and its mechanical structure (case) under different operational conditions including plasma disruption and active flow adjustment for mitigation of pulsed heat load.

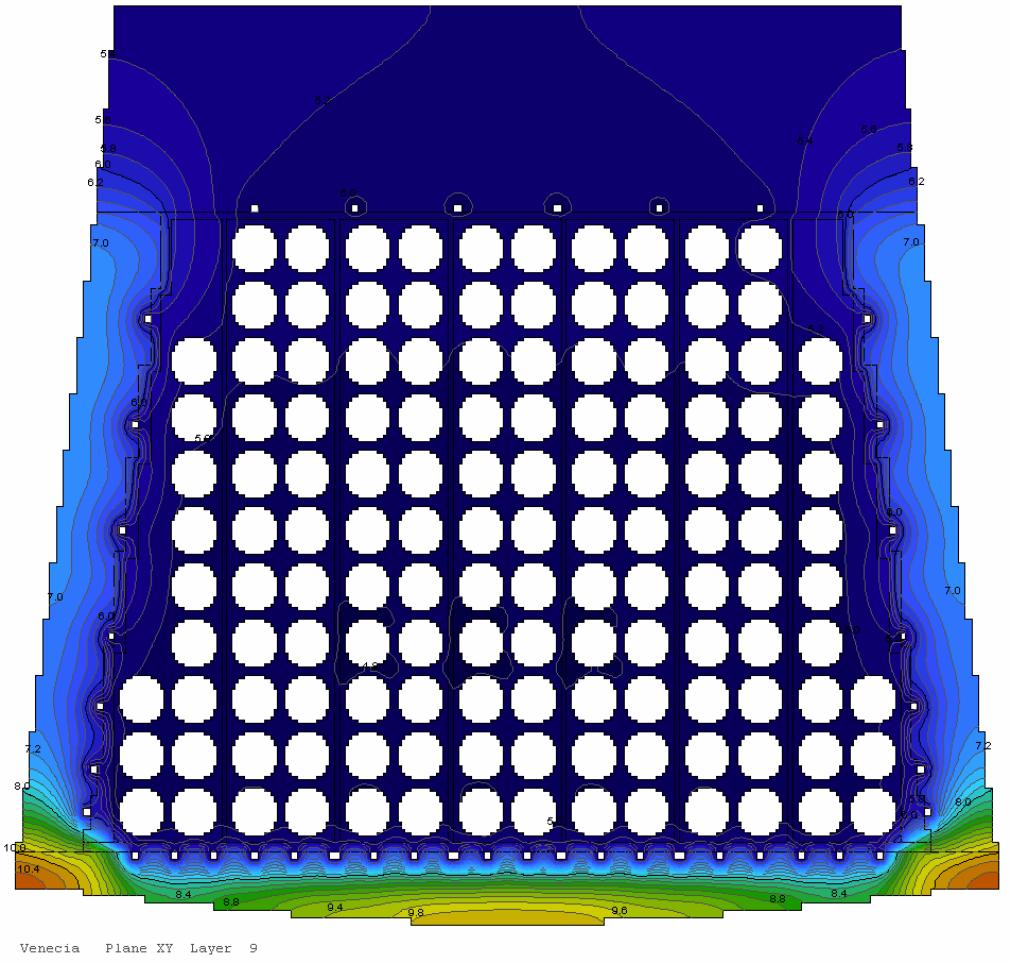
The TFC winding consists of 7 double pancakes each wound with CICC. The winding pack is placed inside a steel case. A TF coil is cooled by two helium circuits: one for the winding and another for the case and structures. Each circuit has a pump to supply a forced helium flow and a heat exchanger to expel the heat. The heat exchangers are placed in a common bath with liquid helium. The model reflects both coil geometry and hydraulic layout.

Model features:

- individual modeling of 14 cable-in-conduit-conductors (CICC) in a 1D approximation for the TF winding pack using a two-channel approach;
- individual modeling of external pipes, cryolines, manifolds and heat exchangers in a 1D approximation;
- total number of different 1D objects exceeds 300;
- detailed description of different transient heat loads over 1D objects;
- quasi-3D simulation of the TFC structures via 2D modeling of 32 cross-sections of the TF case with radial plates and insulation (the total number of mesh nodes exceeds 1,000,000);
- detailed description of different transient heat loads over 2D cross-sections;
- modeling of pressure-mass flow rate characteristics of control valves and pumps;
- modeling of a liquid coolant bath.



TFC cryogenic scheme used in VENECIA simulation of normal operation.
(split between two separate LHe bathes)



Simulated temperature map for TF cross section.

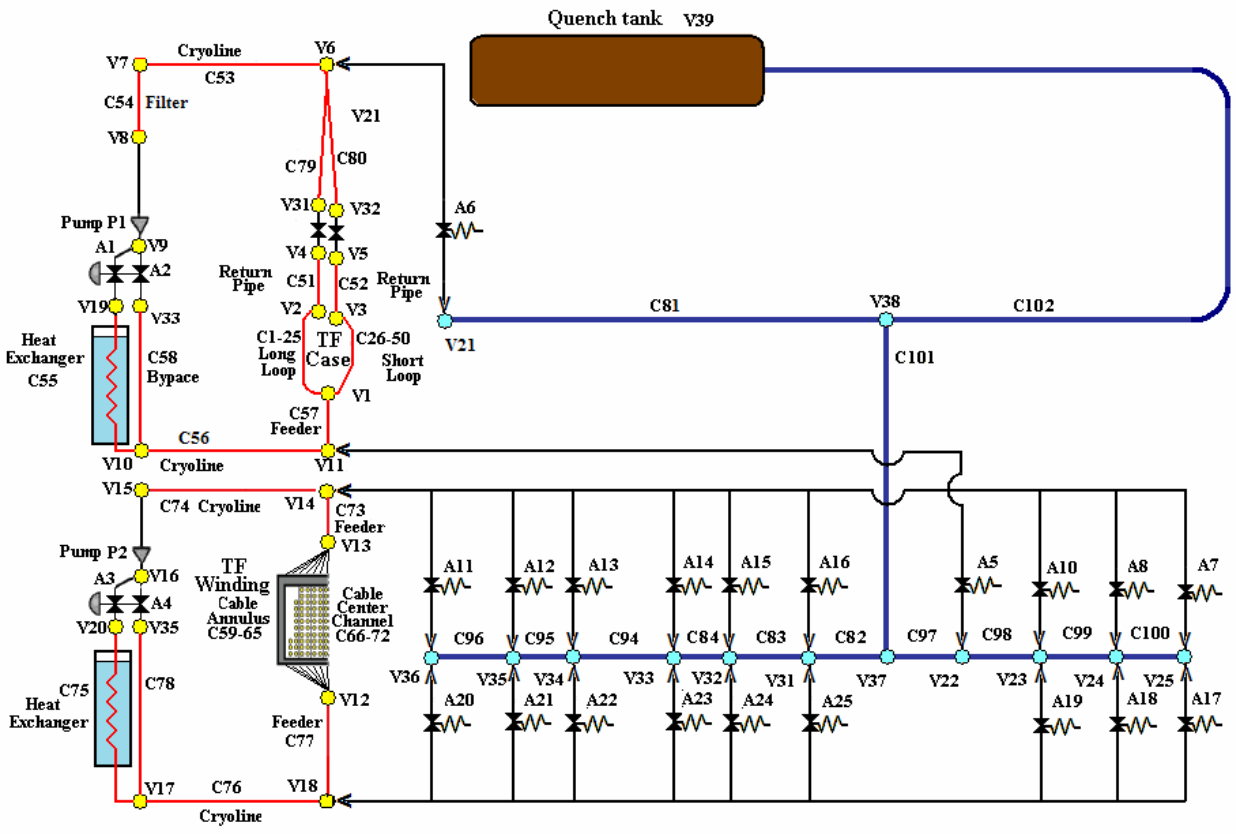
B. Fast energy discharge

The TFC model is also applicable for detailed thermohydraulic analysis of rapid helium evacuation from 18 TF coils following the fast energy discharge related to an increase of the coil structure temperature up to 50K. Huge heat deposition is resulted in release of high quantity of helium (≈ 4000 kg) that should be quickly expelled from the magnets into 80K quench tank located at a long distance from the magnets.

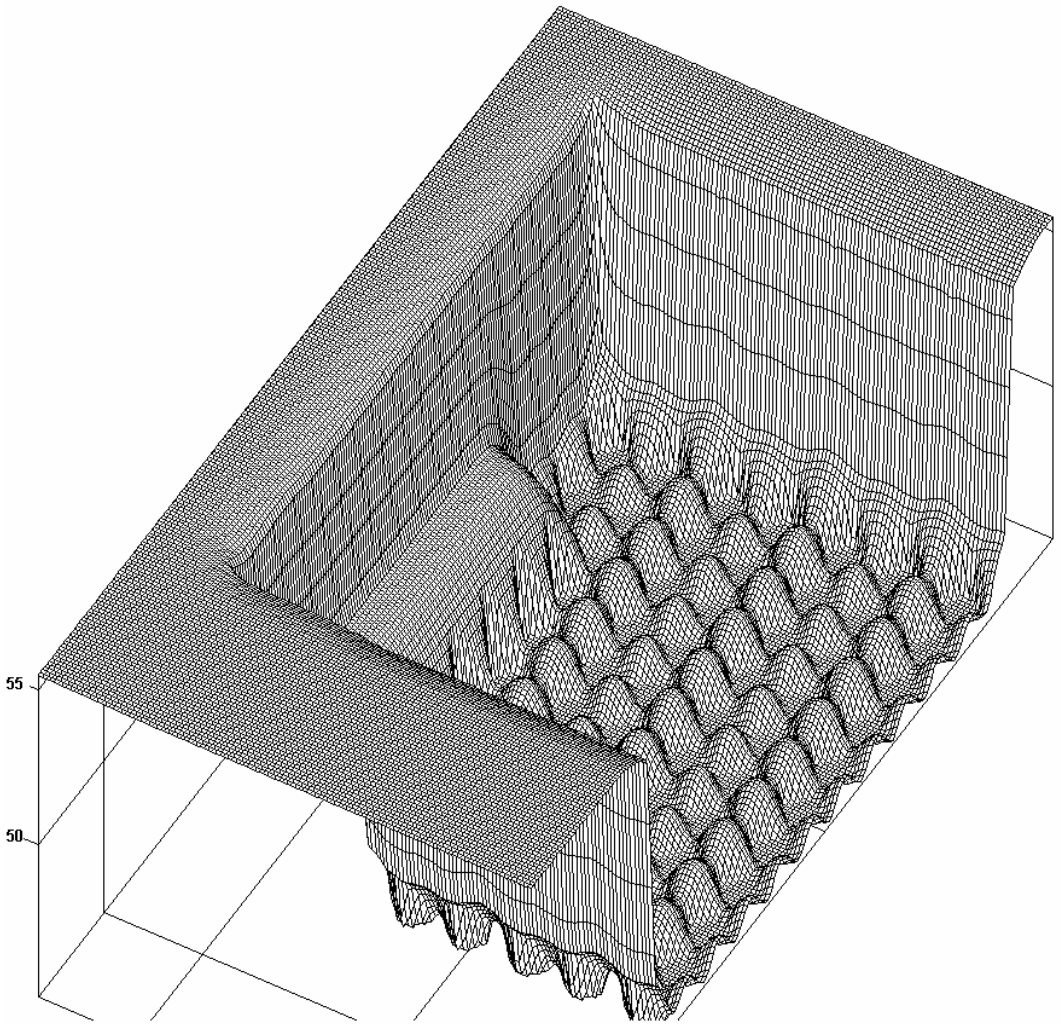
Model features:

- quasi-3D simulation of the TFC structures via 2D modeling of 32 cross-sections of the TF case with radial plates in combination with 14 CICC modelled in 1D approximation as used for nominal cooling conditions
- description of 20 relief valves for fast helium release
- description of a set of long quench transfer lines connected the 18 TF coils with 80K quench tank
- 80 K quench tank (420 m3);

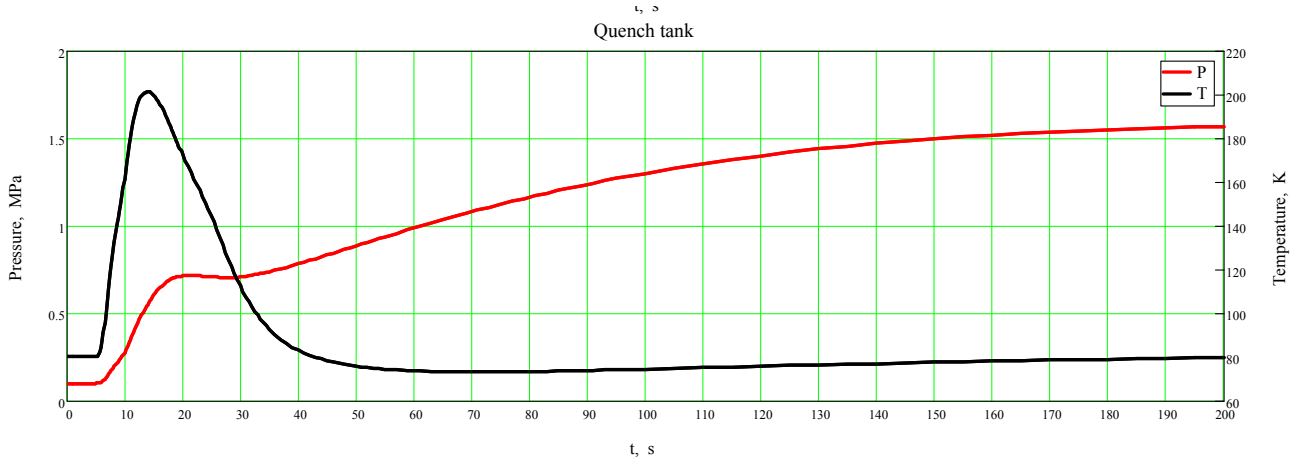
On the basis of the TF quench analysis the hydraulic diameters of quench lines are specified in a way to avoid overpressure of these lines taking into consideration their design pressure of 2 MPa.



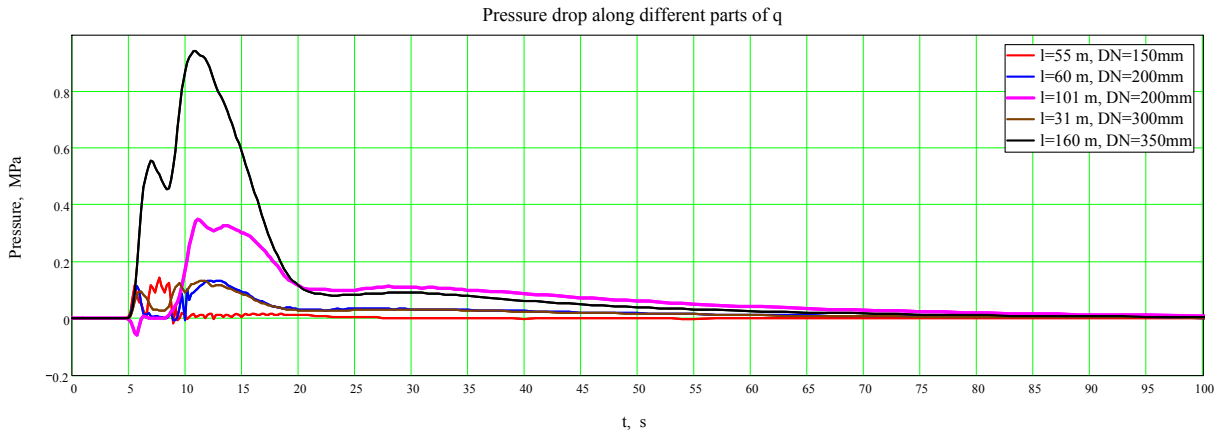
Hydraulic circuit with relief valves and long quench lines



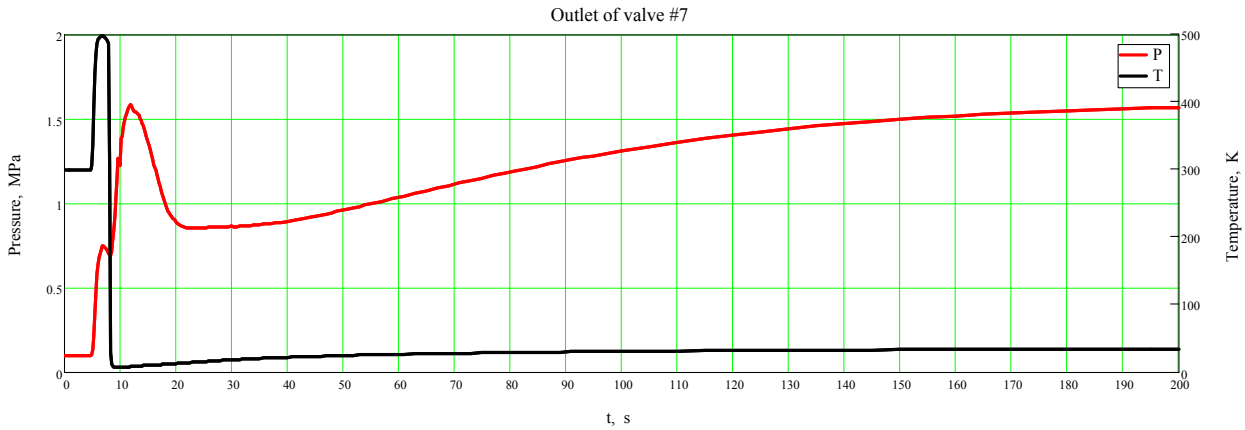
Temperature diagram for a TFC cross-section at the end rapid helium release



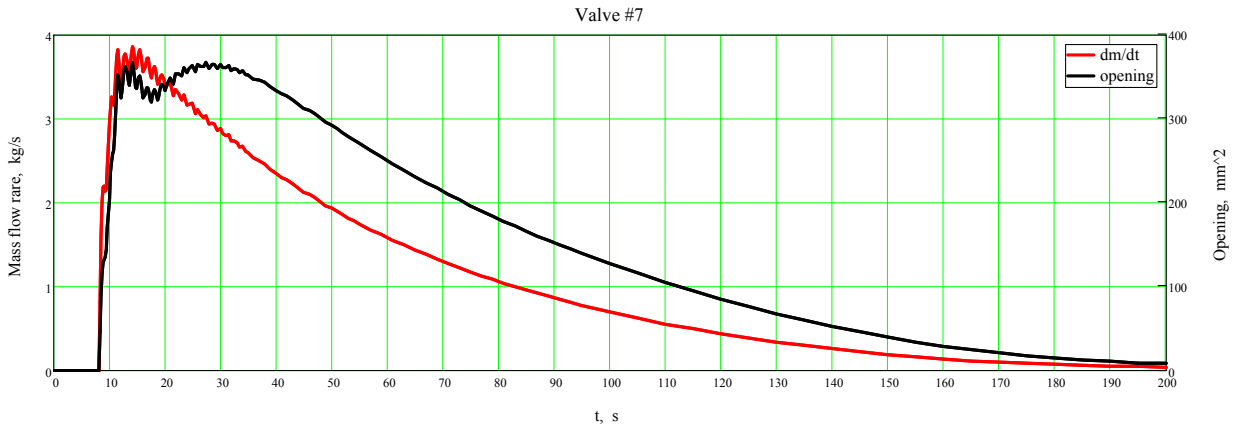
Evolution of temperature and pressure inside 80 quench tank of 400 m^3



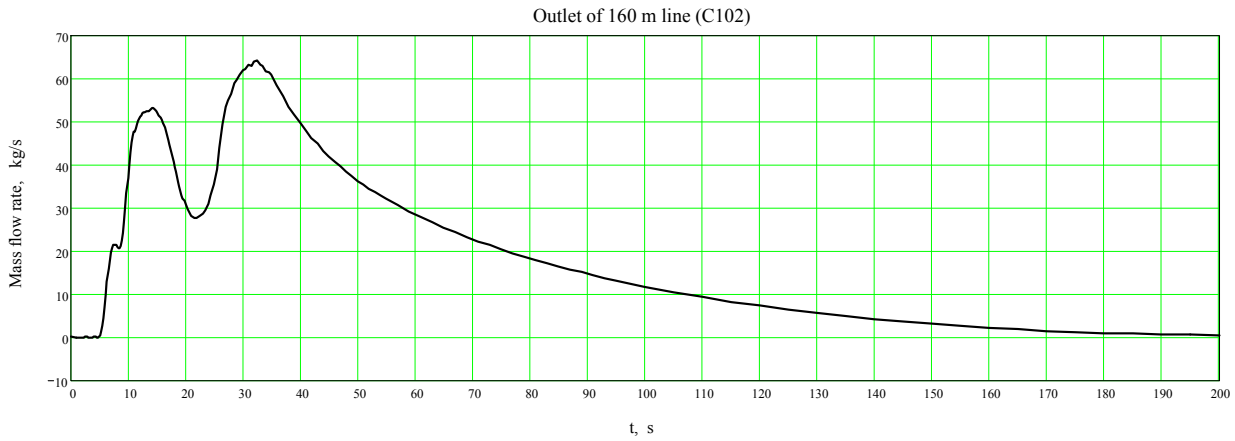
Pressure drop in a set of quench lines



Temperature and pressure evolution at the outlet of one of relief valves



Helium release via one of relief valves and its (theoretical) opening



Evolution of mass flow rate at outlet of common 160m quench line to 80K tank.

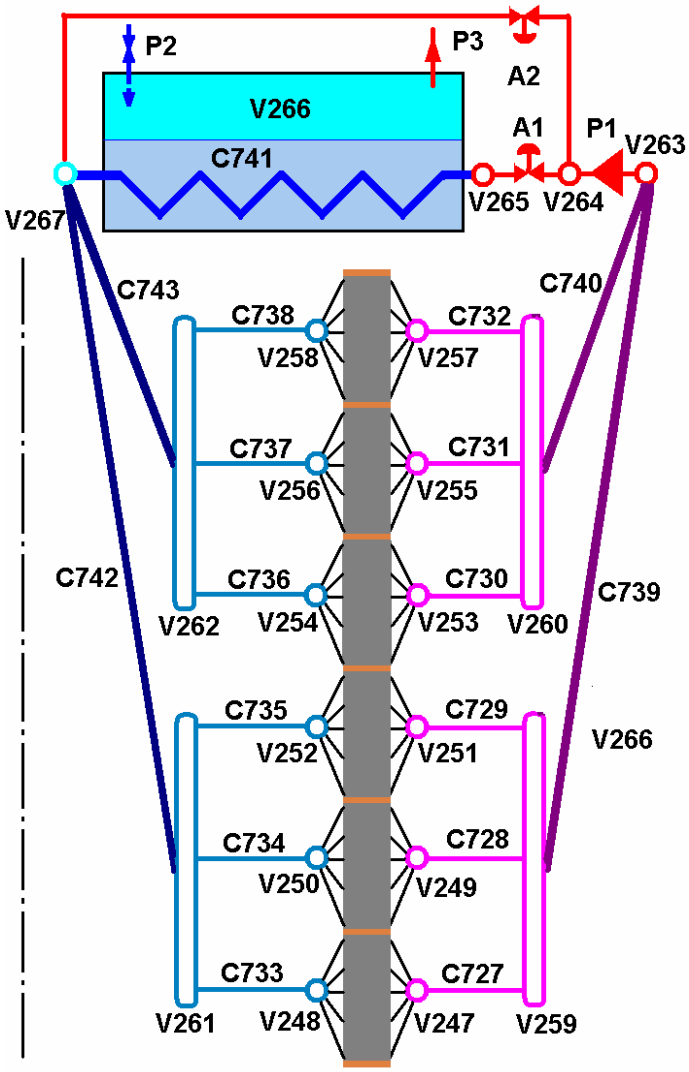
2. ITER. Central Solenoid and Poloidal Field Coils

VENECIA models for the CS and PF coils allow detailed simulations of thermohydraulic behaviour of the CS and PF winding packs together with their cooling circuits under diverse operational conditions.

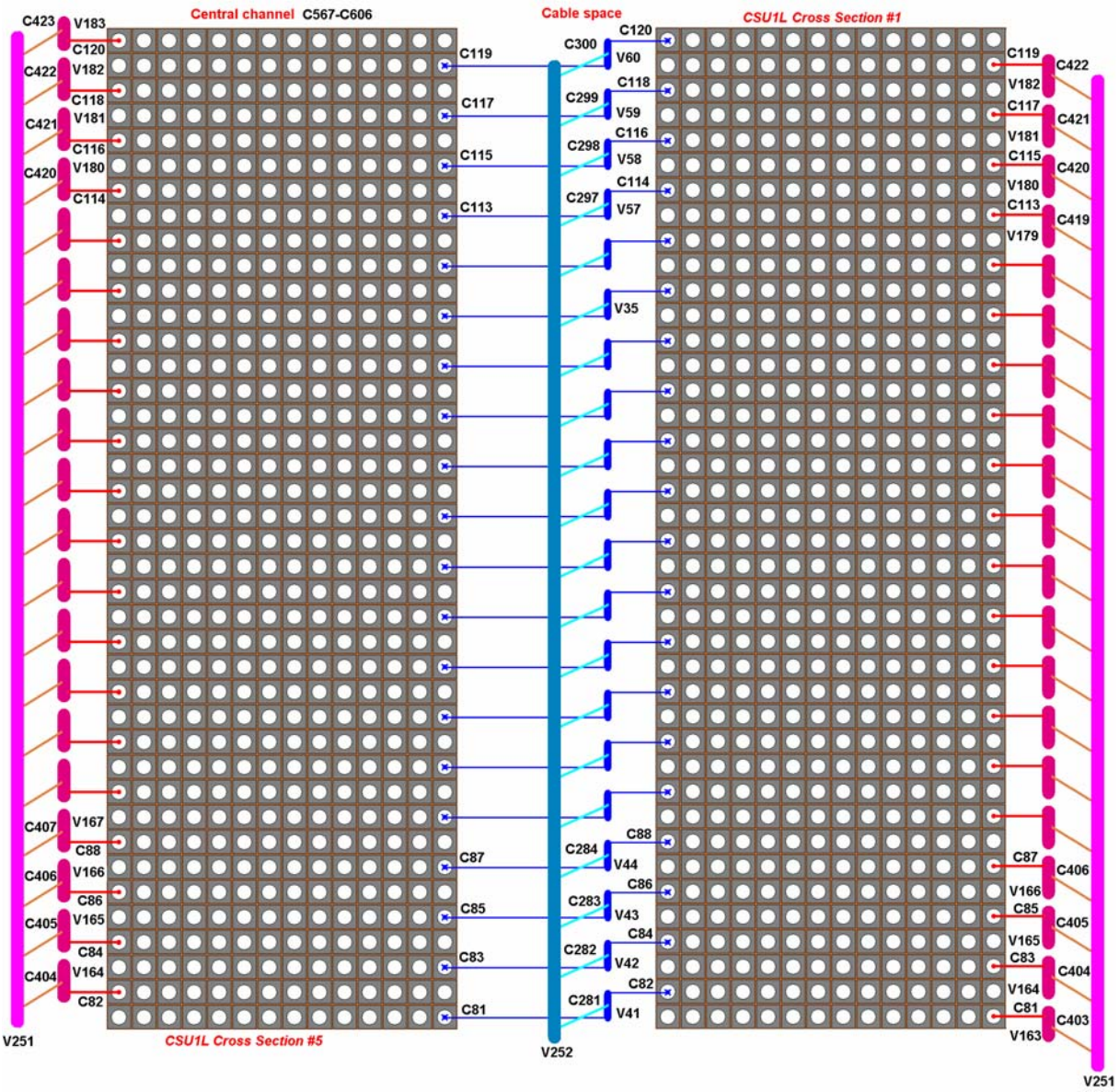
The ITER CS consists of 6 identical sections (sub-coils) stacked vertically to form a monolithic cylinder. Every section consists of 40 pancakes wound with a CICC superconductor. The pancakes inside a section are cooled in parallel. The sections are hydraulically grouped in the upper and lower sets of 3 sections, each set with a separate helium loop.

CS model provides:

- individual modeling in two-channel approach of 240 CICCs in 1D approximation for 6 CS winding sections;
- individual modeling of external pipes, cryolines, manifolds and heat exchangers in 1D approximation;
- total number of different 1D objects exceeds 1000;
- detailed individual description of different transient heat loads over 1D objects;
- quasi-3D simulation of the CS winding sections via 2D modeling of 6 cross-sections with insulation (the total number of mesh nodes exceeds 3,500,000);
- modeling of pressure-mass flow rate characteristics of pumps;
- modeling of a liquid helium bath.
- heat loads are modelled individually for each of 240 pancakes, with regard to AC losses, field gradients and mechanical strains observed in CICC

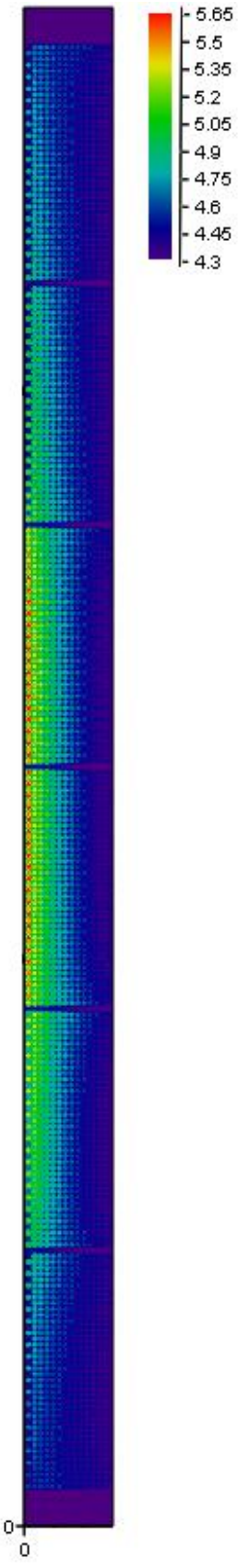


CS cryogenic circuit



Cryogenic circuit and 2D mesh over cross sections of CS lower section

2s



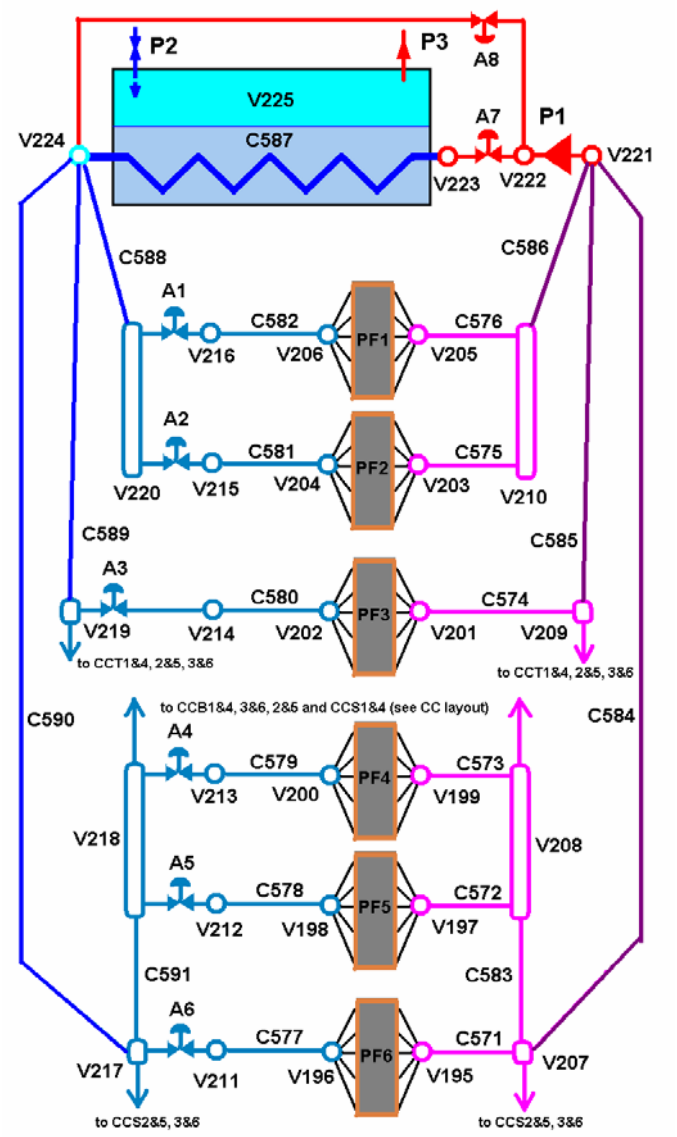
6s

Simulated temperature maps for CS coil cross section at different time points.

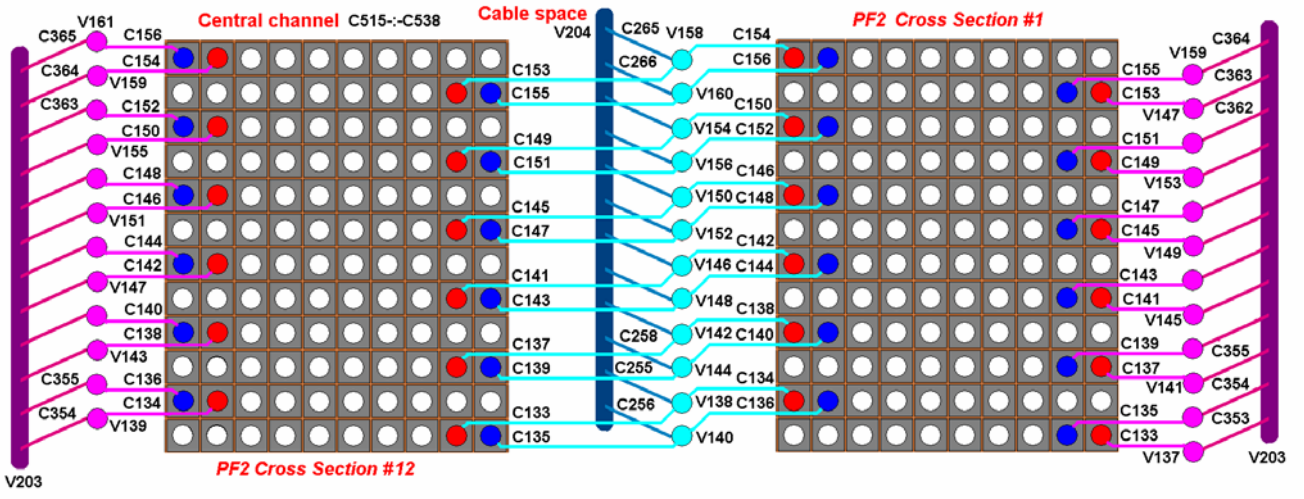
The ITER poloidal field magnet system comprises 6 PF coils and a set of correction coils. A PF winding is made of CICC wound in a “two-in-hand” manner in pancakes. The pancakes are hydraulically connected in parallel. Every PF coil is meshed individually and then are linked in a general model for the PF magnet system. In this model all PF coils and correction coils are hydraulically connected in a single cooling circuit with a circulating pump, a heat exchanger, an LHe bath, and supply / return cryolines. Additionally, a set of control/relief valves is considered to accomodate for pressure drops in different coils.

The PFC model features:

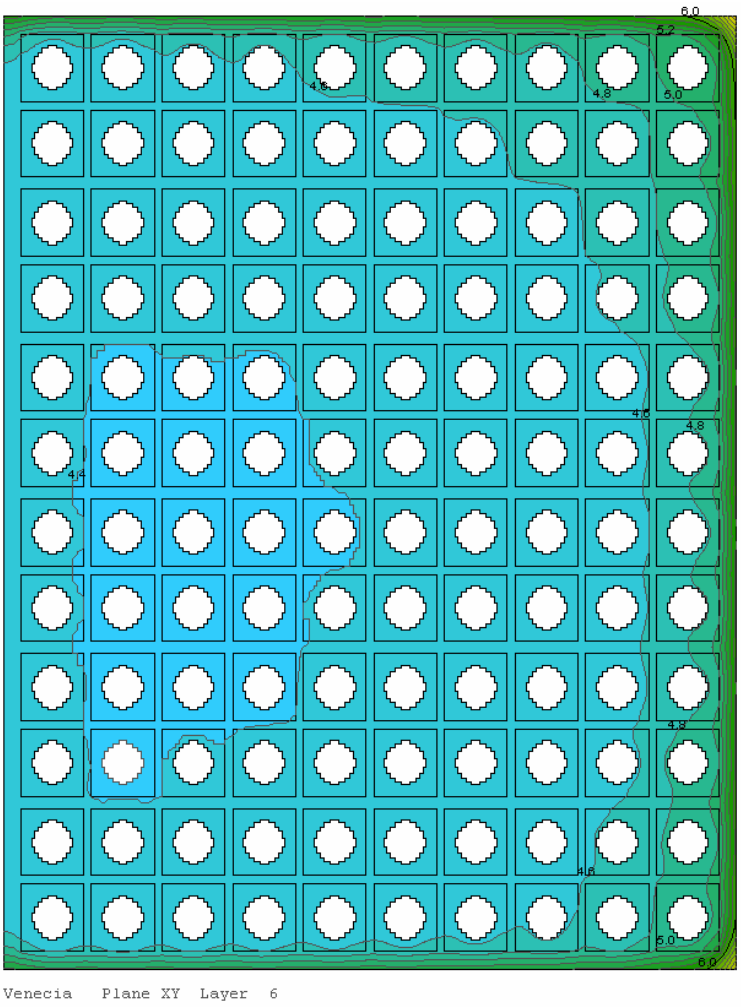
- individual modeling in two-channel approach of all CICCs in 1D approximation for 6 PF coils;
- each PFC pancake is modelled as wounded by CICC two-in-hand;
- individual modeling of external pipes, cryolines, manifolds and heat exchangers in 1D approximation;
- the total number of different 1D objects exceeds 800;
- detailed individual description of different transient heat loads over 1D objects;
- quasi-3D individual modeling each PF coil via 2D modeling of their corresponding cross-sections with insulation (the total number of mesh nodes for 72 cross-sections exceeds 3,100,000);
- modeling of pressure-mass flow rate characteristics of control valves and pumps;
- modeling of a liquid helium bath.



PFC cryogenic circuit



Cryogenic circuit and 2D mesh for PF2 cross sections. About 23,500 mesh nodes for each of 12 sections (totally 282,000 nodes)



Simulated temperature map for PF2 coil

Solvable problems related to helium flow control

VENECIA offers a unique capability to model adjustable helium flow that enables controllability studies for:

- Cryogenic schemes with one and two SHe heat exchanges;
- Mitigation of heat load on the LHe bath with one heat exchanger by adjustment of helium flow by gradual opening of forward stream bypass valve;
- Mitigation of heat load on the LHe bath by adjustment of helium flow in the SHe heat exchangers by speed of SHe circulator (applied for analysis of the ITER TF coils)
- Operation with and without of the cold compressor;
- Mitigation of heat load on the LHe bath by adjustment of helium flow in the SHe heat exchangers via back stream bypass valves from outlets to inlets of these heat exchangers (applied for analysis of the ITER CS);

Specially developed control algorithms provides simulations for

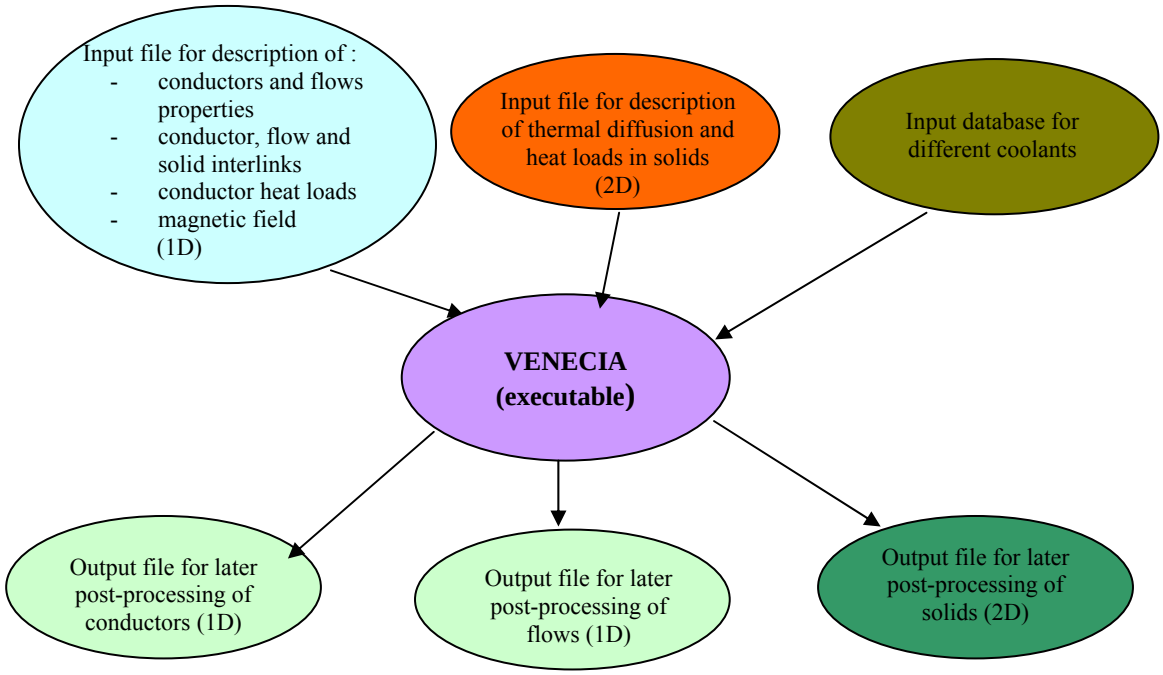
1. adjustment of helium flow in an SHe heat exchanger by means of control/bypass valves; backstream valves; or with circulator pump speed.
2. adjustment of helium flow of a cold vapor compressor and J-T valve of a LHe bath.

STRUCTURE AND MODELING STRATEGY

VENECIA is aimed to complex thermalhydraulic simulation of superconducting and cryogenic systems, including cryogenic plant elements and accessories. A set of basic mathematical models is used to simulate typical components of magnet and cryogenic systems: *coolant flow, conductor, collector, valve, pump, heat exchanher, joint, solid, electric circuit,* etc. Every component is described by an individual set of algebraic, differential equations and equations in partial derivatives. The equations are solved using an intrinsic numerical technique.

Each basic model has a fixed argument list that provides an adequate description of the component to cover typical operation conditions. To simulate the loading conditions each model is linked with a standardized data set defining the loading modes and magnitudes. These data are implemented in VENECIA via the NAMELIST module, which is a unificated core of the code. NAMELIST is constantly modified as new basic models are added. However, the VENECIA philosophy is to keep the possibility for the user to run models developed with earlier versions.

The number of basic models of each type used in simulations depends only on task specifics and adopted assumptions and approach. A complex calculation model of a real magnet with its cooling system is constructed from these basic models by coupling them to each other via hydraulic and thermal links in a way of "lego"-type assembly.



VENECIA structure

Cryogenic circuit is modeled using 0D (lumped objects) and 1D approximations. A set of 1D equations describe heat conduction between coolant flows and solids taking into account hydraulic and thermal coupling.

Heat diffusion in solids is modeled in 2D approximation via a set of cross-sectional cuts associated with properties of different materials and proper boundary conditions.

A special strategy is applied to integrate the 2D model for thermal diffusion in solid structures with 1D model for cooling channels through diverse links and boundary conditions in order to provide comprehensive quasi-3D modeling of a coil. A single algorithm based on a semi-implicit method is implemented to integrate and solve the equations.

MESHING

The 2D model used to simulate thermal diffusion takes into account the interturn and interpancake heat transfer (in a number of coil cross-sectional cuts) and heat load distributions
to adequately simulate the coils response to a range of conditions. This permits a qualitative analysis of the cryogenic system necessary to operate different coils in normal operating conditions (plasma burn, plasma disruption) as well as during fast transients (safety discharge).

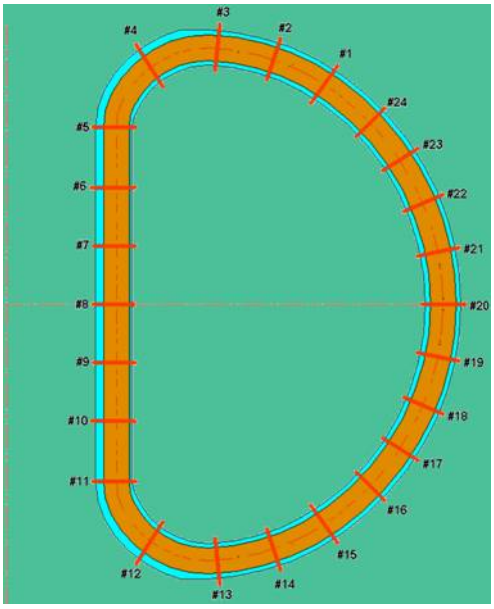
The use of a square nonuniform meshing ensures highly efficient and flexible computations. Model discretization can be varied to a high degree, from a coarse to extremely refined mesh, so as to adapt for the desired approximation accuracy.

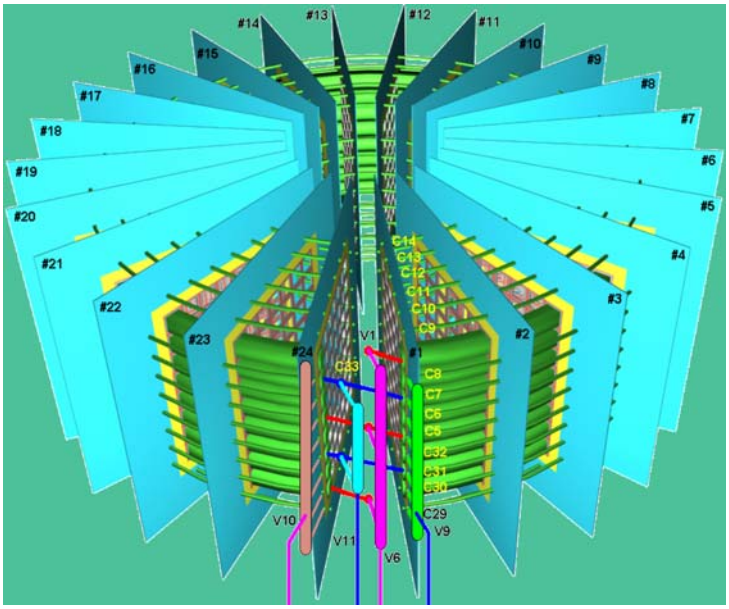
The user can choose between two 2D meshing options:

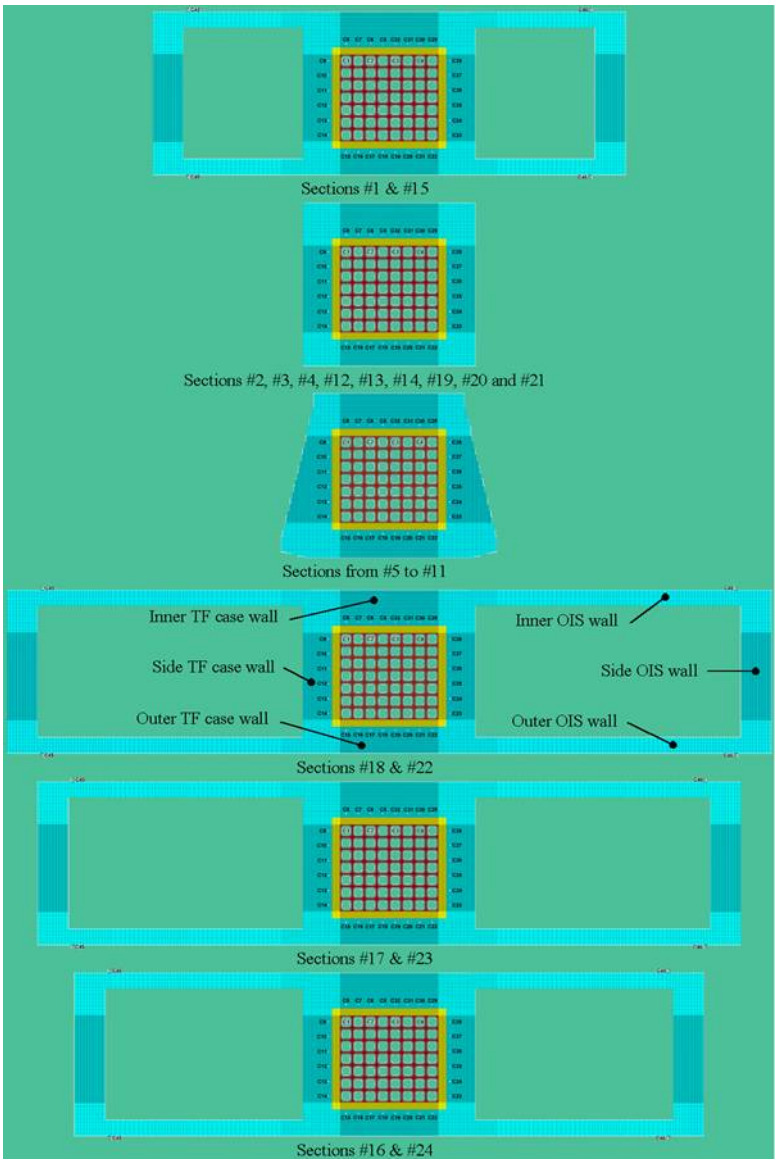
1. Meshing with **VENECIA** pre/post processing tools. The meshes are then treated by the **VENECIA solver.**
2. Meshing with a special **VENECIA** tool operable in AutoCad environment:
   When AutoCAD is run, a dedicated package is started enabling additional commands to build and modify a 2D mesh. The resultant 2D mesh can be saved in a file in an AutoCad-supported or Venecia-supported format*.

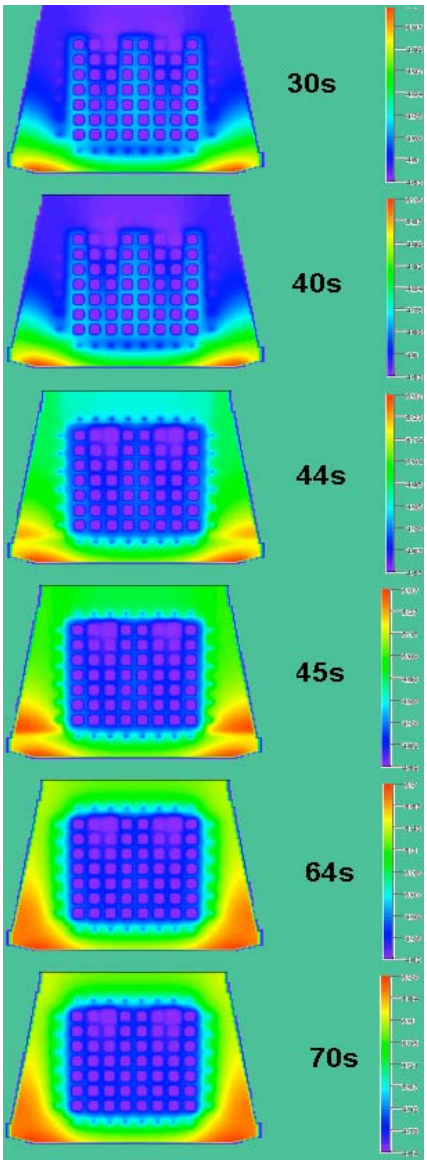
The meshing quality can be inspected with the use of **VENECIA Viewer**.

* Support for additional formats can be implemented on customer's demand.


#1
#2
#3
#4
#5
#6
#7
#8
#9
#10
#11
#12
#13
#14
#15
#16
#17
#18
#19
#20
#21
#22
#23
#24


#1
#2
#3
#4
#5
#6
#7
#8
#9
#10
#11
#12
#13
#14
#15
#16
#17
#18
#19
#20
#21
#22
#23
#24
V1
V6
V9
V10
V11
C29
C43


Sections #1 & #15
Sections #2, #3, #4, #12, #13, #14, #19, #20 and #21
Sections from #5 to #11
Inner TF case wall
Side TF case wall
Outer TF case wall
Inner OIS wall
Side OIS wall
Outer OIS wall
Sections #18 & #22
Sections #17 & #23
Sections #16 & #24

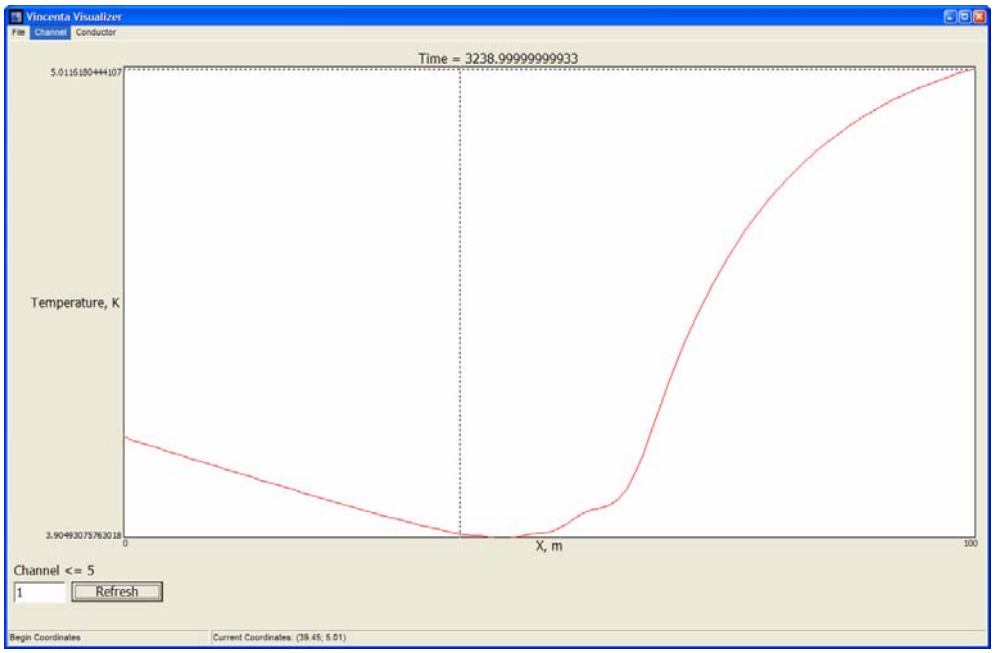

30s
40s
44s
45s
64s
70s

INTERFACE

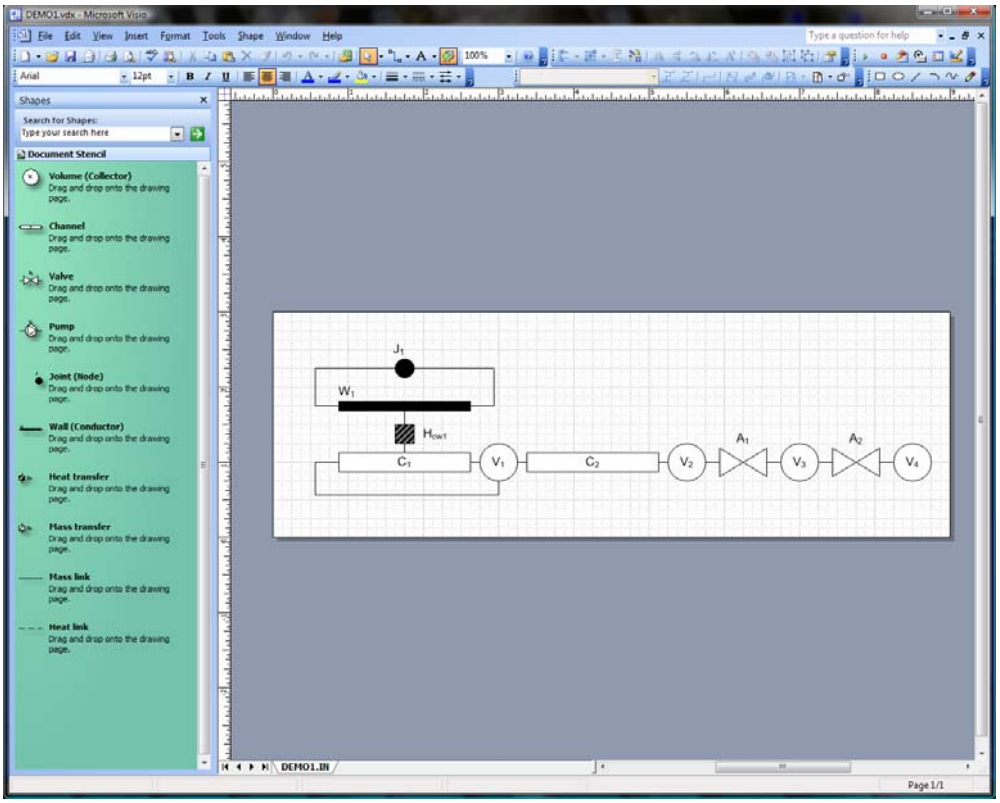
A complex calculation model of superconducting systems involves a bundle of physical phenomena in a variety of objects. To facilitate control and analysis of the bulk of data, VENECIA offers a set of pre- and postprocessing tools.

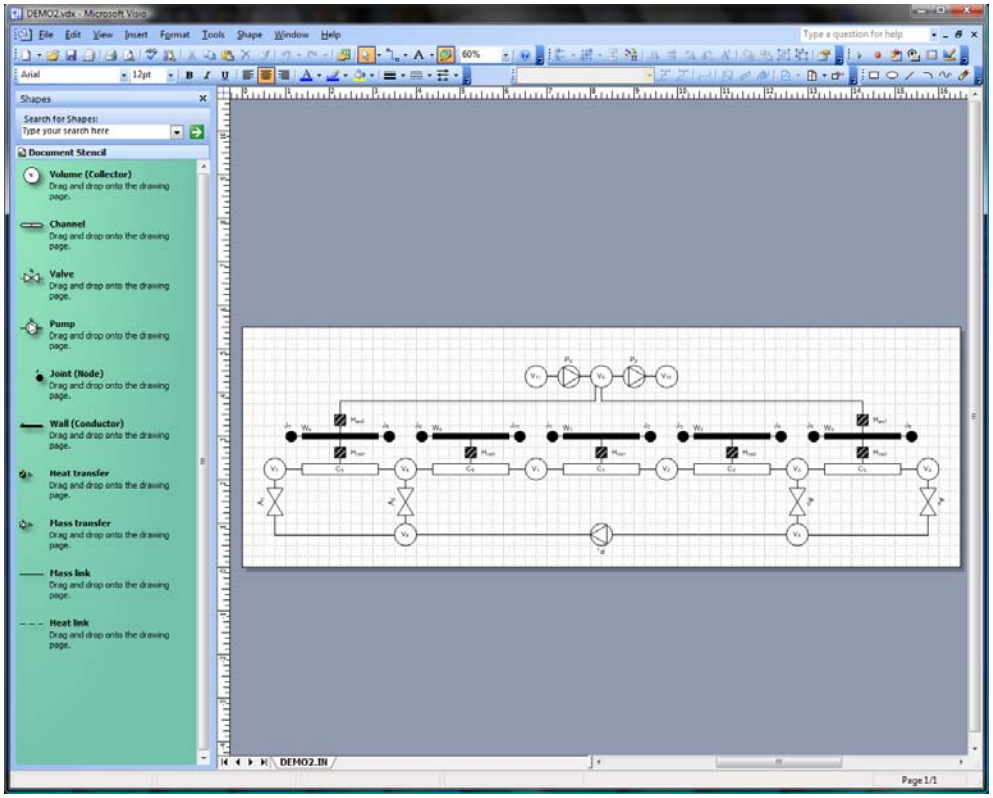
Program shell enables selection of the task directory, checkout of input file and model linkage, visualization of 2D mesh, launch of simulation, real-time monitoring, temperature maps viewing, selection and plotting of results.

> ***Real-time monitor*** shows runtime results and provides graphic representation of real-time parametric evolutions in a form of user-defined plots to give the user a possibility for quick analysis of the simulation.



Schematization engine provides visualization of the linkage between 1D and 0D (lumped) objects forming a modeled cryogenic circuit. Schematics is generated from the Venecia input file and can be viewed and easily re-arranged in Microsoft Visio using drug-and-drop capability. For convenience, an indication is provided for adjacent objects in compliance with the links specified in the input file.



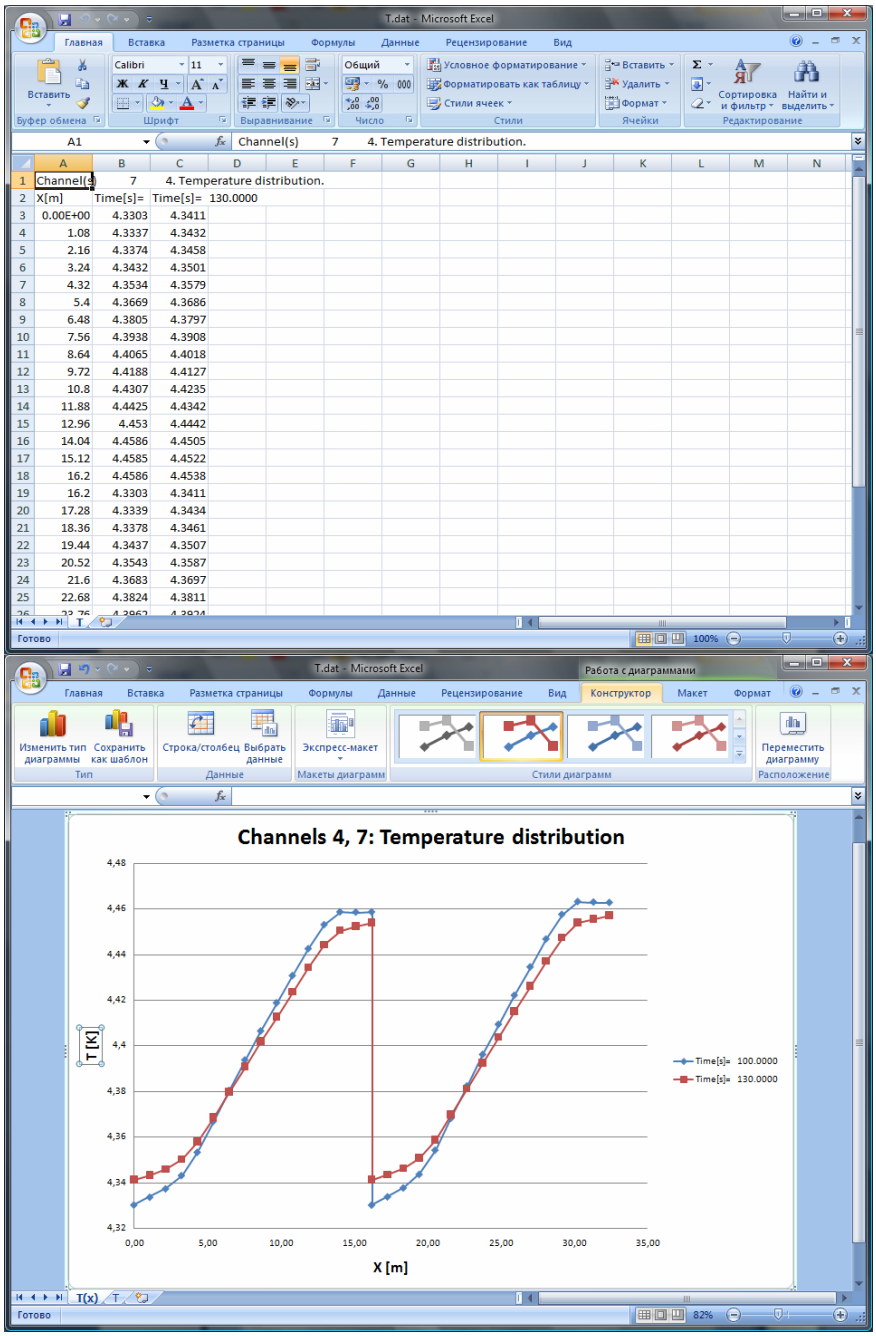


Plotter gives the user a possibility to select outputs, tabulate and plot the curves of interest. The results for plotting are specified in a formatted output file. Selection of the dependences is possible via a prompted menu or common text editor. After selection, a file is generated in the text format suitable for Microsoft Excel.

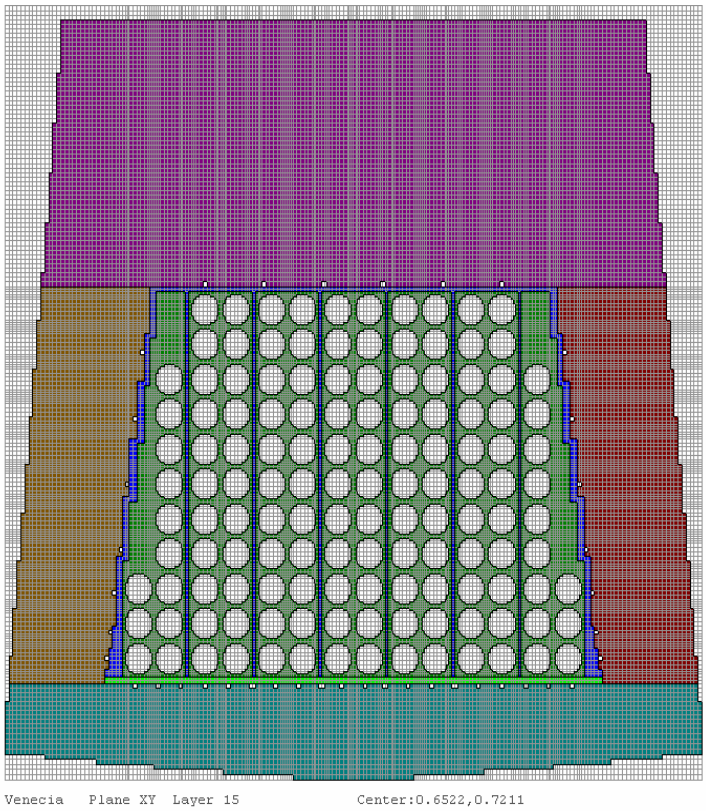
For 0D objects (collectors, joints, pumps, valves) time variations can be plotted as single or multicurve diagrams.

Parameters of 1D objects (channels and conductors) can be displayed as longitudional variations f(x) in the following forms:

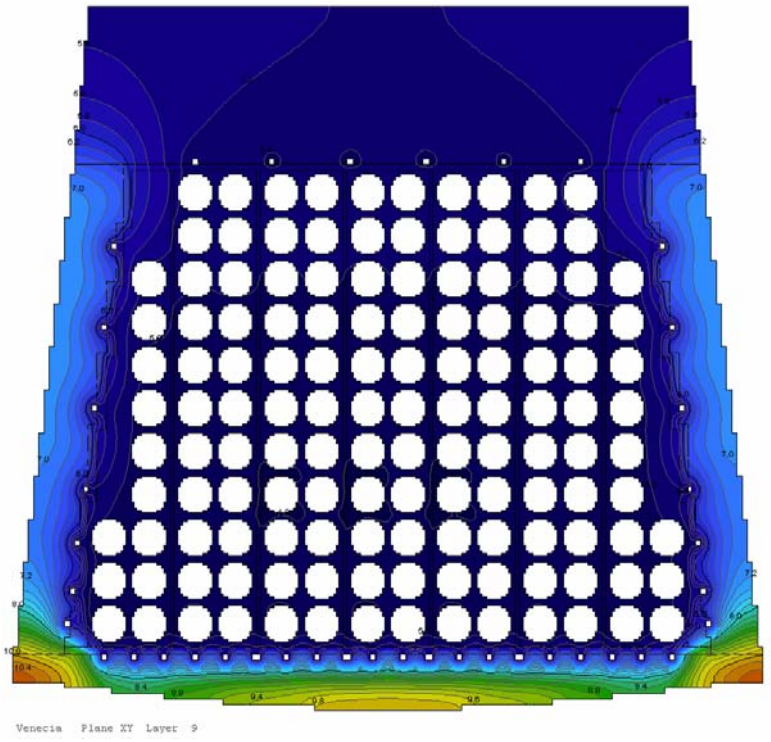
- in one selected channel/conductor for a single time point;
- in several selected channels/conductors for a single time point;
- in one channel/conductor for a set of time points;
- in several channels/conductors for a set of time point. -



2D viewer is used to visualize 2D mesh over selected cross-sections and display calculated temperature maps for any time point. Integration of the cross-sections into a 3D image is also enabled.



Modelized cross-section with 2D mesh and materials allocation



Output 2D temperature map

MATHEMATICAL MODELS

Coolant flow

The core of the program consists in the description of the coolant flow in channels, possibly coupled (exchange of mass and energy), also possibly coupled with walls (or conductors) which can also be coupled to each other. The coolant flow model *channel* simulates transient parameters of a compressible coolant flow inside a channel. *Channel* is described by a set of 1D equations of continuity, momentum and energy conservation laws completed with the transverse mass, momentum and energy transfer terms to take into account the thermal-hydraulic coupling with different flows and solid materials. Generally, the coolant flow in a channel can simultaneously have a thermal contact with some various solid materials and coolant in other channels. Besides, different coolant flows (only identical coolants) inside channels can be hydraulically coupled between themselves in the longitudinal direction (due to the transverse holes) as, for example, in the Cable-In-Conduit Conductor (CICC) with a central channel. The final set of equations describing a transient process for coolant flow in the *i* channel in terms of interaction with flows in *k* channels and *m* conductors has a form:

$$\frac{\partial \rho_i}{\partial t} + \frac{\partial \rho_i V_i}{\partial x} = \frac{\sum_k \Gamma_{ki}^{\rho}}{A_i} \qquad (1)$$

$$\frac{\partial \rho_i V_i}{\partial t} + \frac{\partial}{\partial x}\left(P_i + \rho_i V_i^2\right) = \frac{-2 f_i \rho_i V_i |V_i|}{D_{h_i}} + \rho_i F_i(x) + \frac{\sum_k \Gamma_{ki}^{\rho V}}{A_i} \qquad (2)$$

$$\frac{\partial}{\partial t}\rho_i\left(H_i + \frac{V_i^2}{2} - \frac{P_i}{\rho_i}\right) + \frac{\partial}{\partial x}\rho_i V_i\left(H_i + \frac{V_i^2}{2}\right) + \frac{\partial}{\partial x} q_{HeII} = \frac{\sum_m Q_{mi}^{conv} + \sum_k \Gamma_{ki}^{\rho H}}{A_i} \qquad (3)$$

where ρ, P, H, V – coolant density, pressure, enthalpy and velocity accordingly; f – friction factor; A – coolant cross-sectional area; D_h – hydraulic diameter; Q_{mi}^{conv} – convective heat transfer from conductor m to coolant i per unit of length; $\Gamma_{ki}^{\rho} = -\Gamma_{ik}^{\rho}$, $\Gamma_{ki}^{\rho V} = -\Gamma_{ik}^{\rho V}$, $\Gamma_{ki}^{\rho H} = -\Gamma_{ik}^{\rho H}$ – mass, momentum and enthalpy flux from k to i channel (and vice versa), $\rho_i F_i(x)$ – volumetric force applied to coolant in i channel, $q_{HeII_i} = -\left(\frac{1}{f(T_i)}\frac{\partial T_i}{\partial x}\right)^{\frac{1}{3}}$ - heat flux in turbulent He II flow, $f(T)^{-1}$ - He II thermal conductivity function for turbulent flow.

To determine a transverse mass flux it is assumed that the pressure difference between coupled flows is small enough. In such approach the local transverse mass and enthalpy flux from k to i flow can be obtained in the following simple form:

$$\Gamma_{ki}^{\rho} = S_{ki} \cdot \begin{cases} \sqrt{2(P_k - P_i)\rho_k}, & P_k > P_i \\ -\sqrt{2(P_i - P_k)\rho_i}, & P_k < P_i \end{cases},$$

$$\Gamma_{ki}^{\rho H} = \Gamma_{ki}^{\rho} \cdot \begin{cases} H_k + V_k^2/2, & \Gamma_{ki}^{\rho} > 0 \\ H_i + V_i^2/2, & \Gamma_{ki}^{\rho} < 0 \end{cases}.$$

where S_{ki} is a coefficient with the dimensionality of cross-section area per unit of length between *k* and *i* flows. As the transverse mass flux from *k* flow is suggested to be normal to *i* flow, the momentum transport term is negligible (i.e. $\Gamma_{ki}^{\rho V} = -\Gamma_{ik}^{\rho V} = 0$).

Such description of the transfer processes between flows is applicable to modeling mass and energy exchange if an assumption of a small pressure drop between flows is valid (transverse cross area is large enough). The advantage of this model is a possibility to analyze the influence of transverse coupling on thermal and hydraulic parameters of coupled flows and to estimate the transverse mass and energy transport term.

The following boundary conditions are used to close the system (1)-(3). The coolant pressures at the ends of the *channel* are supposed to be identical to pressures in joined *collectors*. When coolant enters the *channel*, the coolant enthalpy at an appropriate end of the *channel* is assumed to be equal to the enthalpy in the joined collector. At the closed end of the tube the coolant velocity assumed to be zero.

Conductor

A transient temperature distribution in conductor components is described by a 1D equation of heat balance with the transverse conductive and convective heat exchange and Joule heating terms. A temperature distribution across the conductor section is assumed to be uniform. Uniform temperature distribution is a "natural" assumption for a 1D approach and treated as an average temperature for the given cross-section. In the general case, *conductor* could have simultaneously a contact with different coolant flows as well as with other *conductors*. So the equation for a binary conductor *m* including the heat exchange with conductors *n* and coolant flows *i* has a form:

$$\left(A_m^1 C_m^1 + A_m^2 C_m^2\right)\frac{\partial T_m}{\partial t} = -\cos^2\theta \frac{\partial}{\partial x}\left(\left(A_m^1 k_m^1 + A_m^2 k_m^2\right)\frac{\partial T_m}{\partial x} \right) + Q_m^{Joule} + \sum_i Q_{im}^{conv} + \sum_n Q_{nm}^{cond} + \sum_k Q_{km}^{wall} \quad (4)$$

$$Q_m^{Joule} = \frac{I_{op}^2}{\left(\sigma_m^1 A_m^1 + \sigma_m^2 A_m^2\right)\cos^2\theta} g(T_m),$$

$$g(T_m) = \begin{cases} 0, T_m < T_{cs} \\ 1 - \dfrac{I_c(T_m)}{I_{op}}, T_{cs} < T_m < T_c \\ 1, T_m > T_c \end{cases} \quad (5)$$

$$Q_{im}^{conv} = h_{im} \cdot \gamma_{im} \cdot (T_i - T_m) \quad (6)$$

$$Q_{nm}^{cond} = h_{nm} \cdot \gamma_{nm} \cdot (T_n - T_m) \quad (7a)$$

$$Q_{km}^{wall} = \gamma_{km} \cdot k_k^{wall} \frac{\partial}{\partial r}\left(T^{wall}(x,r)\right) \quad (7b)$$

where *T*, C, *k*, A^1+A^2 – conductor temperature, heat capacity, thermal conductivity and cross-section area of components accordingly; *h*, γ –coefficient and perimeter of heat exchange; Q_{nm}^{cond} –conductive heat transfer from the conductors *n* and *m* per unit of length; Q_{im}^{conv} -convective heat transfer from coolant in the channel *i* to the conductor *m* per unit of length; Q_m^{Joule} - Joule heating of the conductor

m per unit of length; Q_{km}^{wall}, k^{wall}, T^{wall} - heat flux to the wall k, thermal conductivity and temperature of the wall k, correspondingly; I_{op} - conductor current; σ - conductor electrical conductance; I_c, T_c, T_{cs} - critical current, critical temperature and current sharing temperature, accordingly.

In these equations the material cross-section A_m is treated as the cross-section in a plane normal to the conductor axis ("twisted" cross-section). The same assumption is applied to the material heat exchange perimeter γ_m. It is taken that the "twisted" material cross-section and the perimeter (for twisted superconducting strands) used in the above equations are defined as $A_{tw} = A_{non\ tw}/cos\theta$ and $\gamma_{tw} = \gamma_{non\ tw}/cos\theta$, where $\theta>0$ is an average twist angle. This generalized angle takes into account the average twist of cabling stages. For non-twisted materials $cos\theta=1$.

The following boundary conditions are used to close equation (4). The temperatures at the ends of the conductor are defined through temperature of connected joints by boundary condition of third kind

$$\kappa_m \frac{\partial T_m}{\partial x} + h_i \cdot \left(T_m - T_j^{joint}\right) = 0 .$$

Collector

The mathematical model *collector* is intended for 0-D simulation of coolant behaviour in a collector and used as a node element for connecting different *channels* and *valves* in series-parallel to one another. The laws of mass and energy conservation define the coolant behaviour inside the collector. For the collector i the following equations are used:

$$\Omega_i \frac{d\rho_i}{dt} = \sum_k G_{ki}^{\rho} + \sum_n G_{ni}^{\rho} \qquad (8)$$

$$\Omega_i \frac{d}{dt}\left(\rho_i H_i - P_i\right) = \sum_k G_{ki}^{\rho H} + \sum_n G_{ni}^{\rho H} + \sum_m Q_{mi}^{conv} \qquad (9)$$

where Ω – collector volume; ρ, P, H – coolant density, pressure and enthalpy correspondingly; $G_{ki}^{\rho}, G_{ni}^{\rho}$ - mass flow from the channel k and the valve n to the collector i; $G_{ki}^{\rho H}, G_{ni}^{\rho H}$ - enthalpy flow from the channel k and the valve n to the collector i; Q_{im}^{conv} - convective heat exchange between coolant in the collector i and the joint m.
The mass flow terms in (8) are assumed to be positive if the flow is coming into the collector considered and vice versa. The energy term in (9) is defined as

$$G_{ki}^{\rho H} = G_{ki}^{\rho} \times \begin{cases} H_k + V_k^2/2, & G_{ki}^{\rho} > 0 \\ H_i & G_{ki}^{\rho} < 0 \end{cases},$$

where H_k, V_k are enthalpy and velocity of coolant outbound at the end of the channel k connected to the collector i. The same equation takes place for an energy flow from a valve.

Valve

The model *valve* is intended for calculation of a He mass flow through cryogenic elements, such as valves, holes, gaps, etc. It is assumed that mass flow through the *valve* is forced by pressure difference in pare collectors to which it is connected.

To associate the mass flow through the valve with the thermodynamic properties of coolant in both collectors we use a simplest classical conception. The flow through the valve is modelled as isentropic expansion ($\delta H - \delta P/\rho = 0$) of the compressible fluid from the inlet pressure (upstream the valve) to the outlet pressure. Depending on a pressure drop between the inlet/outlet the outlet valve pressure is equal to the outside pressure (sub-critical flow out) or critical pressure (critical flow out, the outlet velocity is equal to the local sonic speed). In both cases, the full enthalpy ($h+u^2/2$) of the flow under the isentropic expansion is conservative, that allows calculation of all thermodynamic characteristics of the outlet flow. Two basic parameters are assigned for the valve: the minimum cross-section area A of the valve (which depends on the valve lift) and correction factor μ to take into account a non-isentropic expansion.

For a sub-critical flow the pressure at the valve outlet is equal to the pressure in the outlet collector. In the case of the critical flow the outlet pressure is equal to the critical one. So, for calculation of coolant properties in the valve outlet k the following system of equations is used:

$$H_k + \tfrac{1}{2}U_k^2 = H_\Omega^{in}, S_k = S_\Omega^{in} \tag{10}$$

$$P_k = \begin{cases} P_\Omega^{out}, U_k < C_k \\ P_k^{crit}, U_k = C_k \end{cases} \tag{11}$$

$$G_k = \mu_k A_k \rho_k U_k \tag{12}$$

where H_k, U_k, P_k, ρ_k is the enthalpy, velocity, pressure and density of coolant at the valve outlet; H_Ω^{in}, S_Ω^{in} is the enthalpy and entropy of coolant in the inlet collector; P_Ω^{out} is the pressure in the outlet collector and P_k^{crit} is the critical pressure in the valve outlet. For a real non-isentropic flow the correction factor $\mu_k < 1$ is used.

The set of additional parameters allows control of the valve lift depending on the pressure variations in adjoining collectors applicable for modeling the multi-purpose valves.

Solids

To simulate transient heat diffusion in the winding composite a 2D model is used in the Cartesian or axial-symmetrical approach. A differential equation for temperature over the given cross-section S of the winding k is:

$$C_k(T)\frac{\partial T_k}{\partial t} = q_{V\,k}(x,r,t) + \\ + \frac{\partial}{\partial x}\left(\kappa_k(T)\frac{\partial T_k}{\partial x}\right) + \frac{1}{r^n}\frac{\partial}{\partial r}\left(\kappa_k(T)\cdot r^n \frac{\partial T_k}{\partial r}\right) \quad , for S \times S_t \tag{13}$$

$$T_k(x,r,0) = \psi(x,r), \quad t = 0\,,$$

n=0 – Cartesian coordinates; n=1 – cylindrical coordinates.

The boundary condition of the third kind is formulated for the *cross-section k* having heat exchange with coolant flow *i*

$$\kappa_k \frac{\partial T_k}{\partial n} + h_i \cdot \left(T_k - T_i^{He}\right) = 0$$

where h_i is the heat transfer coefficient and T_i^{He} is the corresponding temperature of coolant inside the *channel i*. At the outer surface of the winding the appropriate boundary condition is formulated.

Pump

The pumping process is modelled as adiabatic compression of fluid from inlet pressure P_{in} to outlet pressure P_{out}. The enthalpy change from H_{in} to H_{out} assumed to be isentropic during compression. The coefficient η=70% is used to take into account non-isentropic expansion for the H_{out} calculation.

In the general case, the mass flow rate through the pump is a function of the pressure head dictated by the given characteristic of the pump. The mass flow rate varies in accordance with SHe enthalpy and pressure in adjacent collectors before and after the pump.

Some simulations were performed for a centrifugal pump using approximation formulae derived from experimental results. A typical operational curve $\dot{m} = f(P)$ of the centrifugal pump is approximated by the formula

$$\left(\frac{m_{op}}{m_{\max}(rpm)}\right)^{1.4} + \left(\frac{\Delta P_{op}}{\Delta P_{\max}(rpm)}\right)^{1.9} = 1,$$

where: m_{op} , ΔP_{op} - operational mass flow rate and pressure head, accordingly; m_{max}, ΔP_{max} - maximal mass flow rate and pressure head for the given rotational speed,: $m_{max}(rpm) = m_{max0}*(rpm/rpm0)$ and $\Delta P_{max}(rpm) = \Delta P_{max0} *(rpm/rpm0)^2$.

A set of another VENECIA models is used to close the described models by linking them to each other. These models describe a variety of insulation elements, perforation etc. New models could be easy added, if necessary.

Coolant properties

Coolant properties are calculated from the unified coolants database. The base consists of a set of tables prepared for each coolant from a known source [3]. Enthalpy *H* and pressure *P* are accepted as independent variables for these tables and form a common 2D mesh for all thermodynamic and thermo-physical properties of the coolant. Special subroutines provide an appropriate approximation accuracy of 0.2% for coolant properties in arbitrary (P,H)-point. A two-phase coolant region is simulated as a homogenous mixture. An equation of state for such mixture has a form

$$1/\rho_{\text{homo}}(P,H) = 1/\rho'(P) + \left[1/\rho''(P) - 1/\rho'(P)\right] \cdot \left[H - H'(P)\right] / \left[H''(P) - H'(P)\right], \quad (14)$$

where ρ', ρ", are the liquid and vapor coolant densities on the boundary line of coolant TS-diagram; *H'*, *H"* are the liquid and vapor enthalpies as a function of pressure *P*.

NUMERICAL SOLUTION

For computations, the analysis equations (1)-(3), (8), (9) were re-written in the non-conservative form using pressure P, enthalpy H and velocity V as flow variables. The Grüneisen parameter ϕ and isentropic speed of sound c were introduced in the equations [2]. Using the basic thermodynamic identity

$$d\rho = \left(\frac{\partial \rho}{\partial P}\right)_H dP + \left(\frac{\partial \rho}{\partial H}\right)_P dH = \frac{1+\phi}{c^2} dP - \frac{\phi \rho}{c^2} dH ,$$

the equations (1)-(3) are transformed to the form

$$\frac{\partial P_i}{\partial t} = -V_i \frac{\partial P_i}{\partial x} - \rho_i c_i^2 \frac{\partial V_i}{\partial x} + \frac{c_i^2 \sum_k \Gamma_{ki}^{\rho}}{A_i} + \frac{\phi_i}{A_i}\left[\begin{array}{l} \sum_m Q_{mi}^{conv} + \sum_k \Gamma_{ki}^{\rho H} + \left(\frac{V_i^2}{2} - H_i\right)\sum_k \Gamma_{ki}^{\rho} + \frac{2 f_i \rho_i V_i^2 |V_i|}{D_{h_i}} A_i \\ -\rho_i V_i F_i(x) A_i - A_i \frac{\partial q_{HeII_i}}{\partial x} \end{array}\right] , \quad (15)$$

$$\frac{\partial H_i}{\partial t} = -V_i \frac{\partial H_i}{\partial x} - c_i^2 \frac{\partial V_i}{\partial x} + \frac{c_i^2 \sum_k \Gamma_{ki}^{\rho}}{\rho_i A_i} + \frac{1+\phi_i}{\rho_i A_i}\left[\begin{array}{l} \sum_m Q_{mi}^{conv} + \sum_k \Gamma_{ki}^{\rho H} + \left(\frac{V_i^2}{2} - H_i\right)\sum_k \Gamma_{ki}^{\rho} + \frac{2 f_i \rho_i V_i^2 |V_i|}{D_{h_i}} A_i \\ -\rho_i V_i F_i(x) A_i - A_i \frac{\partial q_{HeII_i}}{\partial x} \end{array}\right] , \quad (16)$$

$$\frac{\partial V_i}{\partial t} = -V_i \frac{\partial V_i}{\partial x} - \frac{1}{\rho_i}\frac{\partial P_i}{\partial x} - \frac{2 f_i V_i |V_i|}{D_{h_i}} + F_i(x) - \frac{V_i \sum_k \Gamma_{ki}^{\rho}}{\rho_i A_i} , \quad (17)$$

$$t \in [0, \infty), \quad x_{(i)} \in [0, L_i] ,$$

where

$$c^2 = \frac{1}{\left(\frac{\partial \rho}{\partial P}\right)_H + \frac{1}{\rho}\left(\frac{\partial \rho}{\partial H}\right)_P} , \quad (18)$$

$$\phi = -\frac{c^2}{\rho}\left(\frac{\partial \rho}{\partial H}\right)_P . \quad (19)$$

Equations (8) and (9) are re-written as

$$\frac{dP_i}{dt} = \frac{\phi_i\left[\sum_m Q_{im} + \sum_k G_{ki}^{\rho H} - H_i \sum_k G_{ki}^{\rho}\right] + c_i^2 \sum_k G_{ki}^{\rho}}{\Omega_i} , \qquad (19)$$

$$\frac{dH_i}{dt} = \frac{(1+\phi_i)\left[\sum_m Q_{im} + \sum_k G_{ki}^{\rho H} - H_i \sum_k G_{ki}^{\rho}\right] + c_i^2 \sum_k G_{ki}^{\rho}}{\rho_i \Omega_i} \qquad (20)$$

So, the time derivatives for different coolant parameters are separated and assigned in explicit form.

All equations related to *channels* and *conductors* are represented via finite differences with respect to the space variable *x* using the following approximation for the first and second order derivatives in a mesh node "*i*"

$$\frac{\delta Y_i}{\delta x} = \frac{8\cdot(Y_{i+1} - Y_{i-1}) - (Y_{i+2} - Y_{i-2})}{12\cdot h} + o(h^5) , \qquad (21)$$

$$\frac{\delta^2 Y_i}{\delta x^2} = \frac{16\cdot(Y_{i+1} - Y_{i-1}) - 30\cdot Y_i - (Y_{i+2} - Y_{i-2})}{12\cdot h^2} + o(h^4) , \qquad (22)$$

where *h* is a step of uniform spatial discretization. A special attention is paid to approximation of derivatives in the boundary nodes. In **VENECIA,** this approximation is significantly improved and allows provide stable calculation for very fast process at the boundaries (explosion of pipe etc.). As a result of such discretization procedure, the initial system of three partial differential equations for the *P-H-V channel* parameters is transformed into 3*N* ordinary differential equations for the parameters in the mesh nodes with respect to time.

The number of steps (nodes) on the space variable *x* is arbitrary and individual for each *channel*. For the *channels* coupled by heat and mass transfer in the longitudinal direction the number of nodes should be identical.

Integration of the complete system of ordinary differential equations $\frac{d}{dt}\mathbf{Y} = \mathbf{f}(t,\mathbf{Y})$, concerning *P*, *H*, *V* and *T* variables for all nodes of all *channels* and *conductors* (including all equations for *collectors*), is performed by the RK4 method

$$\mathbf{Y}^{t+\tau} = \mathbf{Y}^t + \tau\cdot(\mathbf{k}_1+2\mathbf{k}_2+2\mathbf{k}_3+\mathbf{k}_4)/6 ,$$

$$\mathbf{k}_1 = \mathbf{f}(t, \mathbf{Y}^t) ,$$

$$\mathbf{k}_2 = \mathbf{f}(t+\tau/2, \mathbf{Y}^t+\tau\mathbf{k}_1/2) , \qquad (23)$$

$$\mathbf{k}_3 = \mathbf{f}(t+\tau/2, \mathbf{Y}^t+\tau\mathbf{k}_2/2) ,$$

$$\mathbf{k}_4 = \mathbf{f}(t+\tau, \mathbf{Y}^t+\tau\mathbf{k}_3) ,$$

where τ is the time step for integration.

The numerical method for solving equation (14) is based on a semi-explicit spliting-up method for parabolic partial differential equations [4].

REFERENCES

[1] V.Amoskov, A.Belov, V.Belyakov, O.Filatov, O.Ilyasov, V.Kalinin, M.Kaparkova, V.Kukhtin., N.Shatil, S.Sytchevsky, V.Vasiliev. *Validation of VINCENTA Modeling based on the experiment with the model of the ITER central solenoid*, "Plasma devices and operations", vol. 14, No5, March 2006, 47-59

[2] R.D. McCarty V. Arp and B.A. Hands. Cryopak Software. Cryopak

[3] V.D. Arp, R.D. McCarty, and D.G. Friend, *Thermophysical Properties of Helium-4 from 0.8 to 1500 K with Pressures to 2000 MPa*, NIST Technical Note 1334 (revised), 1998

[4] G.I.Marchuk, *On the theory of the spliting-up method. Numerical solution of partial differential equations* – II. SYSNSPADE-1970, Academic Press, New York – London, 1971

[5] G.Claudet, V.Kalinin, N.Mitchell, P.Roussel, N.Shatil, *Design of the ITER-FEAT cryoplant to achieve stable operation over a wide range of experimental parameters and operation scenarios*, Fusion engineering and Design 58-59, 2001, pp. 205-209

[6] V.Kalinin, R.Haange, N.Shatil, F.Millet, B.Jager et al., *Design and operating features of the ITER 4.5 cryoplant*, Transactions of CEC-03, vol. 49-A, New York, 2004, pp. 176-183.

[7] V.Kalinin, R.Haange, N.Shatil, F.Millet, L.Guillemet et al., *Cryogenic subsystem for nominal operation and fast regeneration of the ITER primary cryosorbtion vacuum pumps*, Transactions of CEC-03, vol. 49-A, New York, 2004, pp. 184-191.

[8] V.Kalinin, E.Tada, F.Millet, N.Shatil, *ITER cryogenic system*, Fusion engineering and Design, 2006, pp. 1-7

[9] D.Bessette, N.Shatil and E.Zapretilina, *Simulation of the ITER Toroidal Field Coil Operation with the VINCENTA Code*, IEEE Transaction on Applied Superconductivity, vol. 16, No2, June 2006, pp. 795-798

[10] B.Rousset, P.Roussel, F.Michel, D.Hitz, Introduction of a saturated bath in VINCENTA models: Application to the cryogenic system for JT-60SA tokamak, presented on CEC conference, 2009

ALPHYSICA

ALPHYSICA GmbH (headquarters&technical support)
Germany, 76135,
Karlsruhe, Unterreut, 6
Phone: + 49 (0163) 904 8561
Fax: +49 (07219) 444 26 55
europe@alphysica.com

ALPHYSICA Inc. (technical support)
USA, 94111, San Francisco,
414, Jackson street, suite #204
Phone/Fax: + 1 415 230 23 63
usa@alphysica.com

ALPHYSICA Ltd. (technical support)
Russia, 191025, St. Petersburg,
ul. Mayakovskogo, 3, office #64
Phone/Fax: +7 812 335 95 04
russia@alphysica.com