INDUSTRIAL BURNER PREMIXED COMBUSTION SIMULATION.
ENEA TEST CASE: HOT.

1. ABSTRACT

Every industrial combustion simulation model has to be tested against standard experimental data. This work presents a comparison between calculated and experimental data of a lean premixed combustion flow with the aim of validating the Turbulent Flame Closure combustion model. The experimental results, relative to the fluid-mechanics and chemical characterization of a Dry Low NOx natural gas premixed burner working at atmospheric pressure designed by Nuovo Pignone, were carried out by ENEA. The study regards also thermal and chemical in-flame measures, to have a complete knowledge of the combustion phenomenon.

The simulation has been performed by using `TANIT`, a FORTRAN code suitable for simulating combustion flows for gas turbines applications.

2. INTRODUCTION

The growing demand to reduce pollutant emissions coming out from combustors induced the majority of makers of gas turbines, to explore new technological solutions. The italian Nuovo Pignone, for low pressure and low-power gas turbines, is testing Dry Low NOx burners which, working in lean premixed combustion regime, can guarantee very low NOx emissions.

The experimental test [1] has been directed by feeding the burner with natural gas supplied by the distribution net of the ENEA-Casaccia Center. In Table 1 is presented the mean composition of the natural gas in volumetric percentage.

This experimental test case has been simulated with the `TANIT` code to validate the TFC (Turbulent Flame Closure) premixed combustion model.

Previous works have been carried out to validate the `TANIT` code [4] using the results of cold tests performed with the same combustor. The aim was to validate the code concerning the fluidodynamic point of view, apart from the chemistry. The results were in good agreement with the experimental data.

The present work is a new step of the same study: now the aim is to validate the `TANIT` code for hot tests.

Helium (He)	0.05%
Hydrogen (H2)	-
Nitrogen (N2)	2.5%
Methane (CH4)	9.5%
Carbon Monoxide (CO)	-
Carbon Dioxide (CO2)	0.1%
Ethane (C2H6)	2%
Propane (C3H8)	1%
Butane (C4H10)	0.1%
Higher Hydrocarbon	0.75%

Table 1: Natural Gas Composition

3. THE MATHEMATICAL MODEL

`TANIT` is a parallel industrial combustion code developed in the ESPRIT "TANIT" project. In the implicit solver of this code can be implemented two different combustion models :

- presumed PDF approach with two-scalars model for non-premixed combustion;
- the TFC model for premixed combustion.

In this study we have considered the TFC model because we are working in regime of premixed combustion. This model [2] have been implemented starting from the following equation for the local progress variables $\tilde{c}$ ($\tilde{c} = 1$ 100% reactants; $\tilde{c} = 0$ 100% products):

$$\frac{\partial \overline{\rho}\tilde{c}}{\partial t} + \nabla(\overline{\rho}\tilde{c}\tilde{u}) = \nabla\left[\left(\frac{\mu}{Pr} + \frac{\mu_t}{Pr_t}\right)\nabla\tilde{c}\right] + \rho_u U_t |\nabla\tilde{c}| G$$

where the suffix u refers to the unburned mixture, the suffix t refers to turbulent conditions, U_t is the turbulent combustion velocity and G is the stretch-factor. Furthermore, $U_t = U_t(k, \epsilon, \eta_u, U_l)$, where η is the thermodiffusivity and U_l the laminar burning velocity. Pr is the Prandlt number.

Chemistry is accounted via U_l that is calculated using the `RUN-1DL` code [3] developed at the Imperial College of London. This code models and numerically predicts laminar reacting-flow phenomena. It has been developed for the numerical simulation of steady or unsteady, laminar one-dimensional and quasi one dimensional, chemically reacting flows (strained, unstrained, premixed, partially premixed, tubular, linearly, cylindrically, spherically symmetrical flames). In this study `RUN-1DL` has been used to simulate a freely propagating, unstrained premixed methane-air flame. The composition of the mixture is that corresponding to $\lambda = 1.7$ ($X_{CH4} = 0.0566$, $X_{O2} = 0.1978$, $X_{N2} = 0.7456$), and has been used a detailed chemistry mechanism. `RUN-1DL` solves the balance equations by using the Newtons' method, starting from an instationary situation and tending to the steady state.

The input parameters of the TFC model are Pr_t, ρ_u, ρ_b, η_u, U_l and the critical velocity gradient g_{rc}. The suffix b refers to the burned mixture. The quantities Pr_t, ρ_u, ρ_b, η_u, U_l and T_b (temperature of the burned mixture) are all calculated by using the `RUN-1DL` code. The turbulent transport is modelled via the Prandlt-Boussinesq hypothesis plus the $k - \epsilon$ [5] model for the eddy viscosity. The density and temperature are evaluated according to the expressions:

$$\overline{\rho} = \left[\frac{(1-\tilde{c})}{\rho_u} + \frac{\tilde{c}}{\rho_b} \right]^{-1}$$

$$\overline{T} = (1-\tilde{c})\, T_u + \tilde{c} T_b$$

where T_u is the temperature of the unburned mixture, ρ_b and T_b are the density and the temperature of the burned mixture considered in equilibrium.

4. THE EXPERIMENTAL TEST CASE

An outline of the burner used for the experimental activity is presented in Fig.1. The model is named K-120 (120 mm is the diameter of the combustion chamber).

Downstream the inlet of air into the distributor there are 12 injectors, each of them has 6 holes ($\phi = 1$ mm). Downstream these injectors it can be placed a group of 12 plane spades necessary to transmit a swirl movement to the gas stream. This test case is without swirl.

Running through the premixing zone, between the injection point and the inlet into the combustion chamber, the mixture speeds up from $v = 12.7 m/s$ to $v = 47.7 m/s$, because the section narrows ($section - ratio = 3.75$). This condition is necessary to have a good safety limit against flash-back.

On the burner head, after the inlet in the combustion chamber, there are 12 injectors ($\phi = 0.9$ mm) to produce a pilot flame which stabilize the main flame. From these injectors can flow at most a 15% of the total gas mass.

A summary of the burner input is reported in Table 2:

To collect the experimental data probes have been located in three sections starting from the horizontal part of the flame-tube, which are shown in Fig.2:

main-fuel injection	2.42 g/s
pilot-fuel injection	0.18 g/s
combustion-air	76.3 g/s
Tair	293 K
Tgas	293 K
P	1 atm

Table 2: Burner Input

- $first - section$ = 5 mm (23.5 mm real)
- $second - section$ = 80 mm (98.5 mm real)
- $third - section$ = 160 mm (178.5 mm real)

5. NUMERICAL SIMULATION

The geometry of the burner is shown in Fig.2. Using the `TANIT` code has been simulated an axisymmetric flame-tube with a diameter of 120 mm. The flame-tube computational grid has $192 * 40 * 1$ cells. The inlet mixture velocity profile has been assumed as uniform (v = 47.7 m/sec). The input parameters have been calculated by using the `RUN-1DL` code and are reported in Table 3.

T_u	293 K
T_b	1500 K
U_l	6.3 cm/sec
ρ_b	0.223 kg/m^3
ρ_u	1.1546 kg/m^3
k	19 m^2/s^2
ϵ	1.25E4 m^2/s^2

Table 3: Input Parameters

The inlet mixture kinetic turbulent energy has been calculated by using the expression [5]:

$$k = 3/2(U_{ref}T_i)^2$$

where T_i is the turbulent intensity and has been taken equal to 0.075.

To calculate the k dissipation has been used the expression [5]:

$$\epsilon = C_\mu^{3/4} \frac{k^{3/2}}{l}$$

where $l = 0.07L$, L is the inlet diameter taken as characteristic width and $C_\mu = 0.09$.

The ignition of the mixture is obtained by assigning $\tilde{c} = 0$ at the inlet flow (unburned mixture) and $\tilde{c} = 1$ as initial condition everywhere in the internal flow field. The air excess index λ has been taken equal to 1.7 as in the experimental test.

Uniform inlet flows have been assumed as boundary condition. All around the inlet and on the external cylindrical boundary have been assumed adiabatic wall conditions. On the axis have been assumed singular line conditions, and on the remaining two planes periodic rotation conditions.

The CFL number used for time step computation is 10. The spatial discretization scheme adopted for the convective terms computation is a first order upwind. The selected solution method for the system of linear equation is BICGSTAB with ILU preconditioner.

The pilot flame has not been taken into account in this work.

6. RESULTS

Figures 3, 4, 5 show the comparison between the experimental and the calculated mean axial velocity. Each figure refers to a different section of the burner, corresponding to the Nuovo Pignone probes locations. In every section the experimental mean velocity near the axis is higher than the one calculated with the `TANIT` code. These differences become smaller starting from around $x = 0.02$ m from the axis.

Figures 6, 7, 8 show the comparison between the experimental and the calculated fluctuations of the turbulent energy at three different sections of the burner. The two curves have quite the same behaviour near the axis and then become different.

Figures.9, 10, 11 compare the experimental and the calculated mean temperature at three different sections of the burner. The results show noticeable quantitative and qualitative disagreements: temperatures are overestimated and locations of the steep temperature gradients are incorrect.

7. CONCLUSIONS

As shown in the previous section the results obtained from the simulation performed with the `TANIT` code are not in good agreement with the experimental data. In particular the differences are very significant for the temperature's curves. We can explain this disagreement pointing out the existing differences in the boundary conditions.

`TANIT` simulation:

- uniform inlet mixture profiles;
- absence of wall cooling;
- absence of pilot flame;

experimental test:

- real inlet profiles;
- presence of wall cooling;
- presence of pilot flame.

The different temperature near the axis may be explained by taking into account the presence of a restrict recirculation zone under the inlet. This cause the combustion in that zone and so hight temperatures. This zone is so restrict that the grid used is not able to take it into account.

Looking at Figs.9, 10, 11 and moving from the axis towards the combustor wall we notice that, in each of the probing sections, the calculated temperature's curves raises earlier and reach higher values than the experimental ones. We can suppose this phenomenon due to the non uniform inlet flow composition: using the code we have assumed a uniform concentration profile, because the TFC model can only deal with homogeneous fuel mixture. The real methane mass fraction profile is reported in Fig.12. We can notice that near the axis the methane mass fraction is higher then far from it. Using inlet constant concentrations may generate richer mixture than in the real situation and this could make combustion take place nearer the axis and lead higher temperatures.

To confirm these hypotheses is necessary to do further work. A new step of this study would be to simulate the same tests by using a lower inlet uniform methane mass fraction (lean mixture). We would expect that the new temperature's curves would be similar to the experimental ones: a postponed slope and a lower maximum temperature.

Furthermore, we can observe that the absence of the wall cooling in the simulation is pointed out by the absence of temperature decrease far from the axis.

References

[1] S.Giammartini, G.Sidoti, "Approntamento della campagna sperimentale su di un bruciatore a fiamma premiscelata per turbine a gas", lavoro svolto nell'ambito dell'accordo di programma Enea-Mica, rif.Obiettivo Intermedio 1.4.1.1M, ERG EIDE 96062.

[2] V.L. Zimont, F.Biagioli, C. Bruno and A.N. Lipatnikov," Test of an Engineering Model of Premixed Turbulent Combustion", 4th European Conference on Industrial Furnaces and Boilers, Espinho-Porto, 1-4 April, 1997.

[3] B. Rogg, "`RUN-1DL`: The universal laminar flame and flamelet code, Lehrstuhl fur Stromungsmechanik, Institute fur Thermo und Fluiddynamik, Ruhr-Universitat Bochum, D-44780, Germany.

[4] W. Bellucci*, V. Zimont*, C. Bruno**, V. Risalvato* and C. Ceccherini***, "Industrial burners premixed combustion simulation". * CRS4 Research Centre, Cagliari, Italy ** Roma University, Roma, Italy *** Nuovo Pignone, Firenze, Italy

[5] H.K.Versteeg, W Malalasekera, "An introduction to computational fluid dynamics", The finite volume method, Longman 1995.

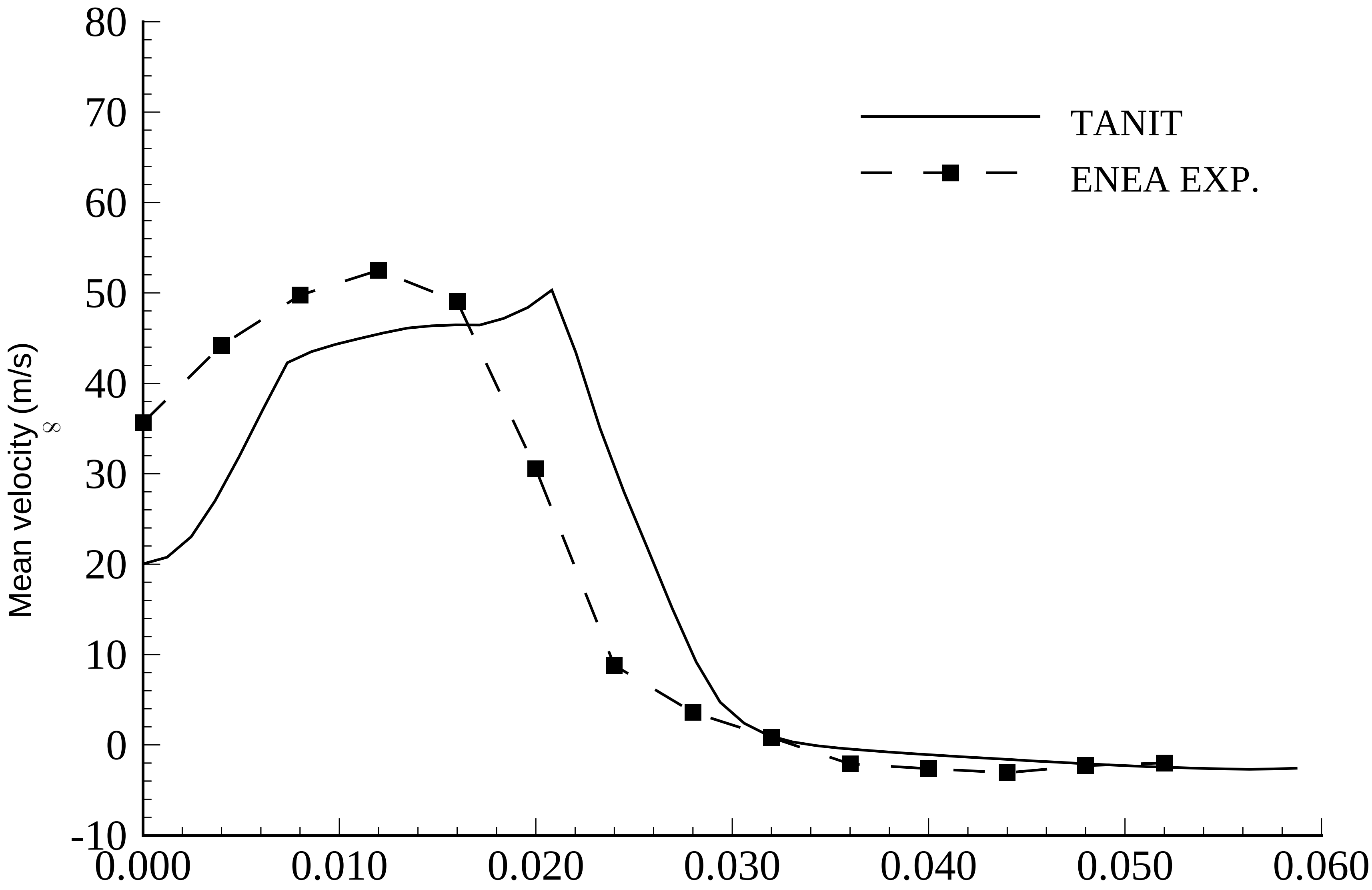

Figure 3: distance from the burner axis(m)   (x=5mm)

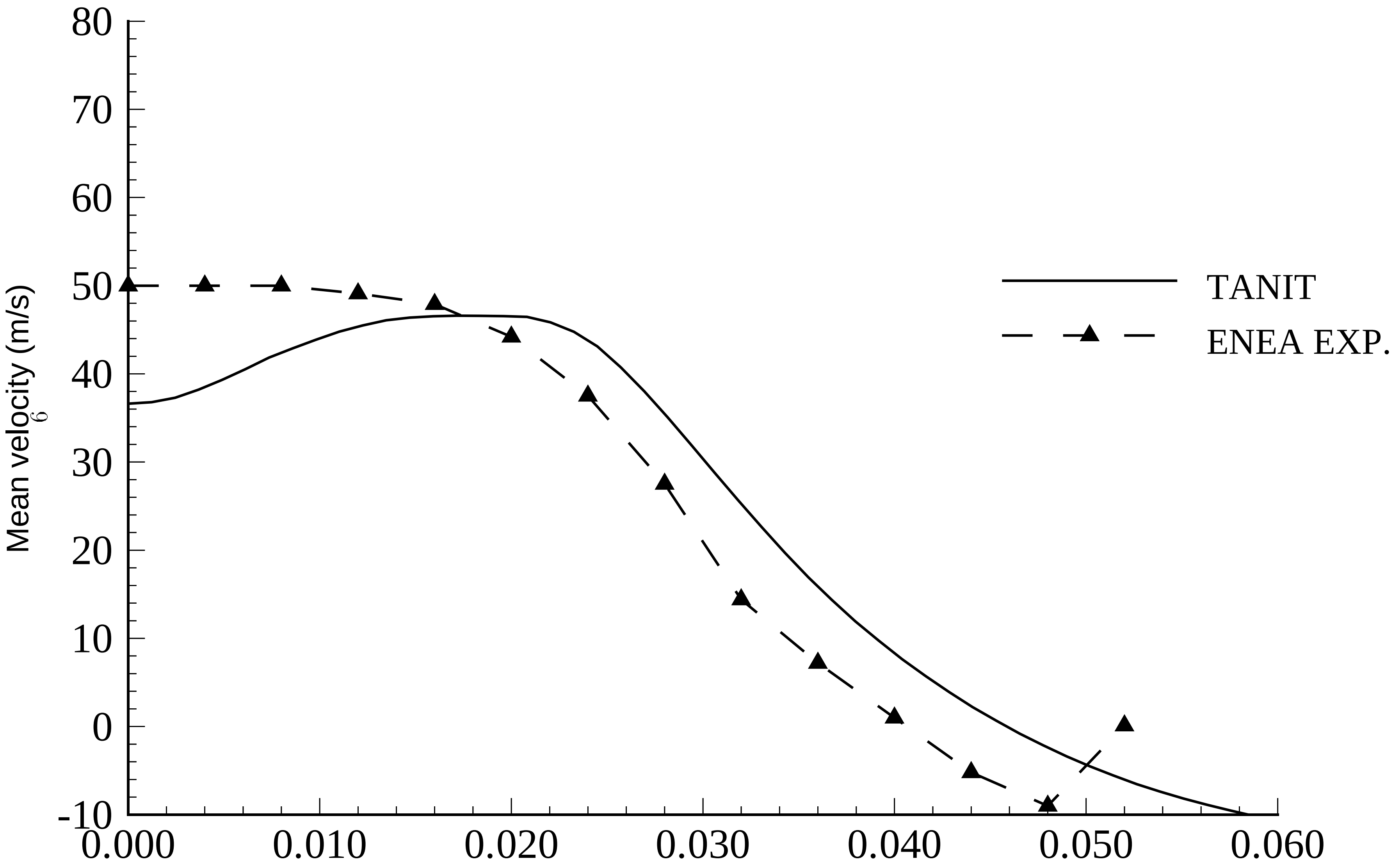

Figure 4: distance from the burner axis(m) (x=80mm)

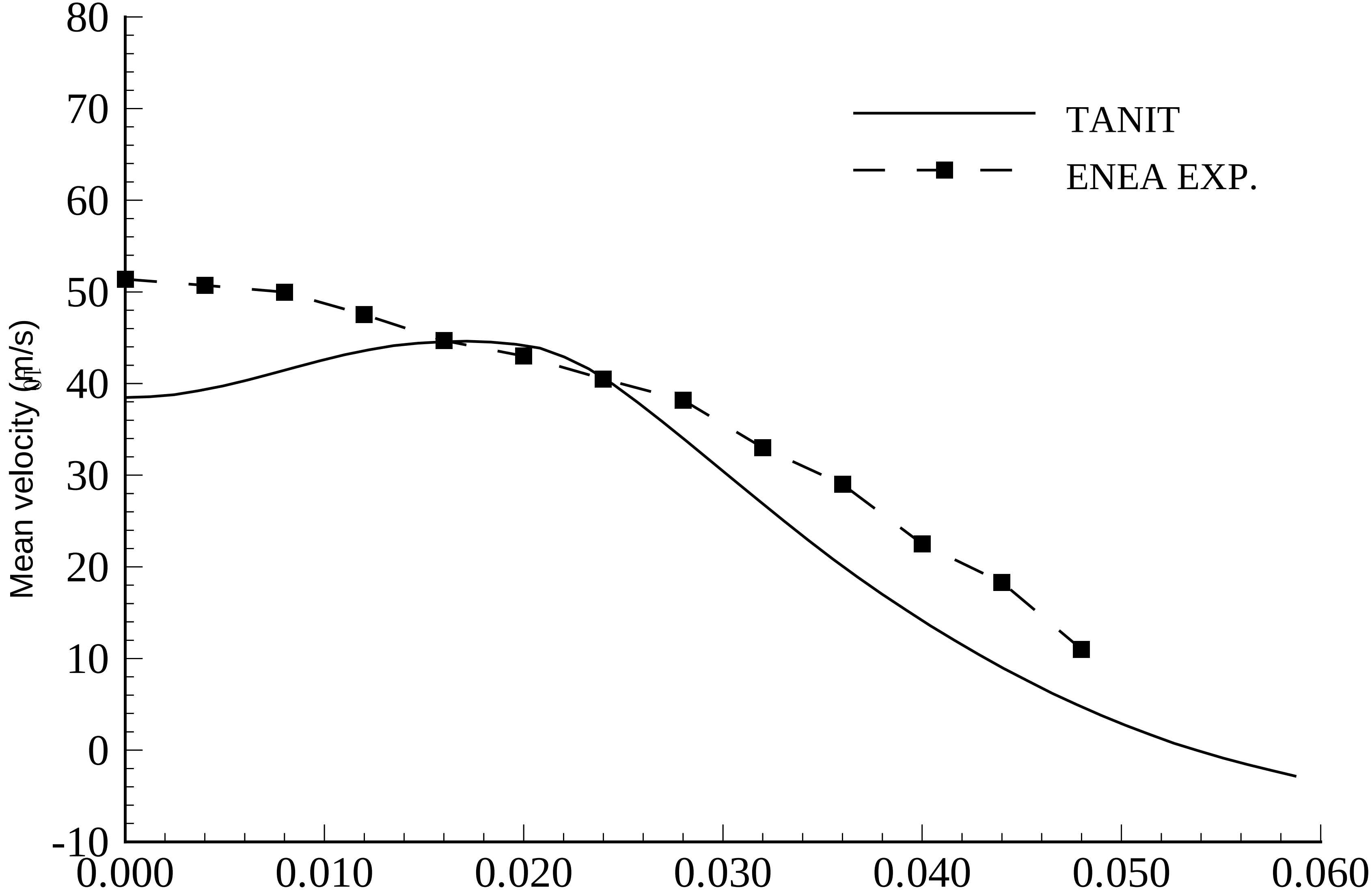

Figure 5: distance from the burnen axis(m)   (x=160mm)

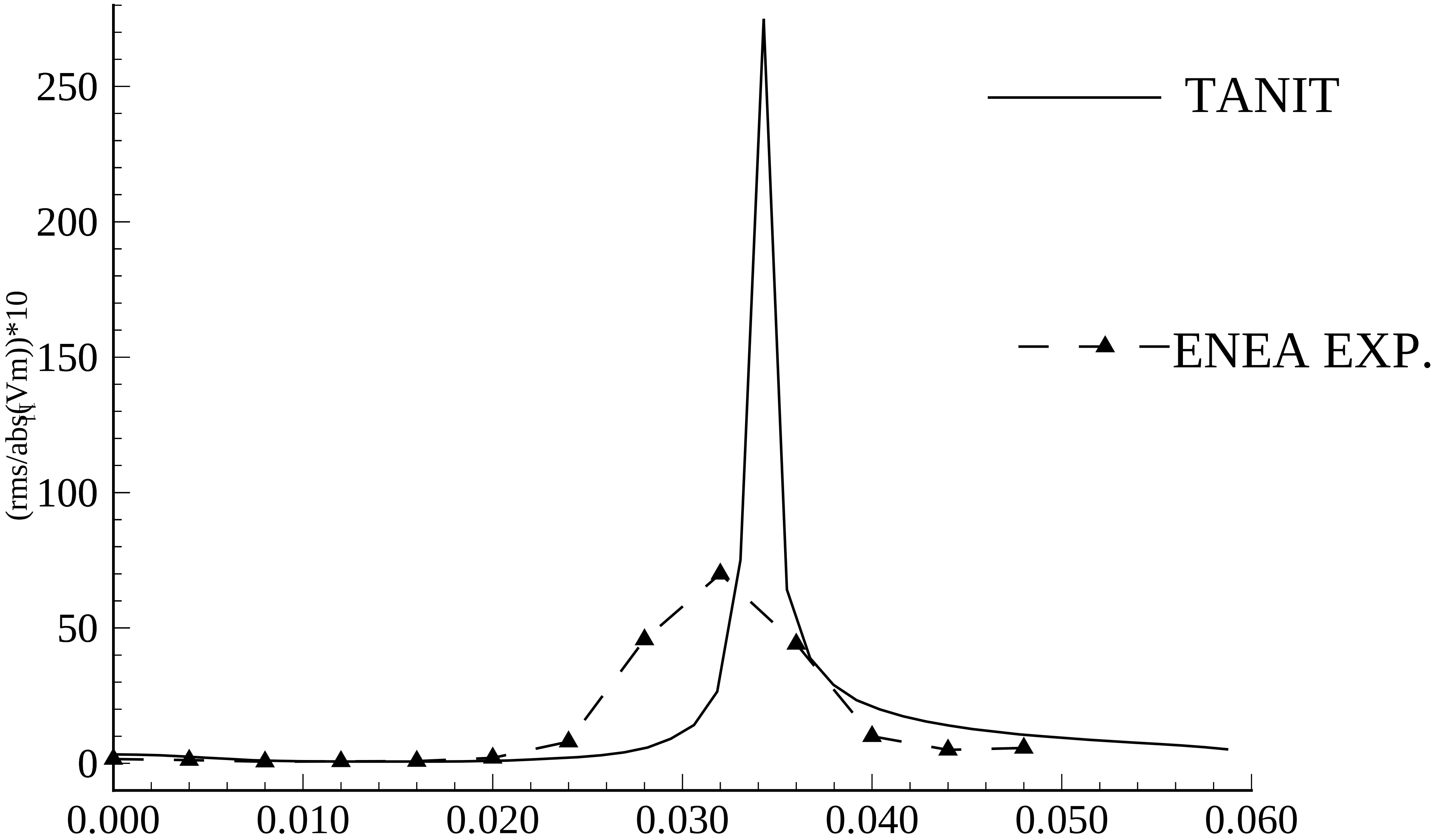

Figure 6: distance from the burner axis(m)   (x=5mm)

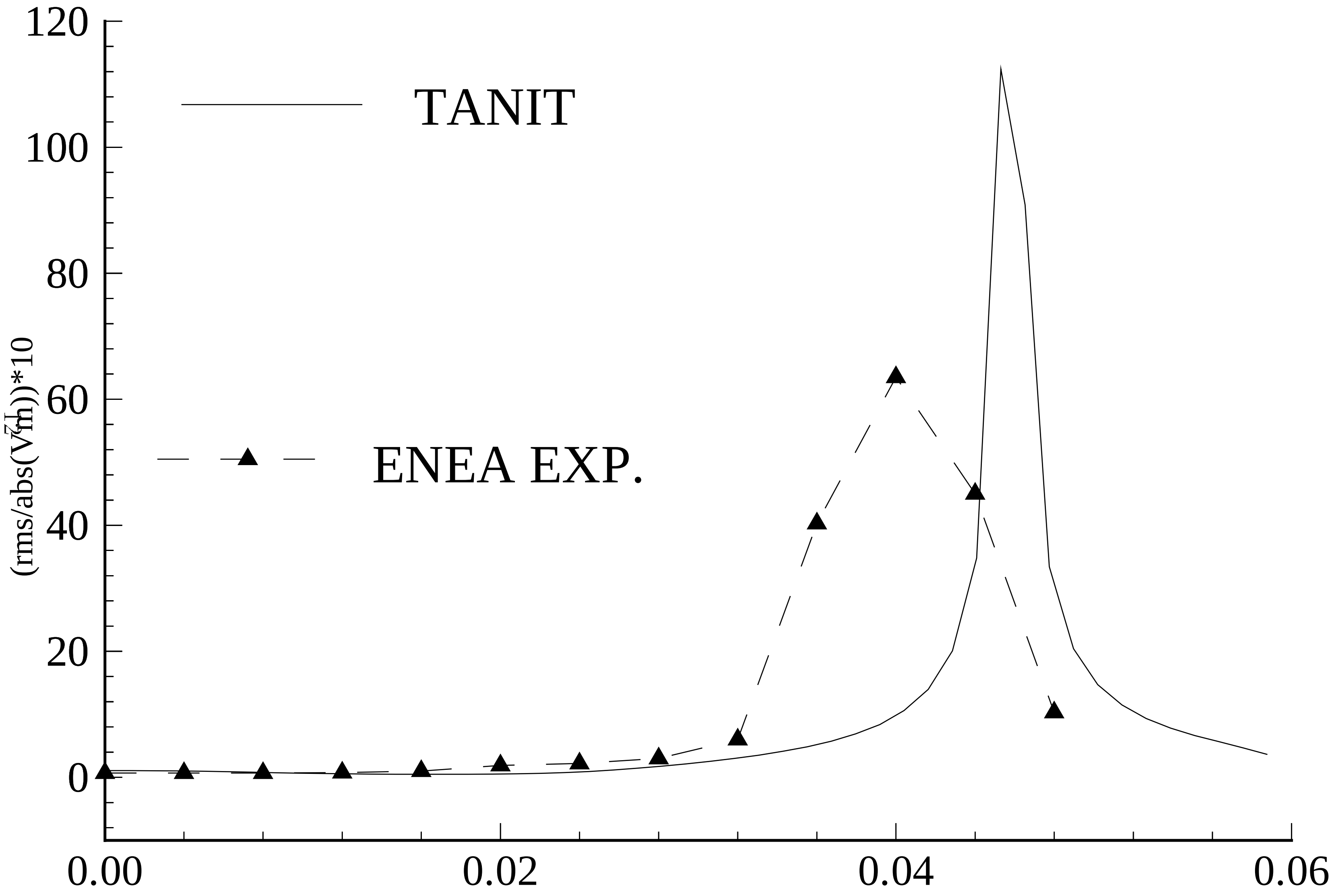

Figure 7: distance from the burner axis(m) (x=80mm)

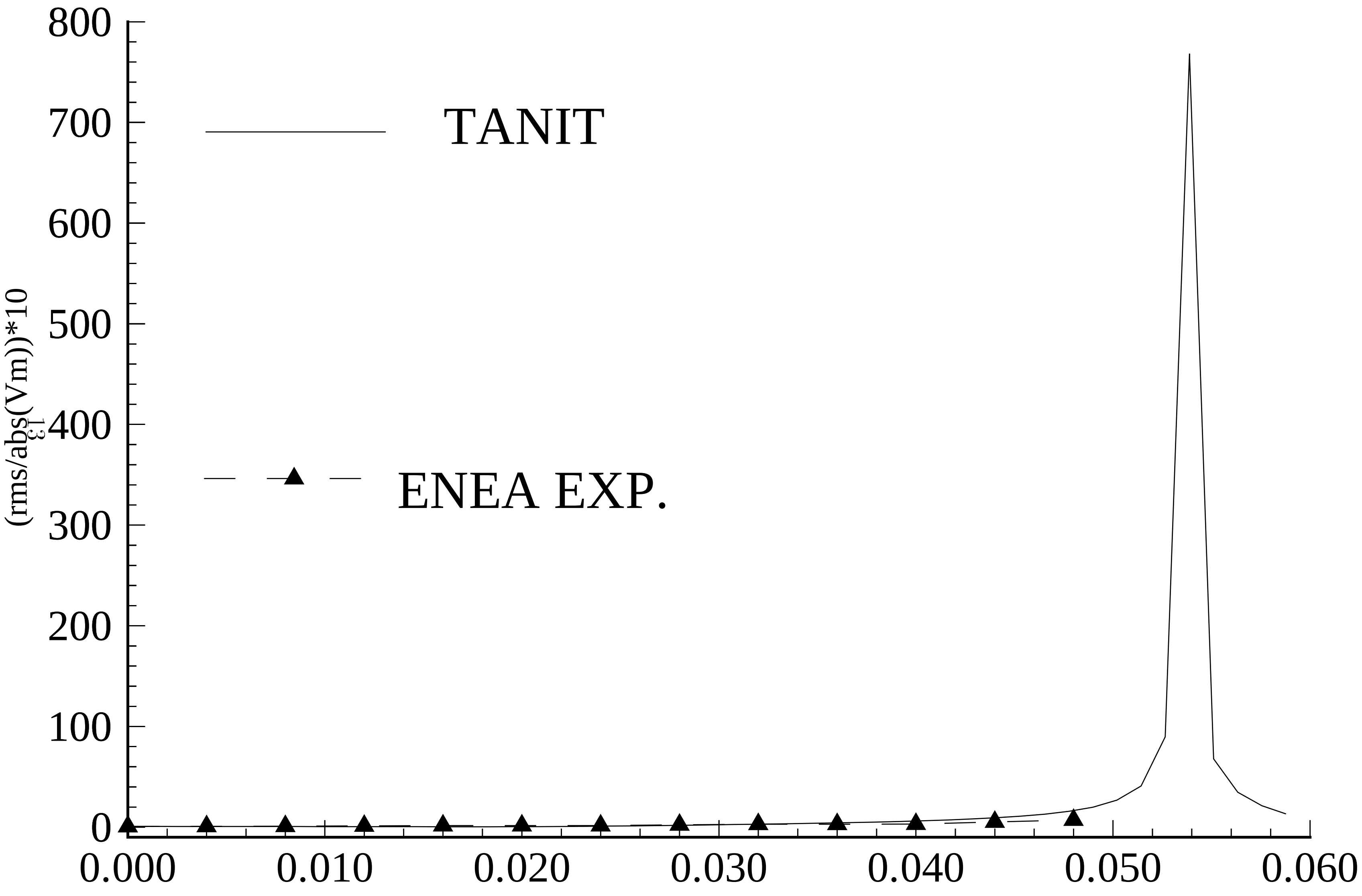

Figure 8: distance from the burner axis(m) (x=160)

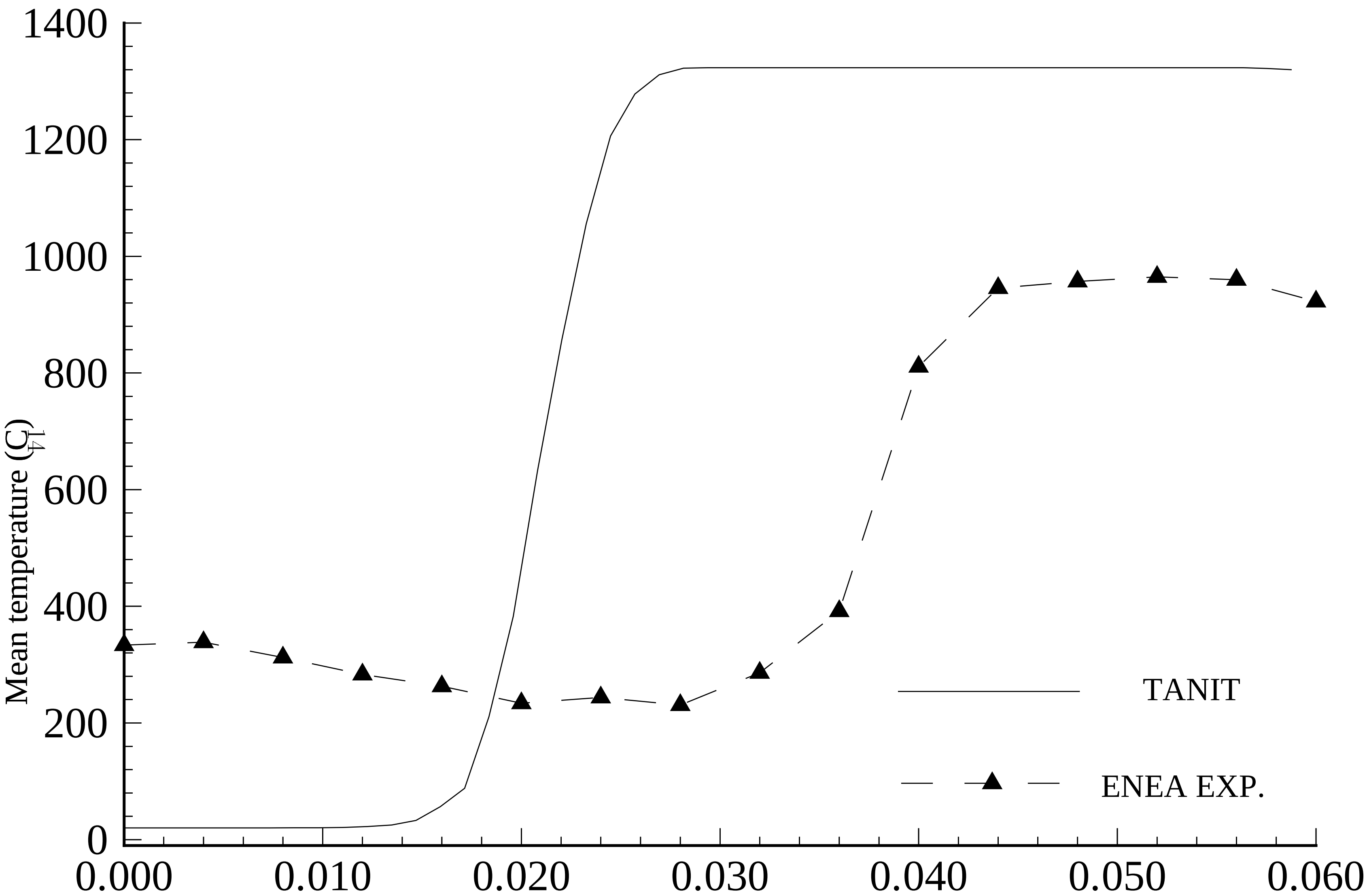

Figure 9: distance from the burner axis(m) (x=5mm)

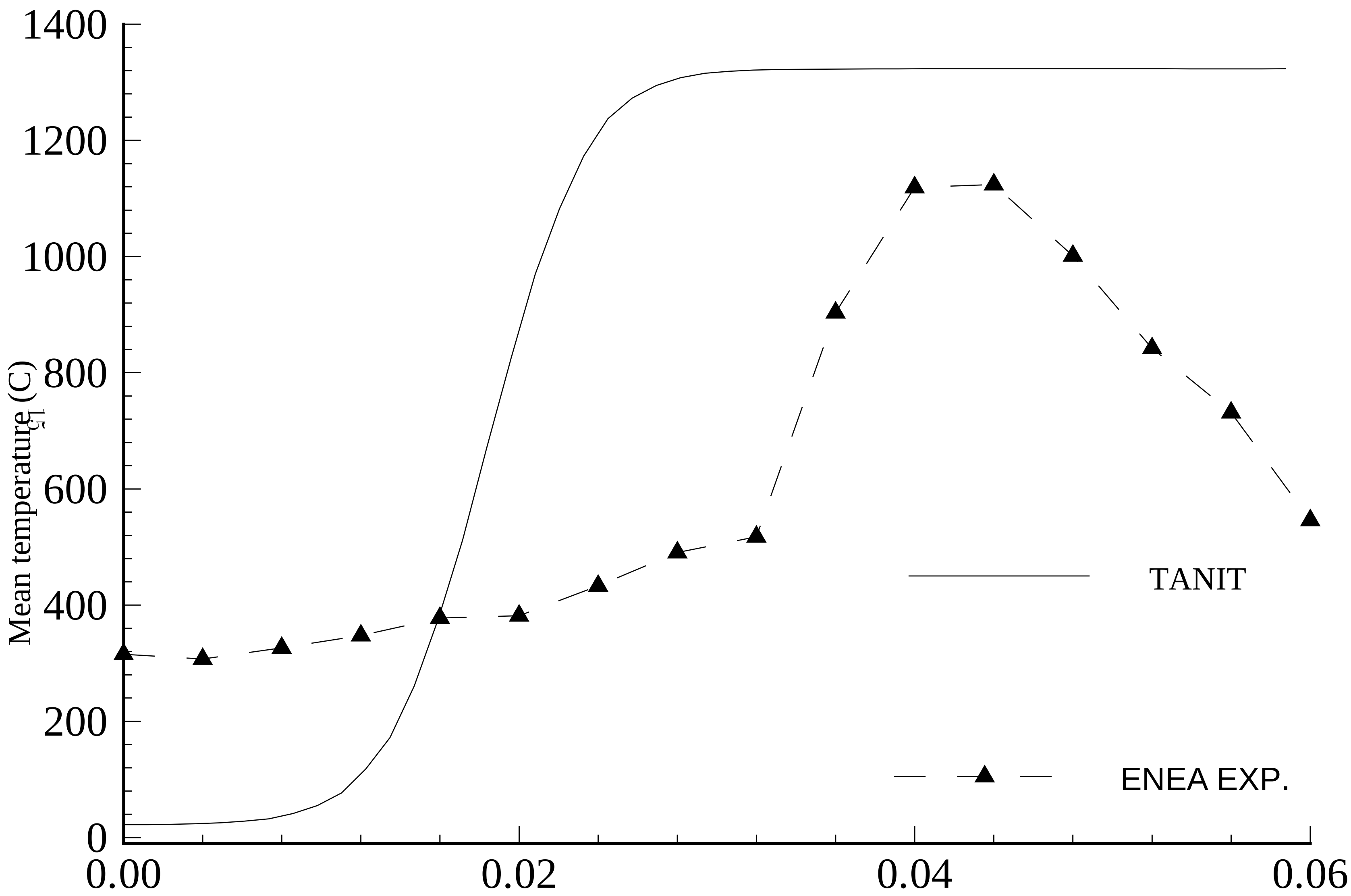

Figure 10: distance from the burner axis(m)   (x=80mm)

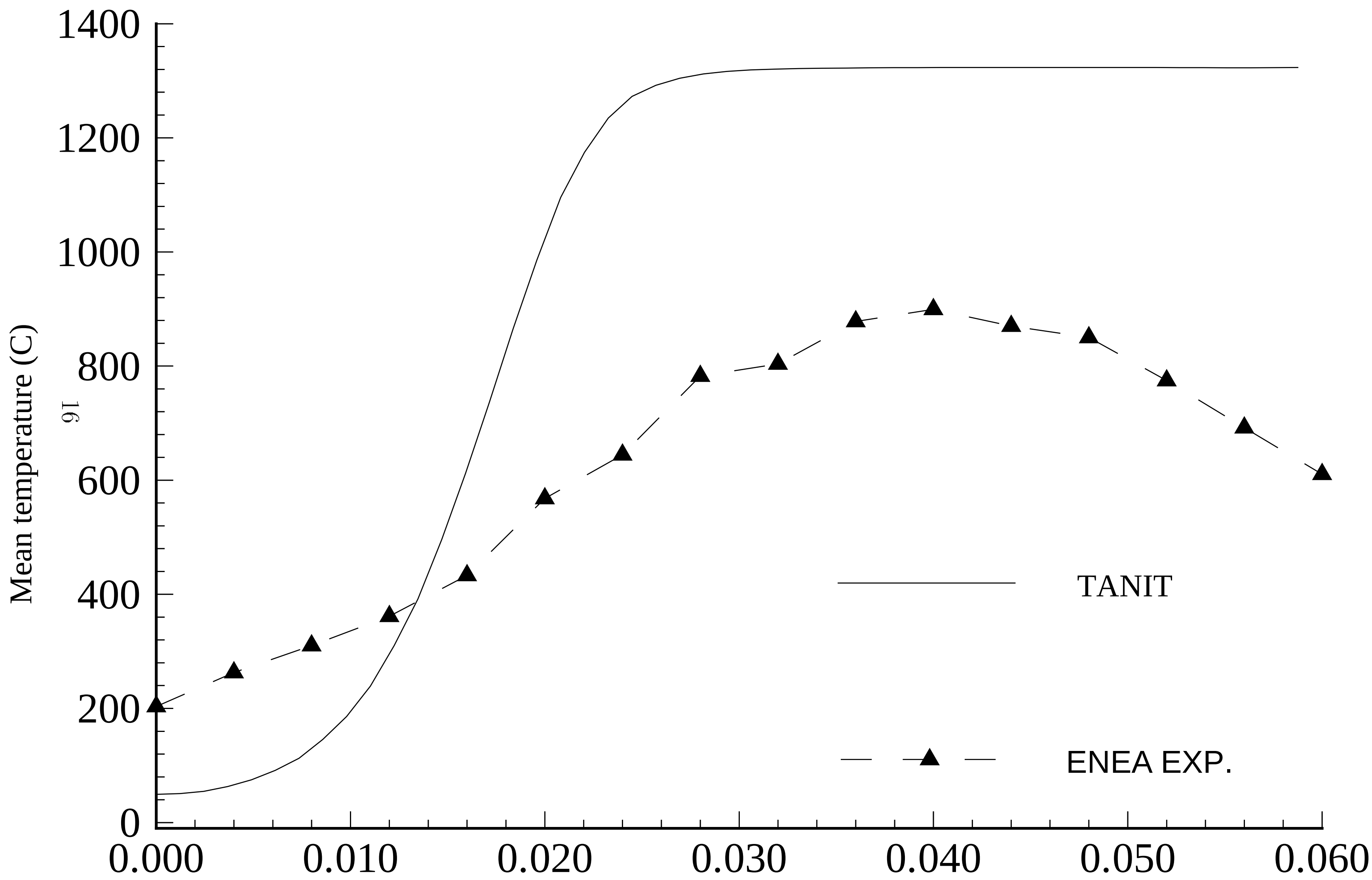

Figure 11: distance from the burner axis(m) (x=160mm)

(2D) || Print || 16 Jun 1998 || meta plt ||

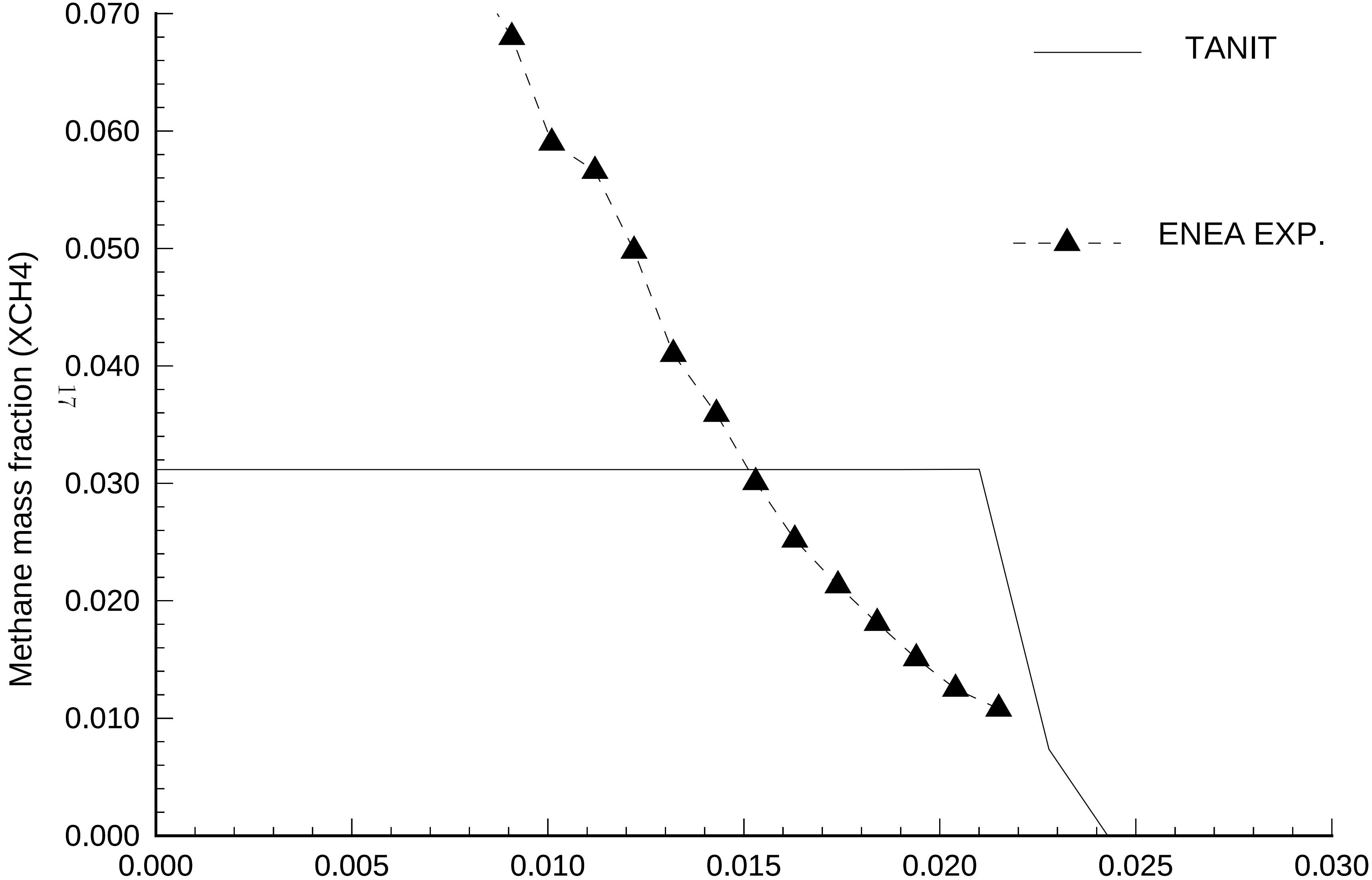

Figure 12: distance from the burner axis(m) (x=0.0m)